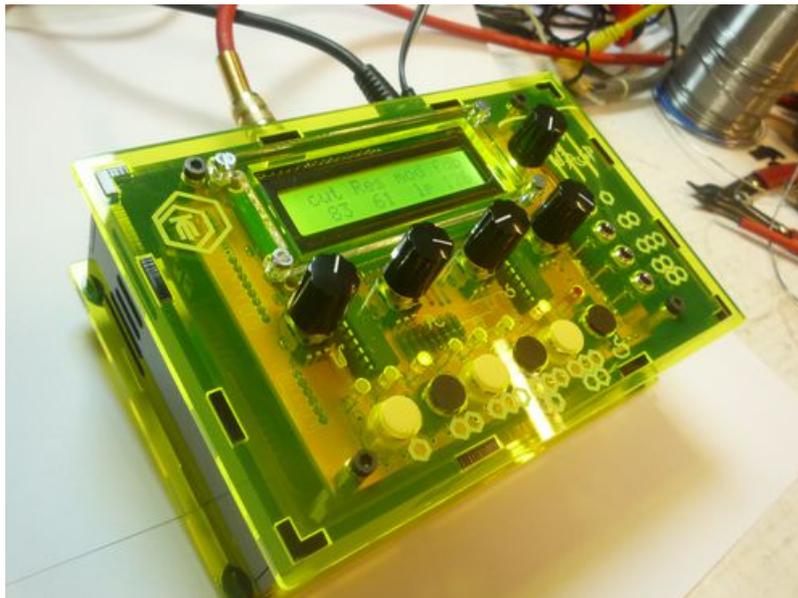


# TubeOhm Phoenix/Shruthi CD4069 Filter edition



Phoenix-W motherboard  
DIY manual english V 0.1  
Datum 19.06.2019



©TubeOhm 2019

Introduction:

Important!

The motherboard works with all available filterboards. However, depending on the filter board, the software of the motherboard must be set to the respective filter board! In the case of the CD4069 filter, the motherboard must be set to 'SHRUTHACON' and saved.

Phoenix/Shruthi now also with a filter which is designed according to the topology of the 1980 EDP WASP synthesizer with the CD 4069. That was really missing.

The special thing about the filter is the C-MOS chip used. CD4069 in the signal path. Normally used as a logic device, this IC is now used as an audio amplifier.

Typical operational amplifiers should be linear. The CD4069 is anything but linear, and this gives it its special sound.

Deep bass, or normal to highly distorted sounds, anything is possible.

And so that the Phoenix is also nicely harmonious there is the black/yellow coloring of the buttons, the cabinet and the circuit boards.

### **Analog inputs:**

Since there have been many requests for the analog inputs of the motherboard - Shruthi/Phoenix - I have developed a circuit which makes the inputs usable and protects them from negative and overvoltage.

Now the Phoenix/Shruthi can be controlled with input voltages in the range of 0..10 Volt via the modulation matrix.

The modulation matrix offers CV1..4 as an external source which can be routed to different destinations as usual.

Overvoltage and negative voltages are blocked by the protective diodes and the Rail to Rail amplifier MCP 6004. Thus the CPU is protected.

Attention, even if the inputs are protected the maximum input voltage should be kept. So 11...12 is no problem - but please not 30 Volt!

The inputs are an option but no MUST ! If no inputs are needed, the components /diodes/resistors can simply be omitted.

Furthermore, the analog inputs are sampled very slowly.

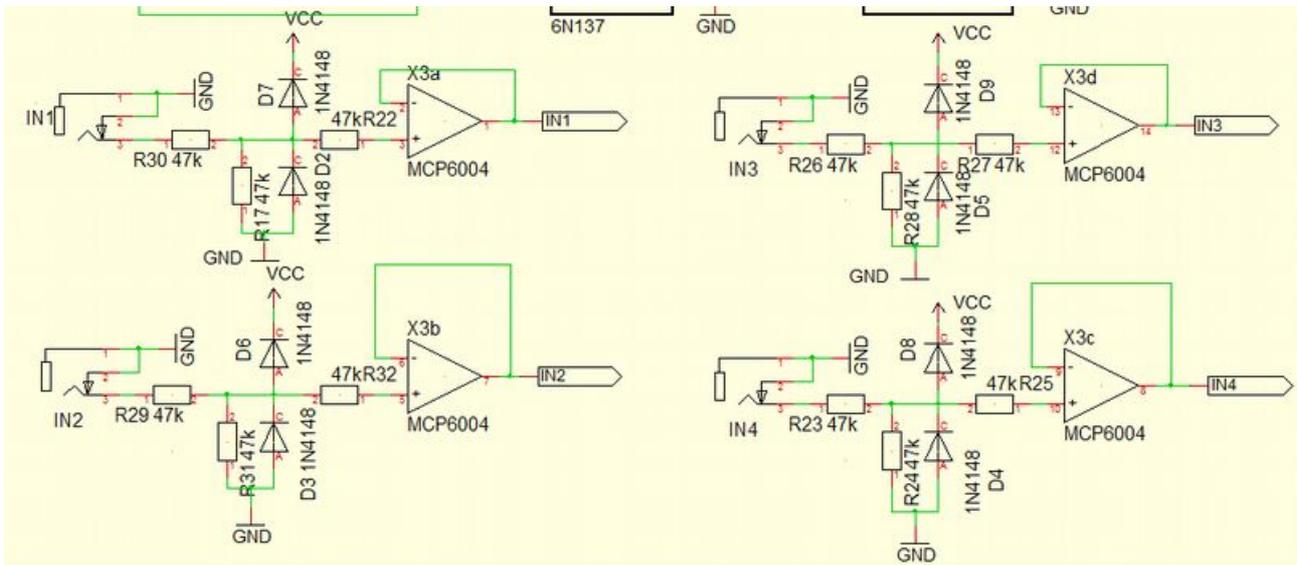
The sampling rate is approx. 10..20 Hz depending on the system.

Possibly there is a programmer from the DIY Scene who can increase the sampling rate a little ??!

### Here is the circuit diagram for the input voltage

The diodes D2 limit the negative voltage to 0.7 Volt.

If the input voltage is greater than 10 volts, the current flows via 47K and D7 to the 5V operating voltage. The Rail to Rail Opamp MCP6004 also offers additional protection against overvoltages at the CPU. It is operated with 0 and 5Volt. So it cannot output a voltage higher than 5 volts/0 volts.



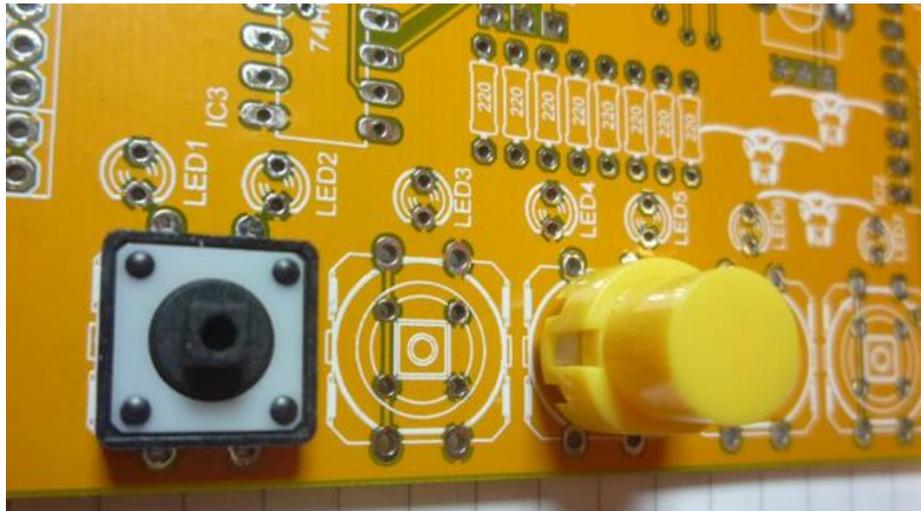
Circuit diagram of the four analog inputs

### Further changes:

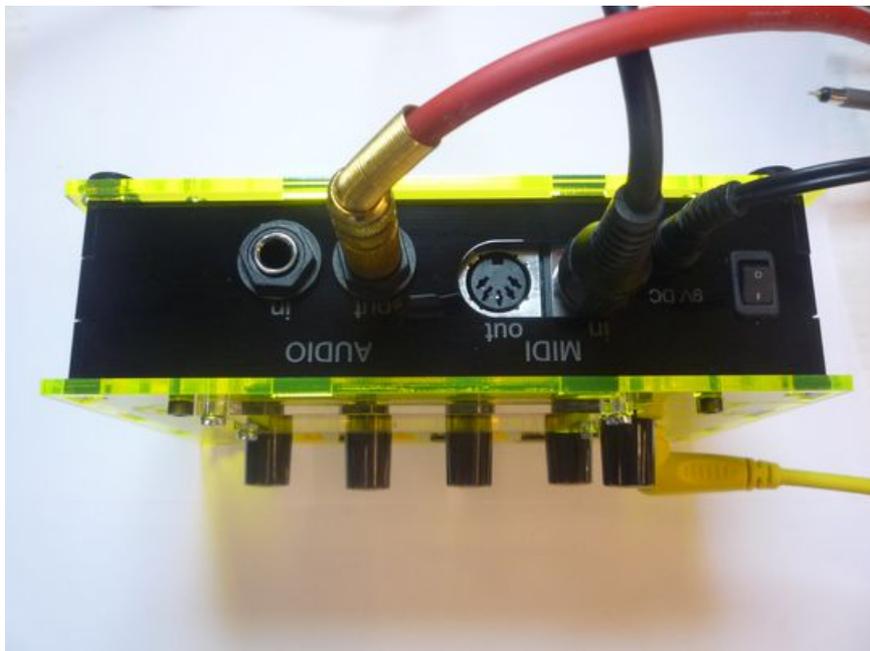
To ensure maximum compatibility to the ur-Shruthi, it is now possible to choose between 2 different buttons.

Normal pushbuttons with caps but also new round pushbuttons can be used. The layout is changed so that both versions fit.





The housing has also been completely redesigned and now has an on/off switch.



So that after half a year the display is still easy to read (and not completely dusted and with grease fingers), there is now also a display protection disk.

## **To the DIY:**

What do I need to build a Phoenix/Shruthi ?

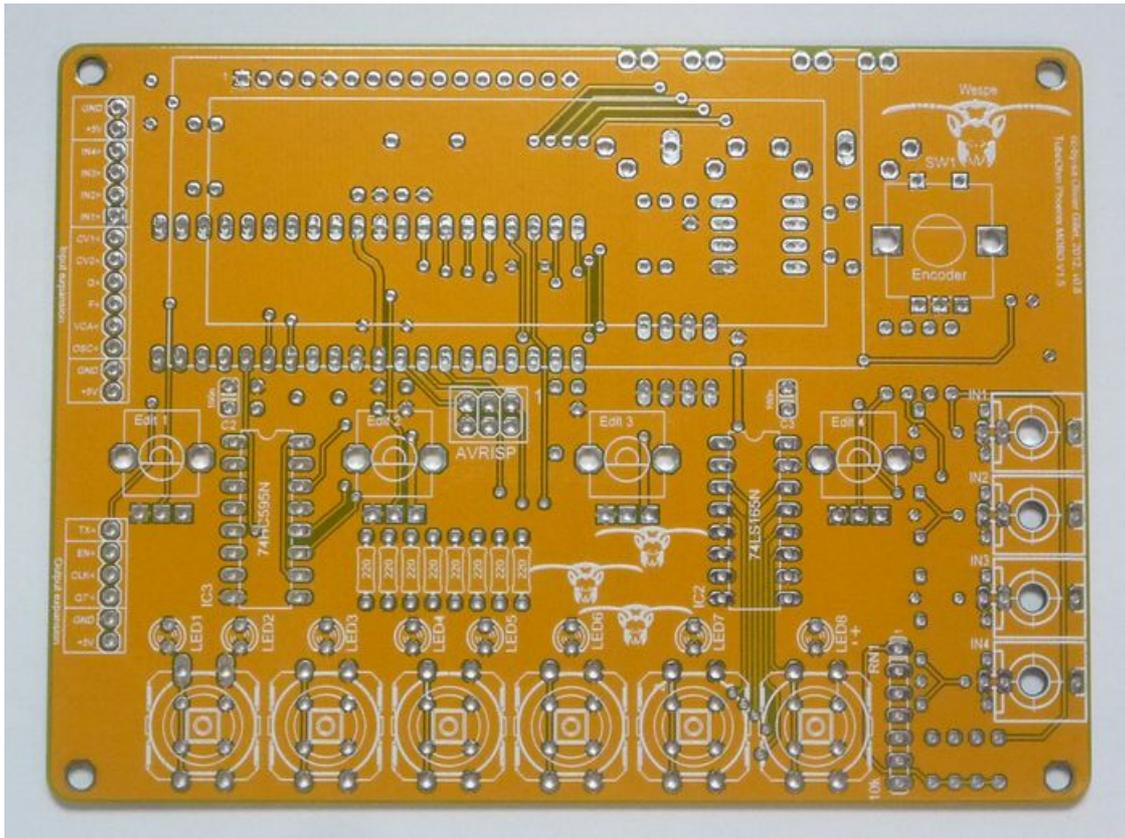
- 1:) First of all some know how in electronics and electrical components
- 2:) a good side cutter and screwdriver
- 3:) a soldering iron and good soldering tin
- 4:) a multimeter would be desirable and a good idea of how to use it
- 5:) one DC power supply 9Volt 600 mA, plug 5,5 mm /2,1 mm, center positive  
The power supply is not included in the kit!
- 6:) Midi keyboard , and an amplifier

All electronic parts as well as the pre-programmed Atmega CPU are available in the DIY Kit.

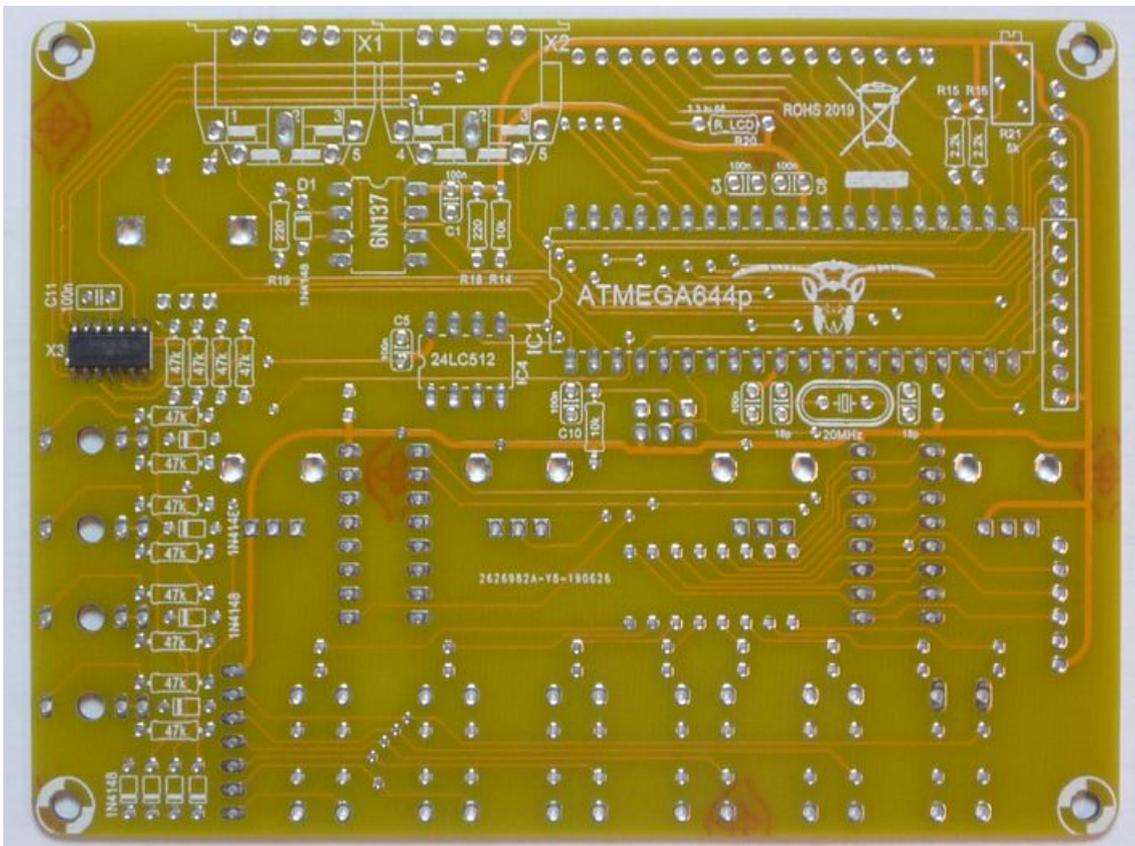
## **OK now let's switch on the soldering iron and get started.**

The motherboard is assembled first from below, all components are soldered. When everything is finished at the bottom, the board is assembled from the top and the components are soldered. The LCD display is soldered at the end.

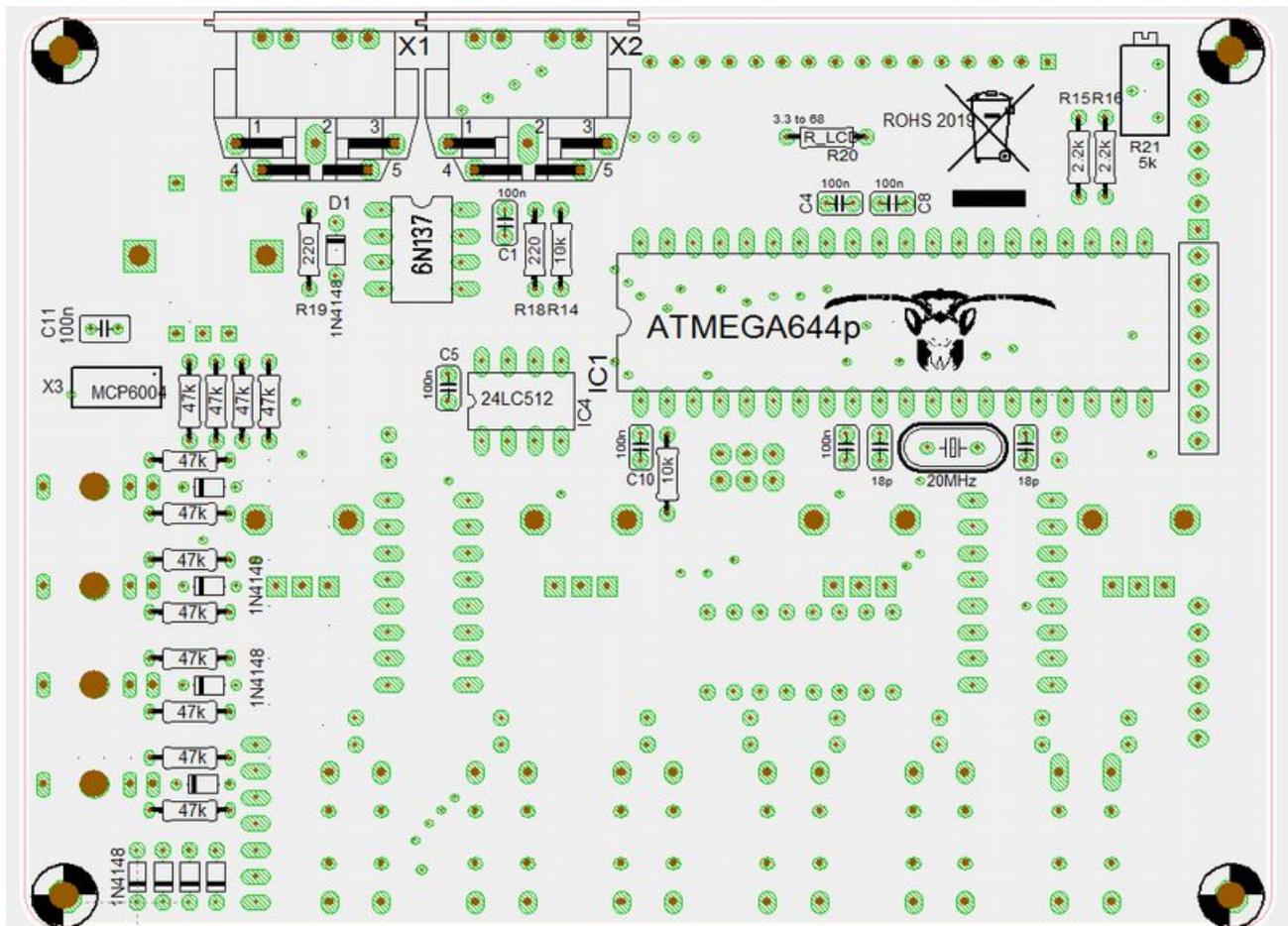
Here is the motherboard from above



and here's the motherboard from downstairs. That's where we start !  
Attention, the SMD IC MCP6004 rail to rail is already soldered on to make it easier for you !



Here now the placement plan from below

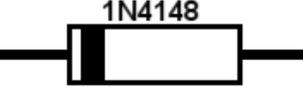


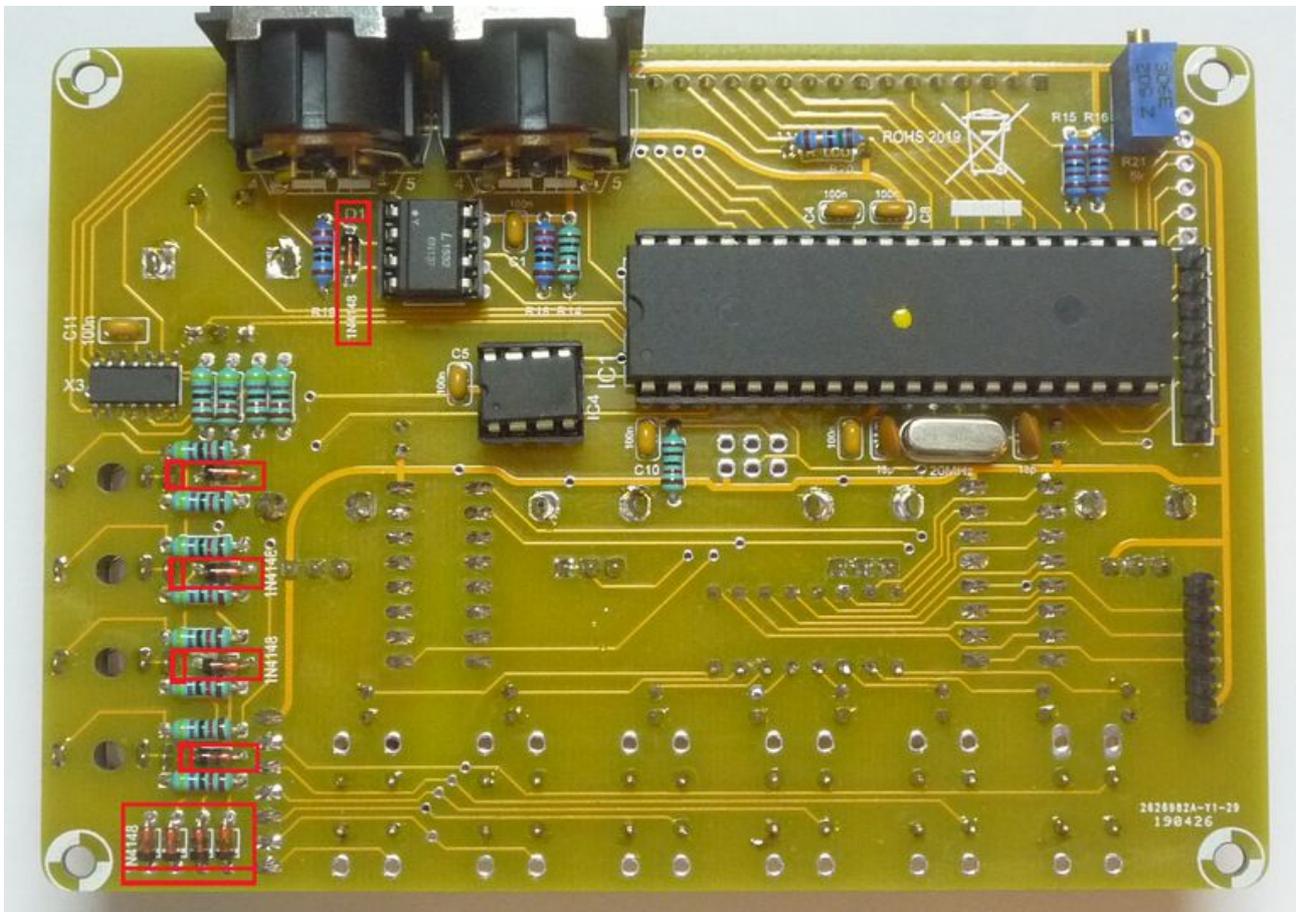
The following components are assembled from below:

- 1:) 2x2,2Kohm resistors
- 2:) 2x10Kohm
- 3:) 2x220ohm
- 4:) 1xR\_LCD  
\*\*\* Depending on the LCD, this resistor can easily be bridged with a wire. The supplied LCD displays have a built-in resistor and work with 5 volts. Other LCD displays can also be used. With these the pre-resistor must be calculated!
- 5:) 12x47Kohm
- 6:) 9xDiode 1N4148
- 7:) 7xCapacitor 100nF [104]
- 8:) 2xCapacitor 18pF
- 9:) 1xPoti 5K , R21
- 10:) 1x Quartz 20Mhz
- 11:) 2x8pin IC socket,1x 40Pin IC socket
- 12:) 2xMIDI socket



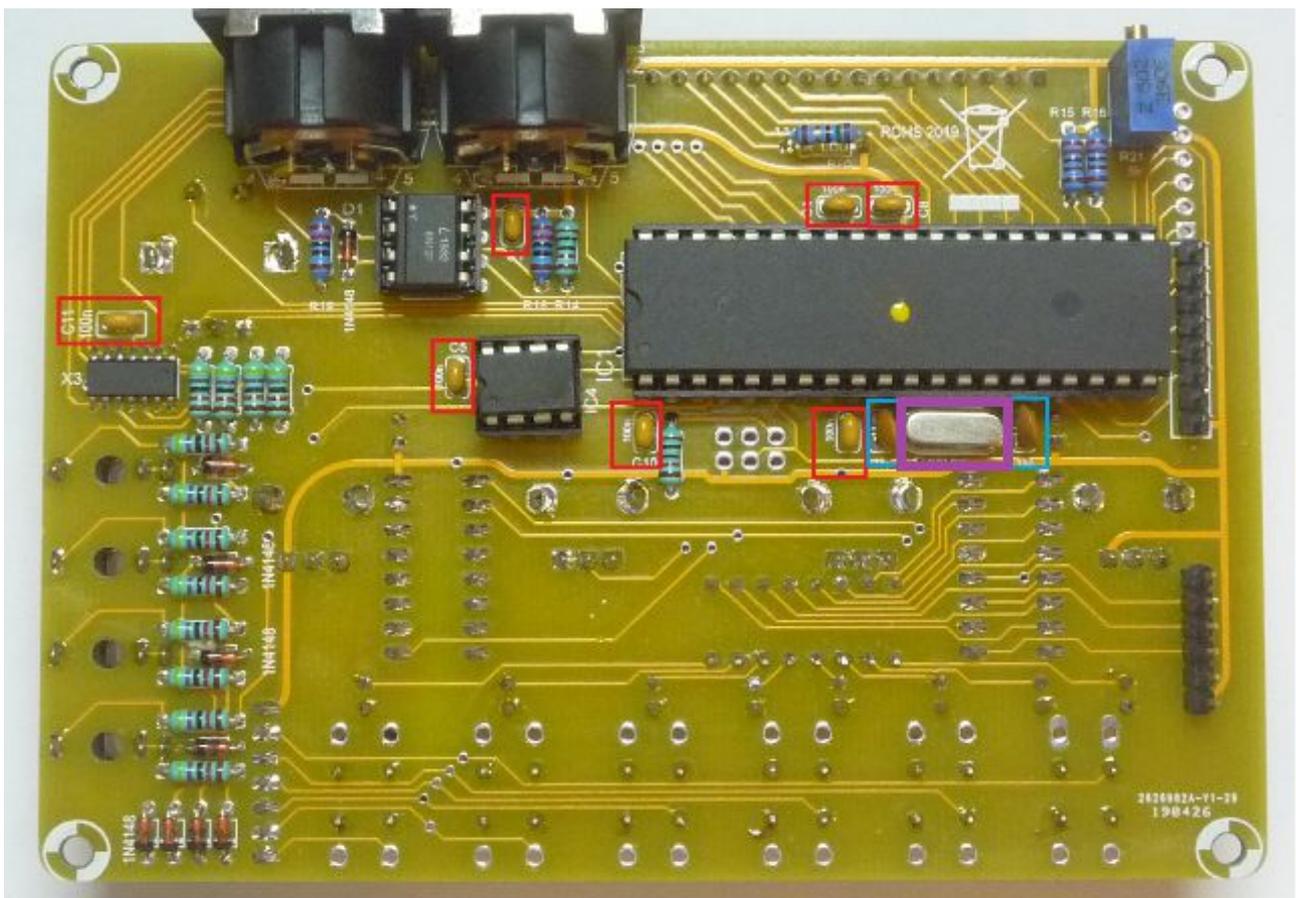
Now the diodes are soldered in

Image	Description	Quantity	Notes
 1N4148	Diode 1N4148	9	Attention polarized <b>color</b>



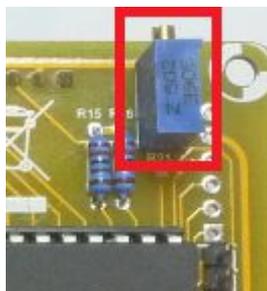
## 7x100nF Capacitors, 2x 18 pF Capacitors, 20Mhz

Image	Description	Quantity	Notes
	7x100nF marked [104]	7	color is
	2x18pF marked [18]	2	color is
	20 Mhz cristal / Quarz	1	Color is

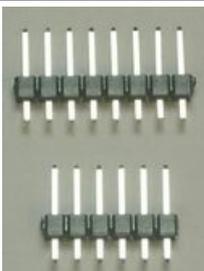


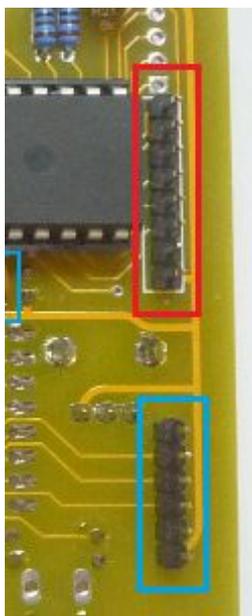
### The 5K poti for contrast control of the LCD display

Image	Description	Quantity	Notes
	Trimmer 5 Kohm	1	Value 5kohm marked with [Z502]

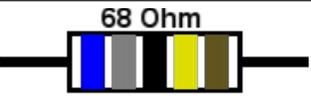


### The connectors to the filterboard

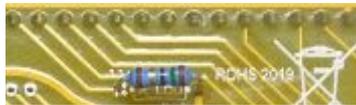
Image	Description	Quantity	Notes
	Pin-header Stiftleiste  1x 6pin, 1x 8 pin	1	1x8pin pin header 1x6pin pin header



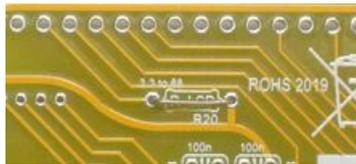
The series resistor for the LCD lighting. Depending on the LCD display, a solder bridge can also be used.

Image	Description	Quantity	Notes
 <p>68 Ohm</p>	Pre Resistor for the LCD or a bridge *depends of the LCD	1	blue,gray,black,gold,brown

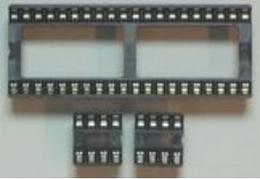
Choice one is a resistor >>68 ohm

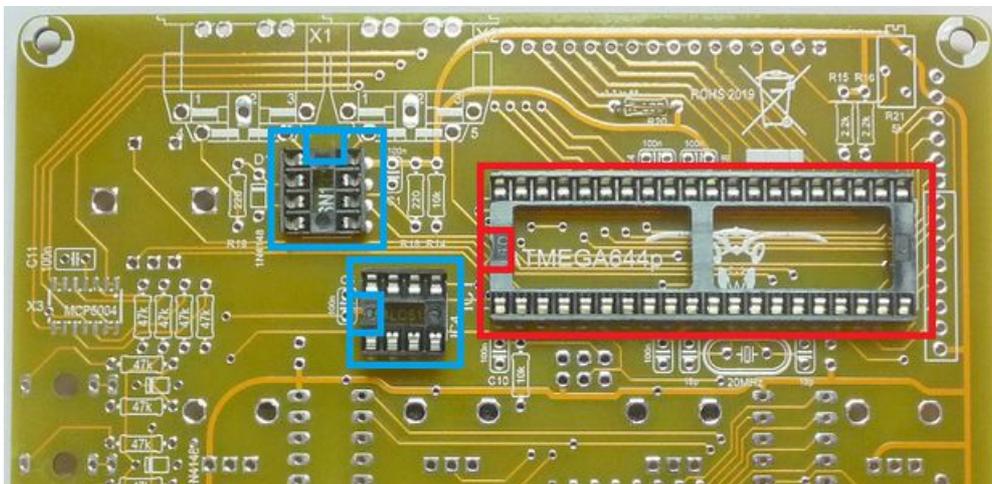


Choice two is a bridge >>0 ohm

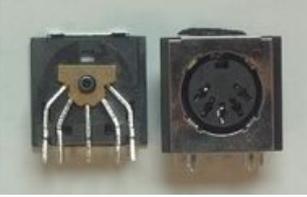


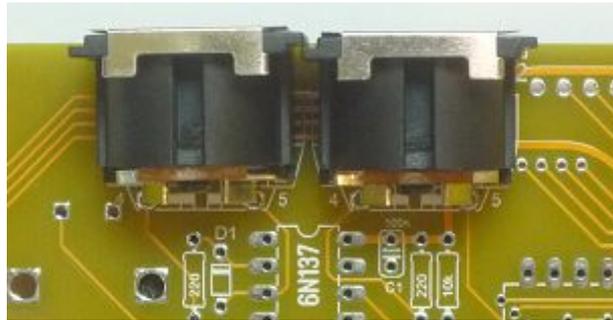
IC Socket 8 pin and 40 pin

Image	Description	Quantity	Notes
	1xIC socket 40 pol  2xIC socket 8 pol		Attention to the orientation <b>1xIC socket 40 pol</b>  <b>2xIC socket 8 pol</b>

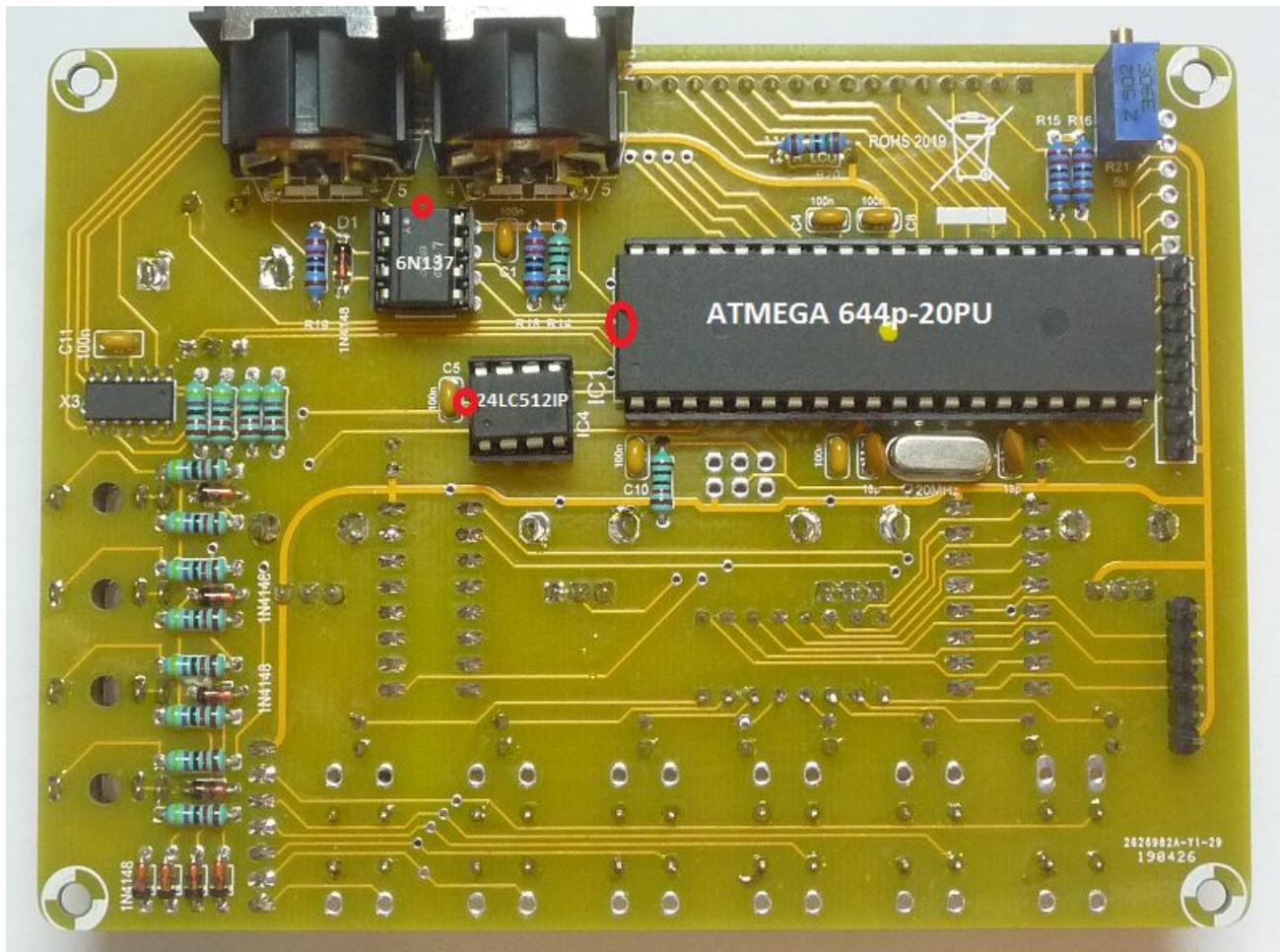


## 2xDIN 5 MIDI

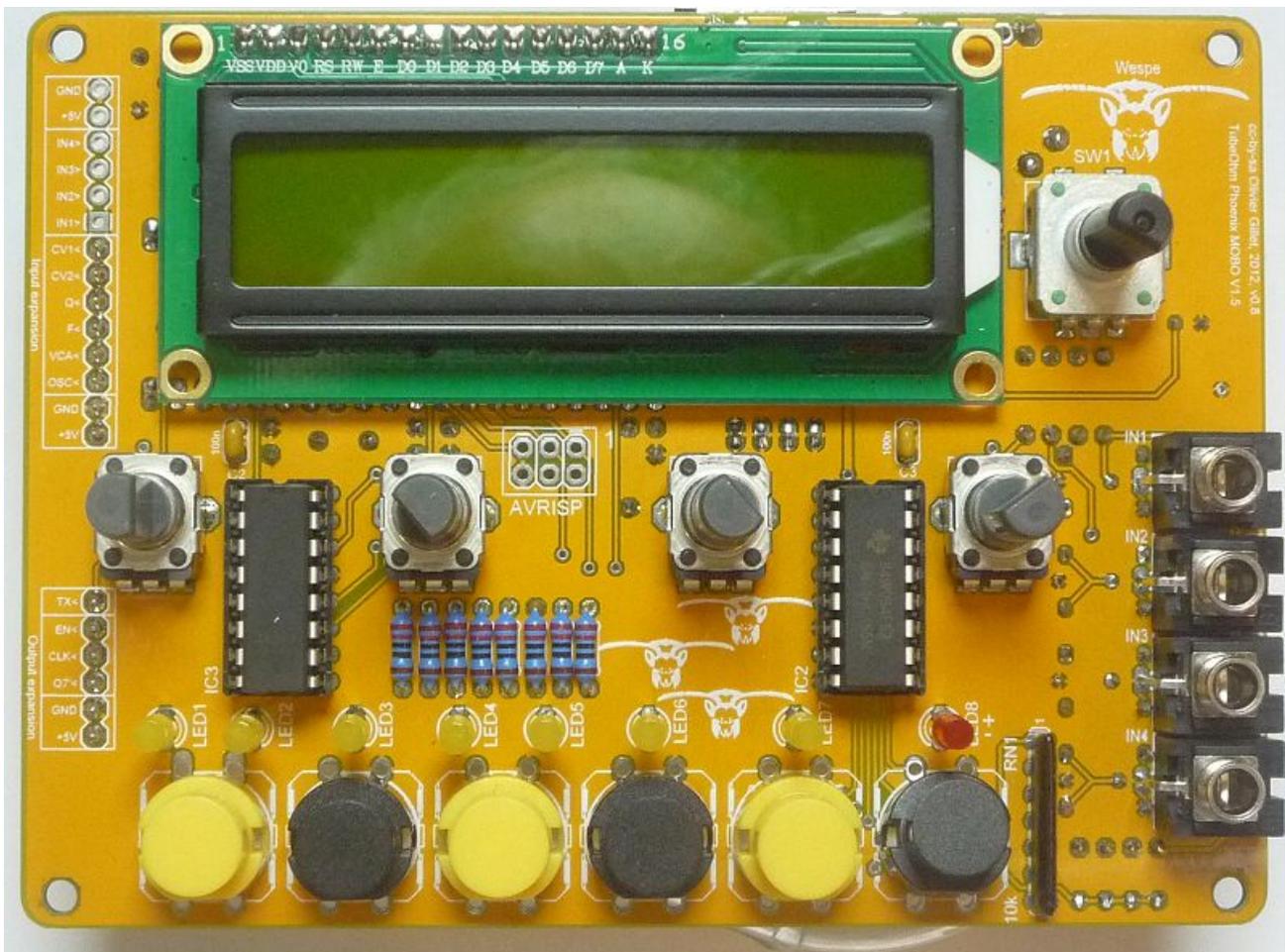
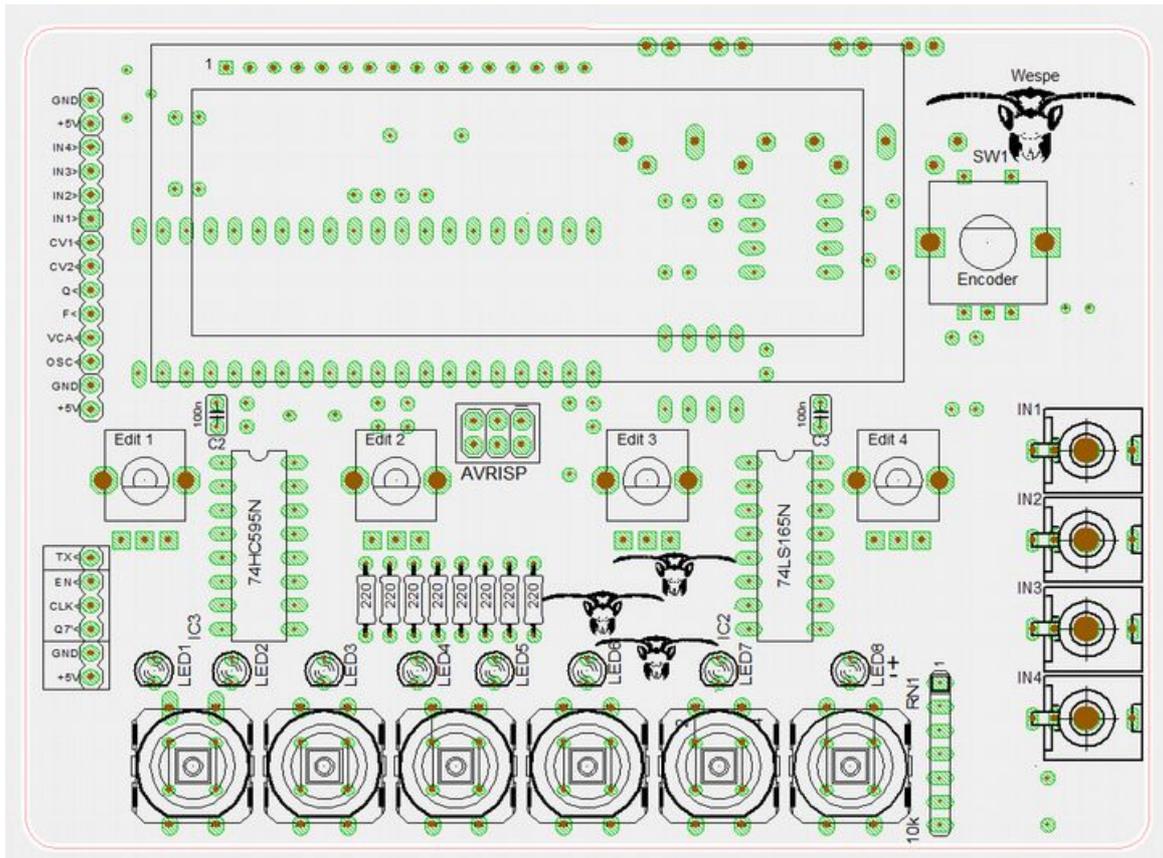
Image	Description	Quantity	Notes
	2xDIN 5 PIN MIDI		



Now the backside of the board is ready and should look like on the picture. The IC's are only inserted when the front side is also assembled and soldered.



Now we come to the front of the board.

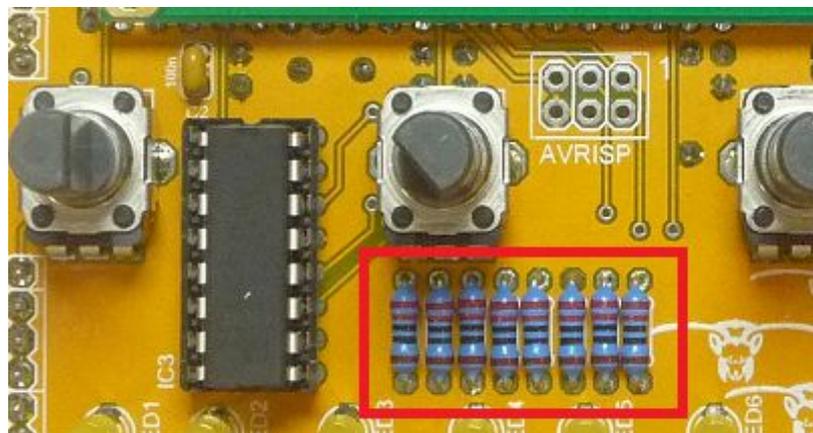


## The following components are assembled from above:

- 1:) 8x220 Ohm resistors
- 2:) C2,C3 =2x100nF marked [104]
- 3:) 1x RN1 resistor array, pin1 is marked by a white dot
- 4:) 2x16 pin IC socket for 74HC595 and 74HC165
- 5:) 7xLED yellow, 1x LED red (far right)
- 6:) 1X (2x3pin) AVRISP connector strip and a 16 pin connector strip for the LCD display
- 7:) 4xVC-in sockets , IN 1...4
- 8:) 4x 10 Kohm Potis , Edit1...Edit 4
- 9:) the encoder
- 10:) 8 buttons, round or square depending on DIY kit.
- 11:) one 6pin and one 8pin male connector

Let us now begin with the 220 Ohm resistors!

Image	Description	Quantity	Notes
	220 ohm	8	red,red,black,black,brown <b>color is</b>

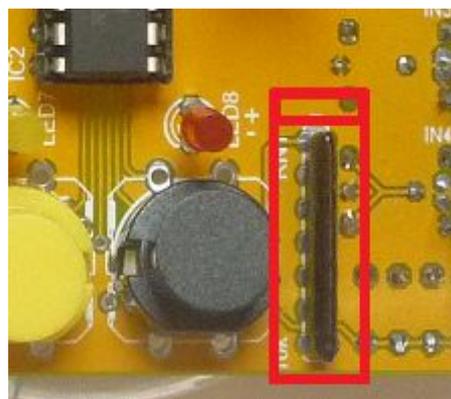


## 2x100nF capacitors

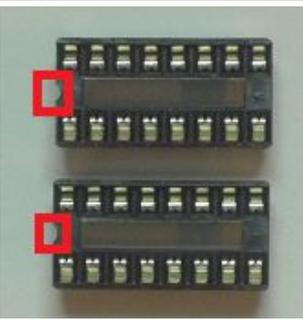
Image	Description	Quantity	Notes
	2x100nF marked [104]	2	color is

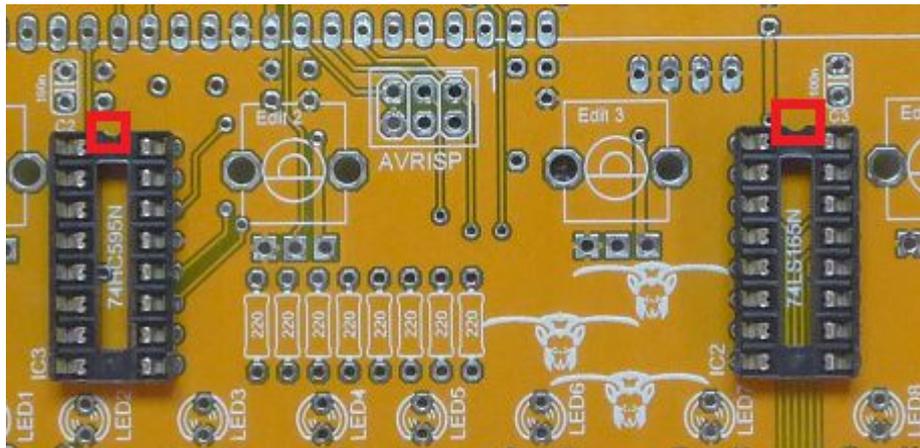


Image	Description	Quantity	Notes
	10 Kohm	1	marked A103G Pin 1 is marked with a ring/point color is



## 2x16 pin IC sockets

Image	Description	Quantity	Notes
	2x16 pin IC sockets	1	marked color is



### The buttons and LED's.

They should be soldered straight so that they pass through the holes of the front panel later.

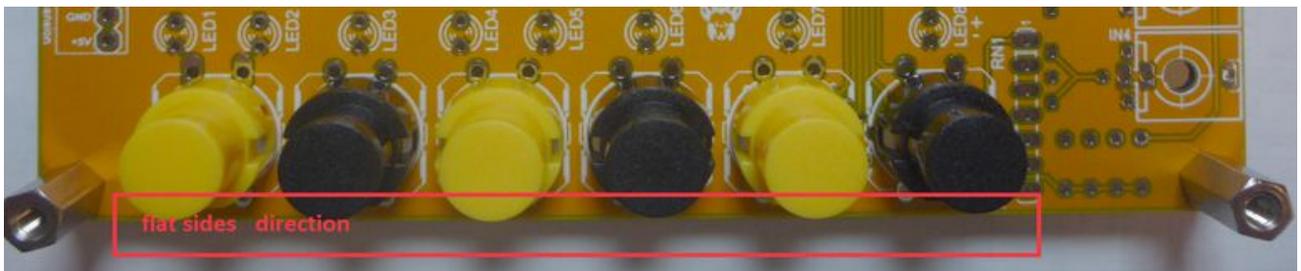
The LED's must not be more than 13 mm from the base plate to the top.  
I'd like to go. 10 mm are the spacers, 3 mm is the Plexiglas.

The following trick is possible. Screw the four 10mm spacers into the corners.  
Then you equip the LED's and the buttons without soldering them.  
Then screw the front panel onto the spacers. Now you have a template and can solder the LED's and switches.

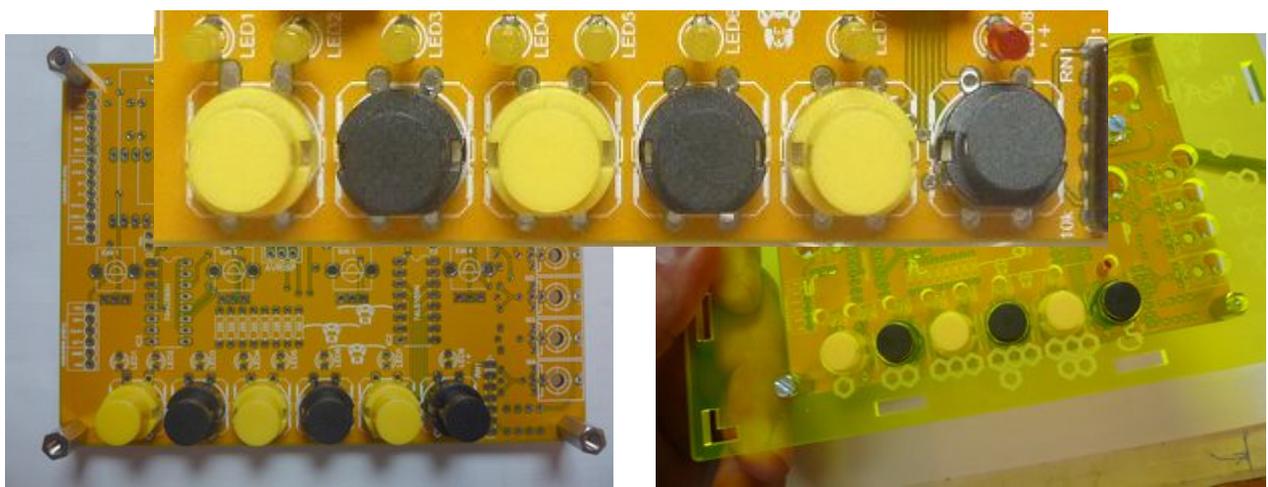
First solder one leg, align the component and then solder the remaining legs.

## 7 LEDs yellow, 1 LED red, 6 buttons- 3x yellow, 3x black

Image	Description	Quantity	Notes
	LED yellow 7 LED red 1* on position 8	7+1	Attention , MINUS is the short leg
	three yellow , three black switches	3y+3b	<b>Attention, the switches have a flat side.</b> You must solder the switches in the right direction !!

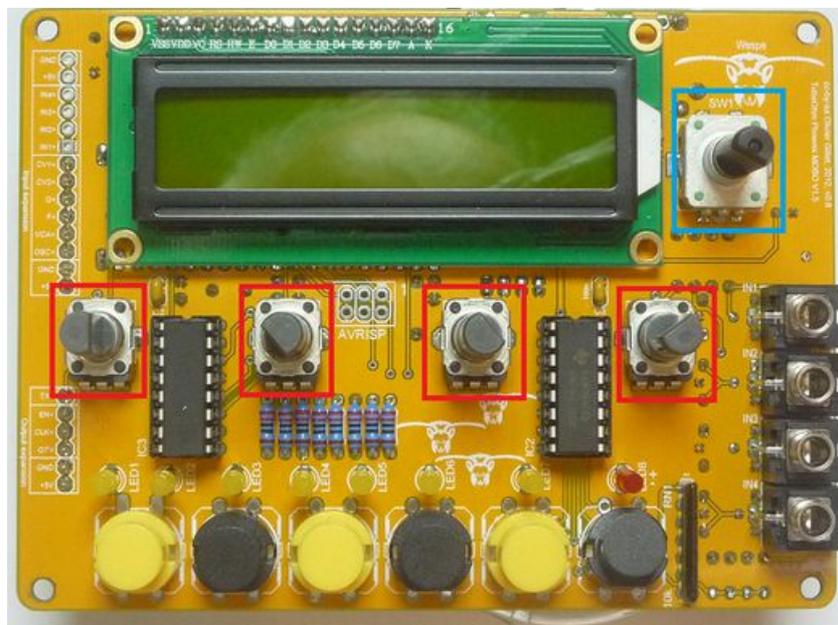


After you have soldered the buttons and LED's straight and in the right height it should look like this.



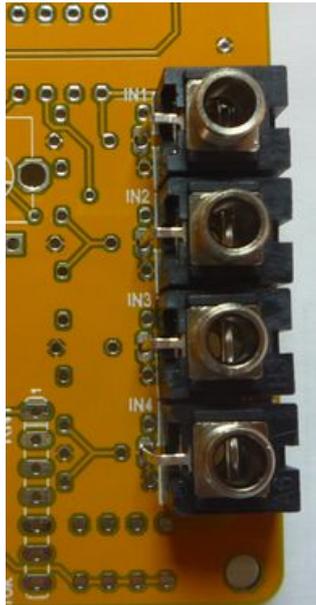
**So, now come the potentiometers and the encoder**

Image	Description	Quantity	Notes
	4xPoti 10 Kohm	4	color is
	1xEncoder 24	1	color is



**Now the 4x 3,5mm sockets are soldered in.**

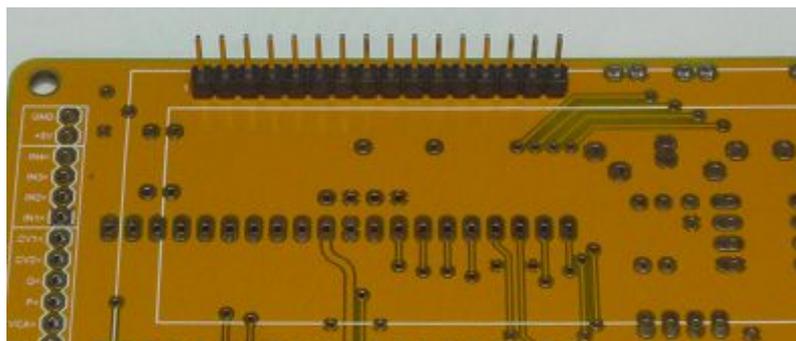
Image	Description	Quantity	Notes
	4x Jack- Socket	4	PJ301M-12 or PJ398SM



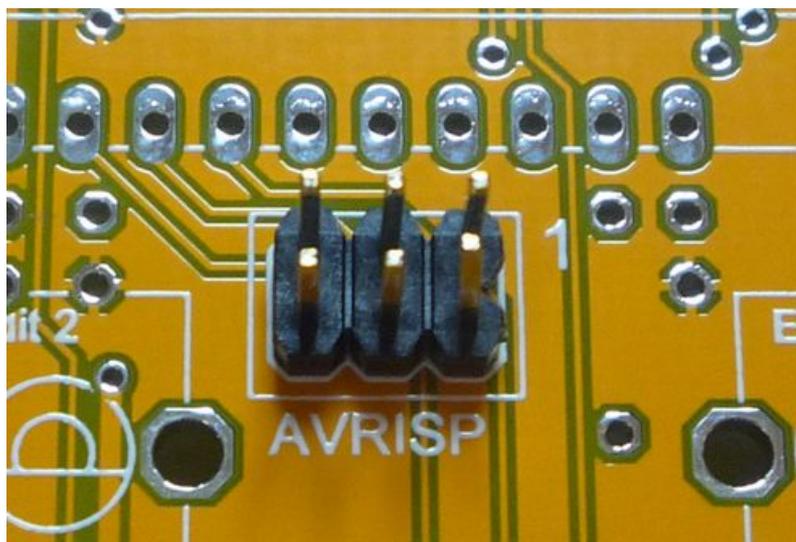
**LCD 1x16 connector strip and AVRISP (2x3) connector strip**

Image	Description	Quantity	Notes
	1x16 pin header	1	
	2x3 pin header AVRISP	1	You can solder it but you must not !!

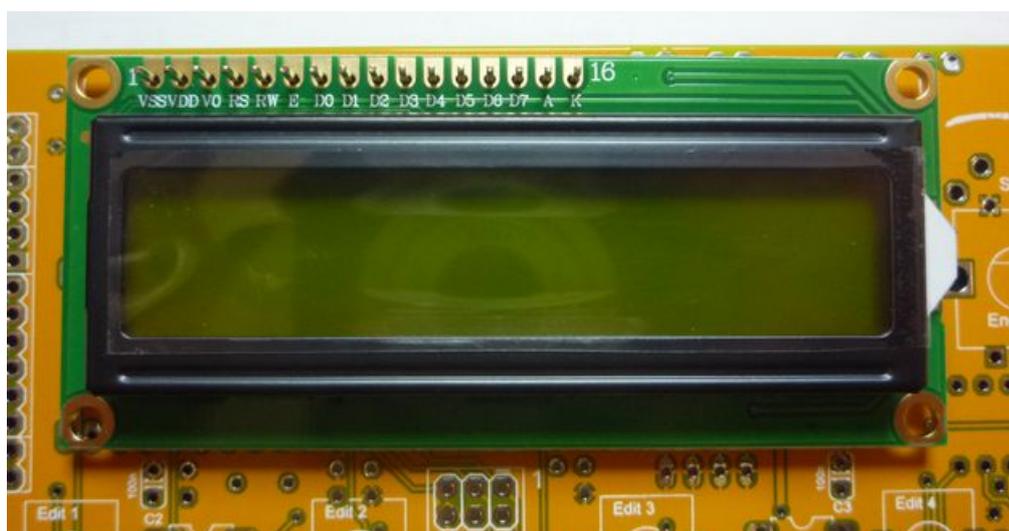
Position of the 16 pin header for the LCD display



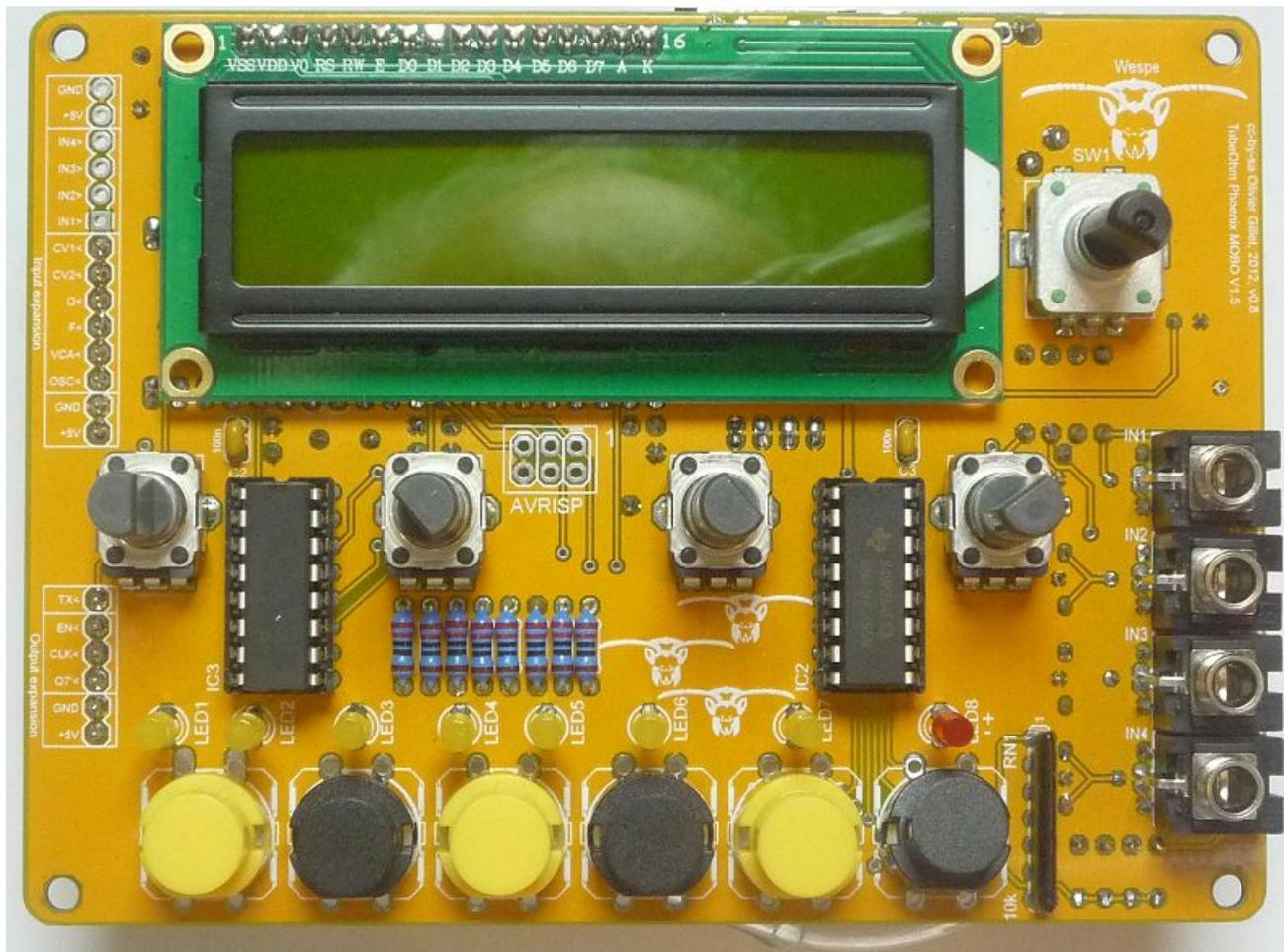
Position of the 2x3 AVRISP pin header



Now the LCD display is plugged onto the pin header and soldered. Please make sure that the LCD display is soldered.



Now your motherboard should look like this



Congratulations, the motherboard is now ready

The motherboard has the license CC-BY-SA  
The Files /Circuit diagrams / Boards -Files  
will be published on our website.