

LXR Assembly Instructions

 [available in french, too. Thanks to Egnouf for translating!](#)

This page provides you with all information necessary to assemble your kit. Please read and follow the instructions carefully. Even if you are just a few hours away from playing with your new synth, don't work in a hurry. A stupid mistake will take longer to fix than to take your time and work thoroughly.

If you have one of the early batch 1 PCBs (revision 0.4) please continue reading [here](#).

Pitfalls - Read First!!!

There are a few points to consider before starting your build.

- The R_LED resistors don't need to be populated. Only R7 needs a 220 ohm resistor, the others can be left unpopulated. There are some old pictures from the previous version in the instructions that still show the R_LED resistors in place. They are now replaced with the resistor networks.
- Solder the display and its connector last! The correct order is: Midi jacks, display connector pin header to PCB from top, then the display onto the connector. Any other order will **NOT** result in a successful build!
- Solder the 7805 voltage regulator with the heatsink attached, to guarantee the correct distance to the PCB!

Bill of Materials

The BOM for the project can be found [here](#).

Schematics

The schematics for the frontpanel are available as a PDF [here](#),

The schematics for the mainboard [here](#).



Preparations

Tools you'll need:

- Soldering iron
something like the [ERSA TIP 260](#) is sufficient
- Soldering iron stand
for example [this one](#)
- Solder
use thin solder, 1mm or less. Something like [this](#). Do not use the thick plumbing solder!
For beginners solder containing lead is recommended.
- Wire cutter
- [Solder sucker](#)
not essential if you do everything right, but very handy if you have to desolder anything.
- Multimeter
- Screwdriver
- a 2.1mm DC wallwart power supply (7-9V DC, **center pin positive, at least 600mA**).
- some electrical tape (Isolierband)

It is assumed that you already know how to solder. If you never soldered before, we strongly recommend you to solder a small and cheap kit first to learn the basic skills. Curious Inventor have a nice soldering guide [available here](#).

Disclaimer

We are not responsible for any assembly related failures such as non working kits due to build errors, accidents involving electricity and/or hot soldering irons and not even angry family members due to excessive soldering sessions.



Lets start!

Power up your soldering iron, put on some nice music or your favourite radio show and enjoy the build!

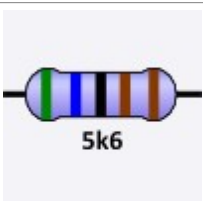
Bottom of the PCB

We start on the bottom of the board. There you will find the power supply and the MIDI circuitry, as well as all the connectors to the outside world.

Step 1 – Resistors

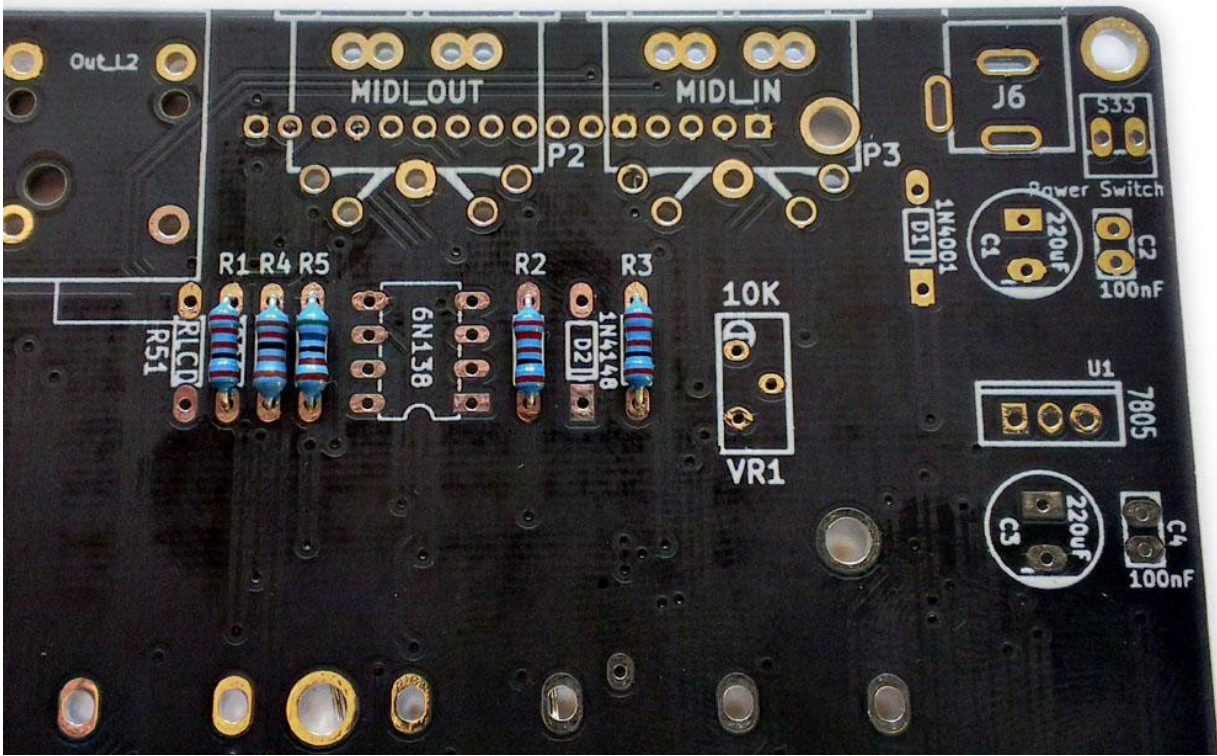
All parts with a part number beginning with R are resistors.

Resistors don't have a polarity, so it doesn't matter which way around you solder them in.

Image	Description	Amount	Part nr.	Notes
 A resistor with color bands: red, red, brown, gold. The value 221 is printed below it.	221 Ohm resistor	3	R1 – R3	
 A resistor with color bands: brown, black, red, gold. The value 1k2 is printed below it.	1k2 resistor	1	R4	
 A resistor with color bands: green, blue, brown, gold. The value 5k6 is printed below it.	5k6 resistor	1	R5	
not populated	Only needed if you have a custom LCD instead of the OLED provided with	0	R51	Only for LCD users! Controls the backlight brightness.



	the kit. Value depends on the LCD type.			
--	---	--	--	--





Step 2 – Diodes

Part numbers starting with a 'D' are diodes. Unlike the resistors, **the diodes are polarized!**

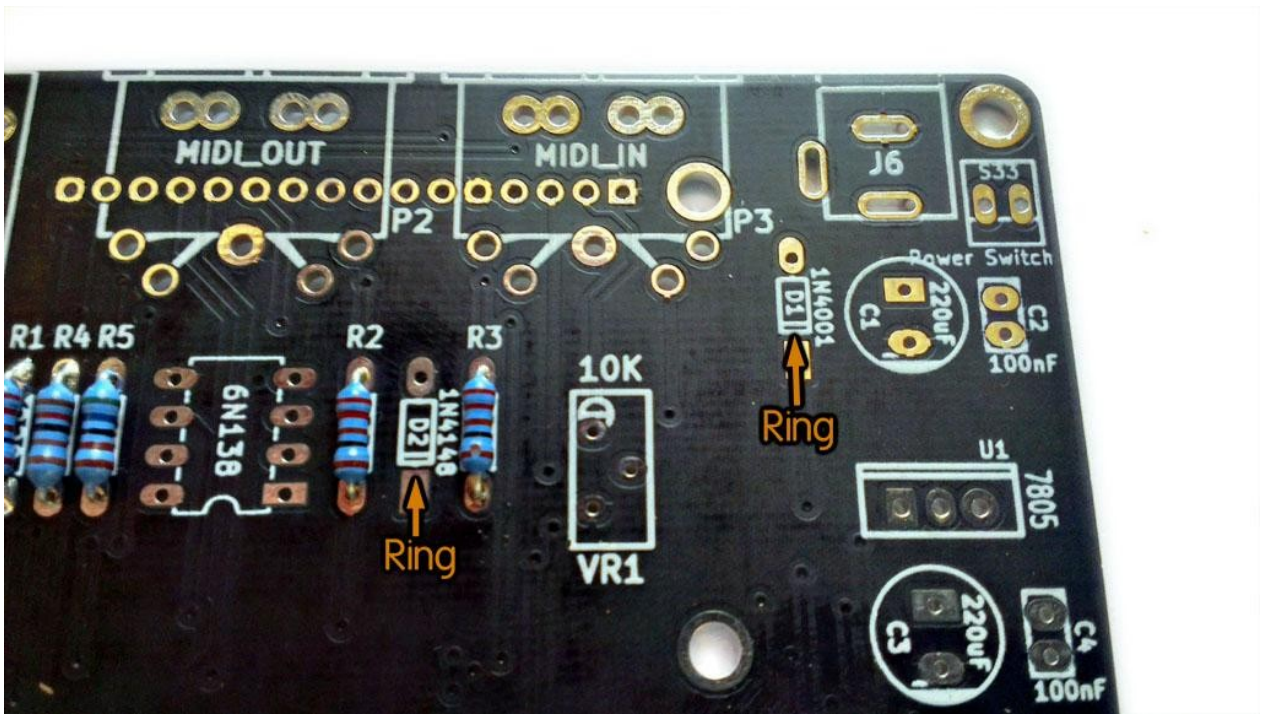
Image	Description	Amount	Part nr.	Notes
-------	-------------	--------	----------	-------



	<p>1N4001 diode</p> <p>Function: reverse polarity protection if a wrong PSU is attached to the DC jack.</p>	<p>1</p>	<p>D1</p>	<p>Polarized part!</p> <p>Silver ring.</p>
	<p>1N4148</p> <p>Function: protection diode for the optocoupler input</p>	<p>1</p>	<p>D2</p>	<p>Polarized part!</p> <p>Black ring</p>

- One side of the diode is marked with a ring around its body. This ring can also be found on the silkscreen as a line. The ring for both diodes should face away from the MIDI connectors.





Step 3 – Capacitors

All the capacitor part numbers start with a 'C'.

The kit uses different kinds of capacitors. **The electrolytic ones are polarized.**

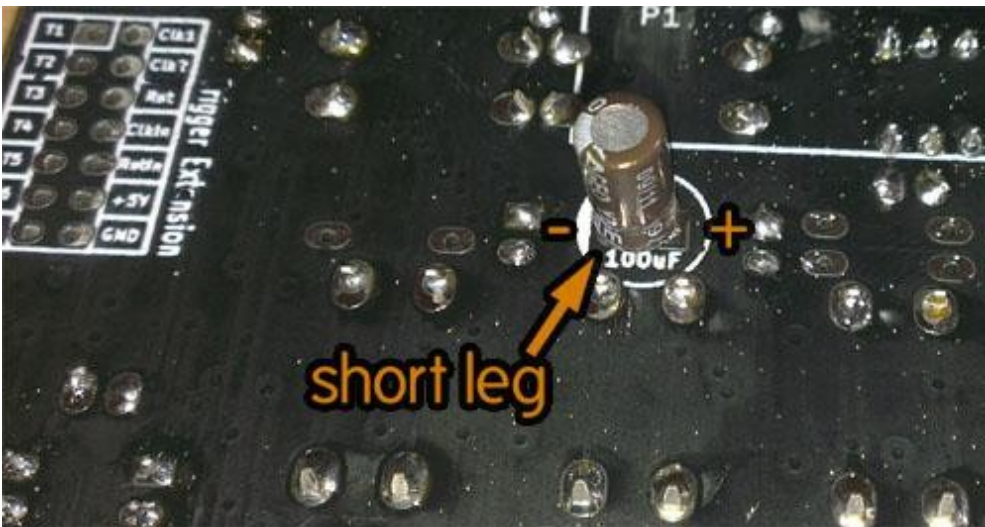
Image	Description	Amount	Part nr.	Notes
	Electrolytic capacitor 220uF Function: smoothing capacitors for the PSU	2	C1, C3	Polarized part! The shorter leg is negative, the longer leg is for the positive voltage.
	Electrolytic capacitor 100uF Function: smoothing capacitor for the SD-card power supply.	1	C23	
	100nF ceramic capacitor Function: smoothing capacitors for the PSU	2	C2, C4	

- solder the 220uF and 100nF caps for the PSU. Watch out for the polarity of the two 220uF caps! The short leg goes into the round pad. The 100nF caps are not polarized. Insert them either way.





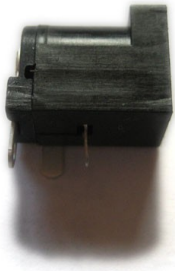
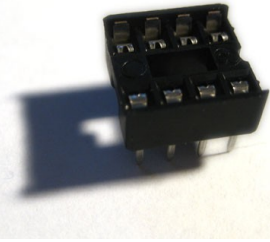

solder the 100uF cap below the mainboard connector P1. Once again, this is a polarized part. Short leg to round pad.



Step 4 – Voltage regulator and DC jack

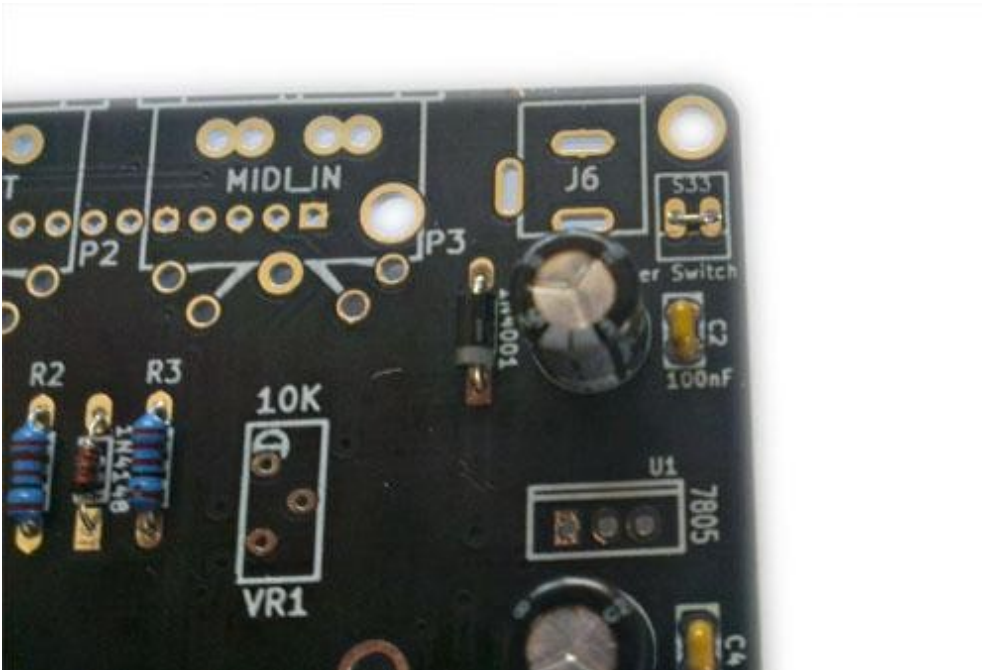
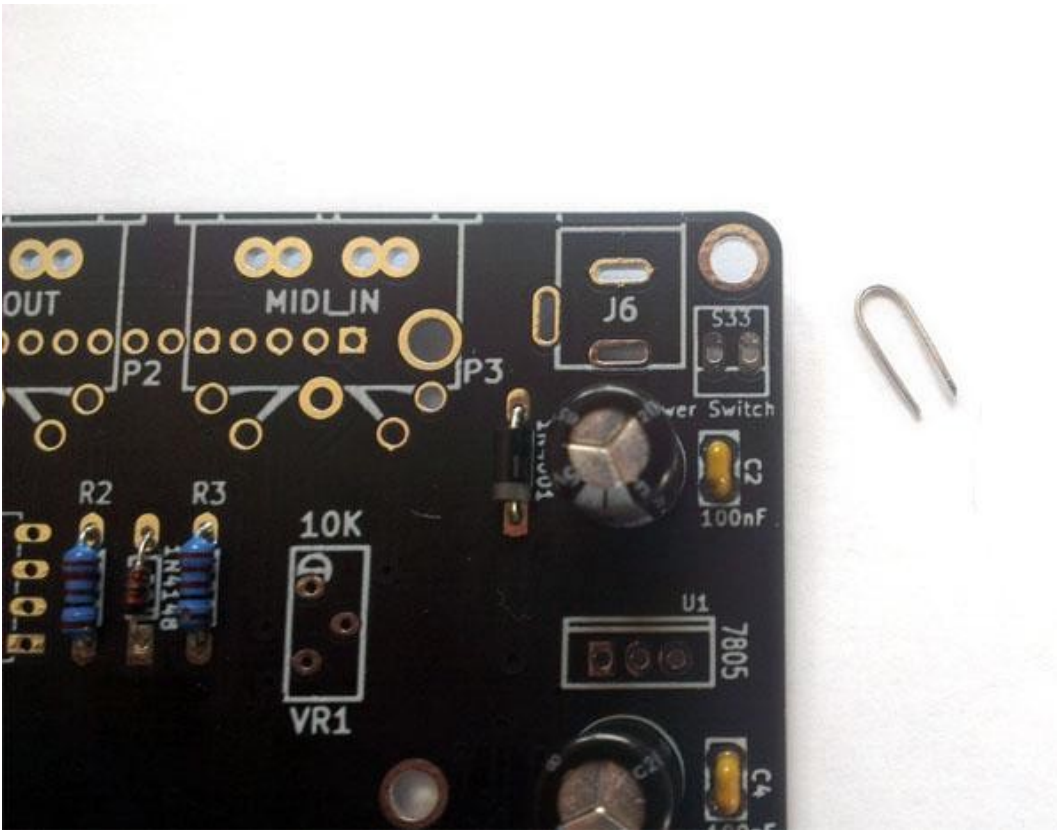
Image	Description	Amount	Part nr.	Notes
	Heat sink for the voltage regulator	1		
	M3 screw	1		
	7805 voltage regulator Function: takes an unregulated input voltage and makes it a stable 5V voltage.	1	U1	Do not solder without attached heatsink!



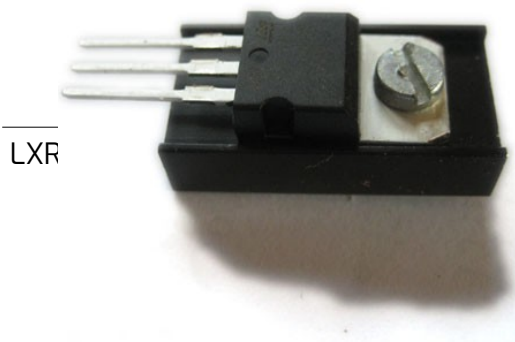
	DC jack	1		
	IC socket DIP 8	1	OK1	Notch has to be the same direction as the print on the PCB
	Bridge	1	-	This is just a normal piece of wire. use one of the cut off resistor legs from the previous steps.

- Start by soldering the bridge to position 'S33'. You can use one of the cut off resistor legs from the previous steps. If you like to use a power switch for your build (not included in the kit) you have to attach your switch instead.





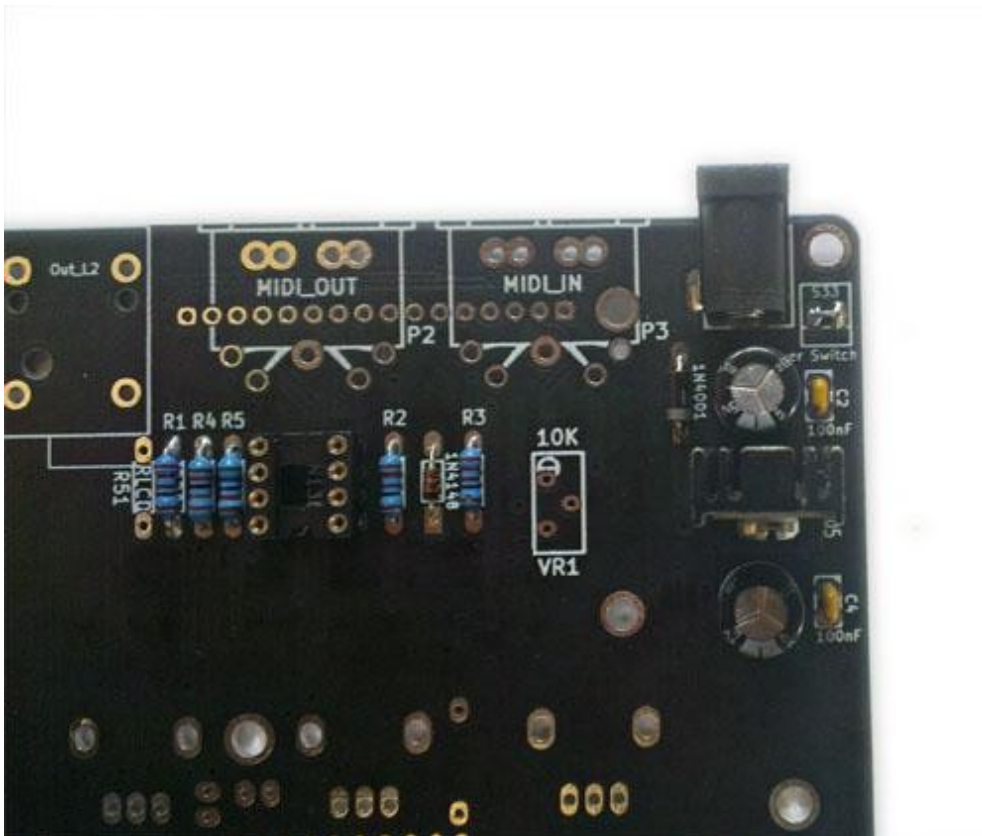
- Solder the DC jack.
- Attach the heatsink to the voltage regulator using the M3 screw.



- Solder the voltage regulator. The side with the heatsink has to face the DC jack.



- Solder the IC socket. Do not yet put any IC inside the socket. Align the notch of the socket with the notch on the PCB silkscreen.

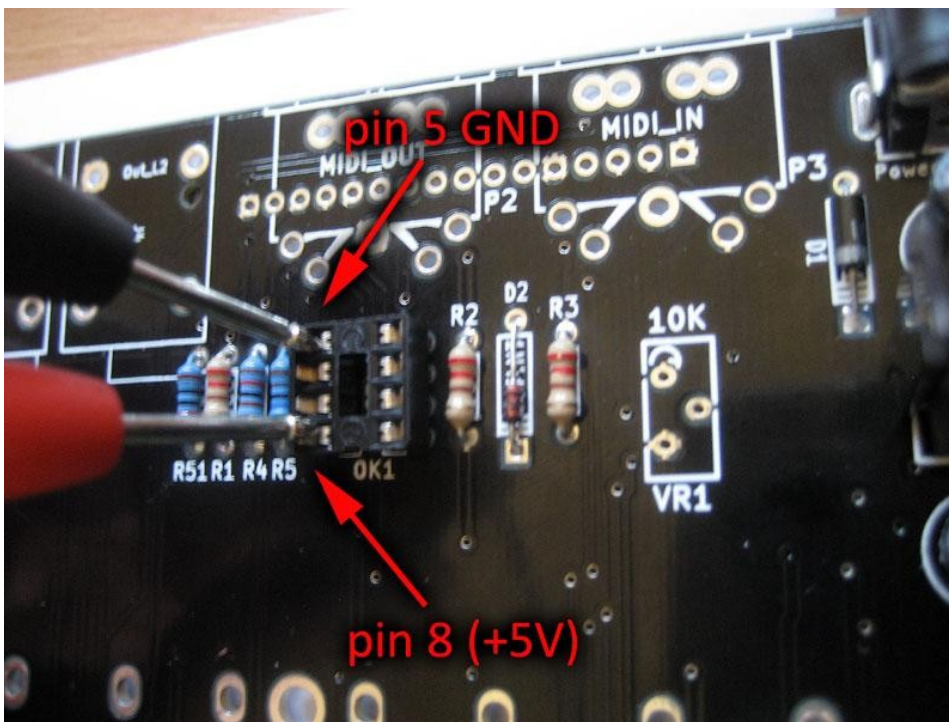
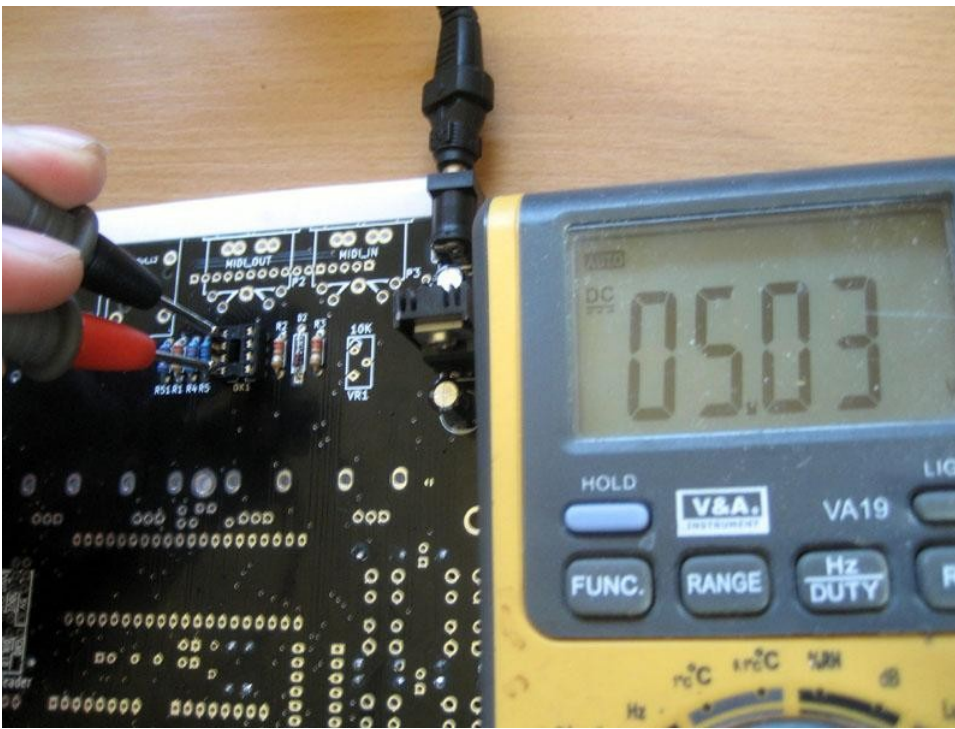


Step 5 – Time for a first test

Congratulations! You just finished soldering the 5V PSU of your new drum machine. Now is a good time to test if it is working.

- Attach your power supply to the DC jack (7-9V DC, **center pin positive, at least 600mA@7.5V**).
- Use your multimeter to measure the voltage between pin 5 (GND) and pin 8 (+5V) of the IC socket 'OK'
- Your multimeter should show about +5V





Troubleshooting – My multimeter does not show 5V

- 4.9V or 5.1V is ok, too!
- If it shows -5V you just mixed up the test points (red and black probe switched)
- Is the wire bridge on position S33 soldered?
- Is the diode orientation correct? (both rings point away from the midi jacks)



- Is the electrolytic capacitor polarity correct?
- Is your power supply polarity correct?
- Is the orientation of the 7805 correct? It should not get hot at this point!
- If nothing works, head over to the [forum](#). Do not continue your build if the PSU is not working!

Step 6 – Connectors

As a last step we will solder all the connectors but the USB jack to the bottom of the PCB.

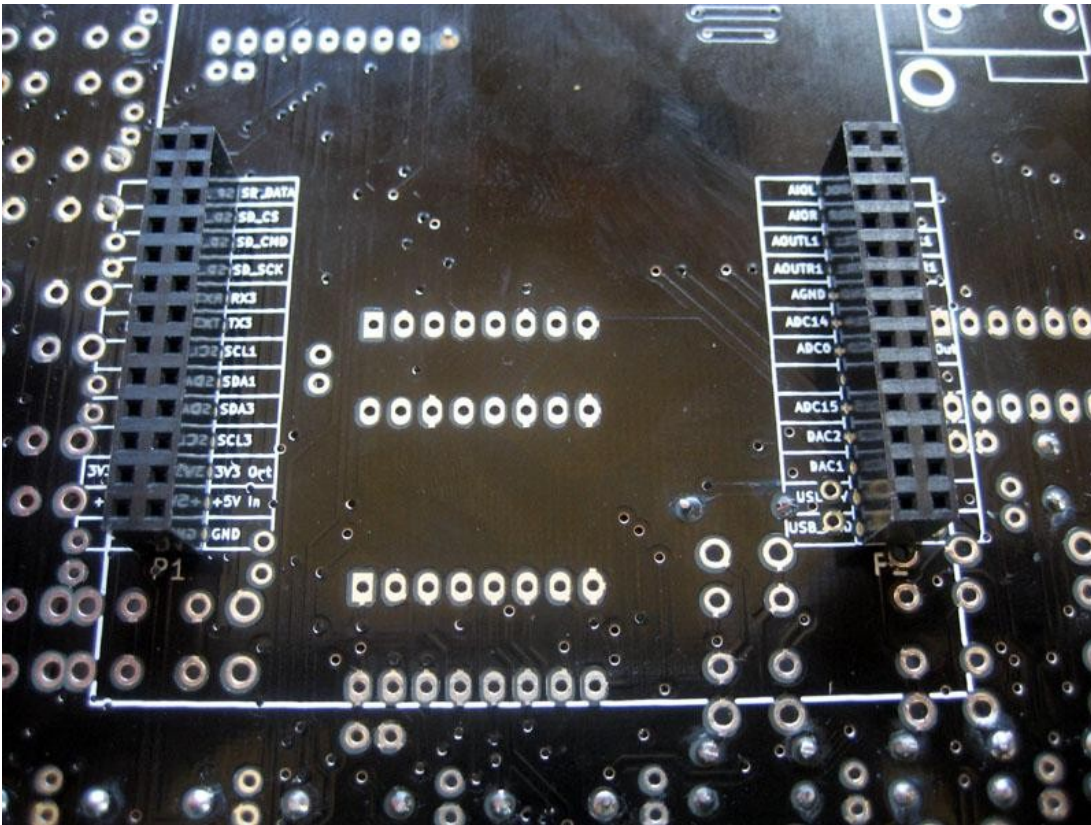
Image	Description	Amount	Part nr.	Notes
	MIDI jacks	2	P2, P3	
	Audio Jacks	4	J2, J3, J4, J5	



	Female 2x13 header, 2.54mm	2	P1, P2	
	USB jack type B	1		
not populated	Contrast trimmer 10k only needed if you use a custom display instead of the OLED provided with the kit.	1	VR1	Only for LCD users!

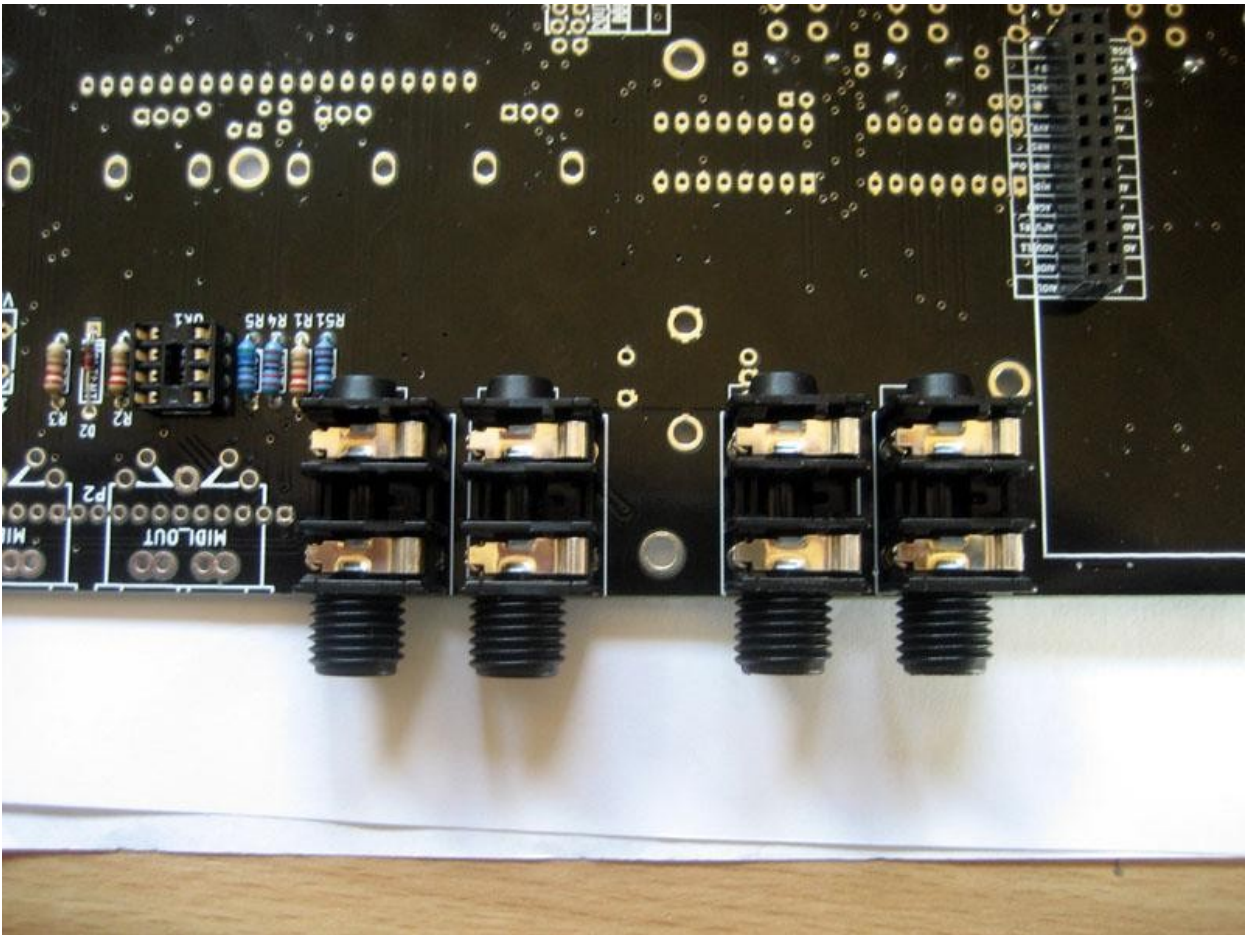
- solder the female connectors P1 and P2. Make sure they are completely flat on the PCB! If they are not, you may have trouble installing the mainboard later on. It may be easier to install them completely flat, if you attach them to the mainboard for soldering. This ensures that the mainboard will fit once the connectors are soldered.





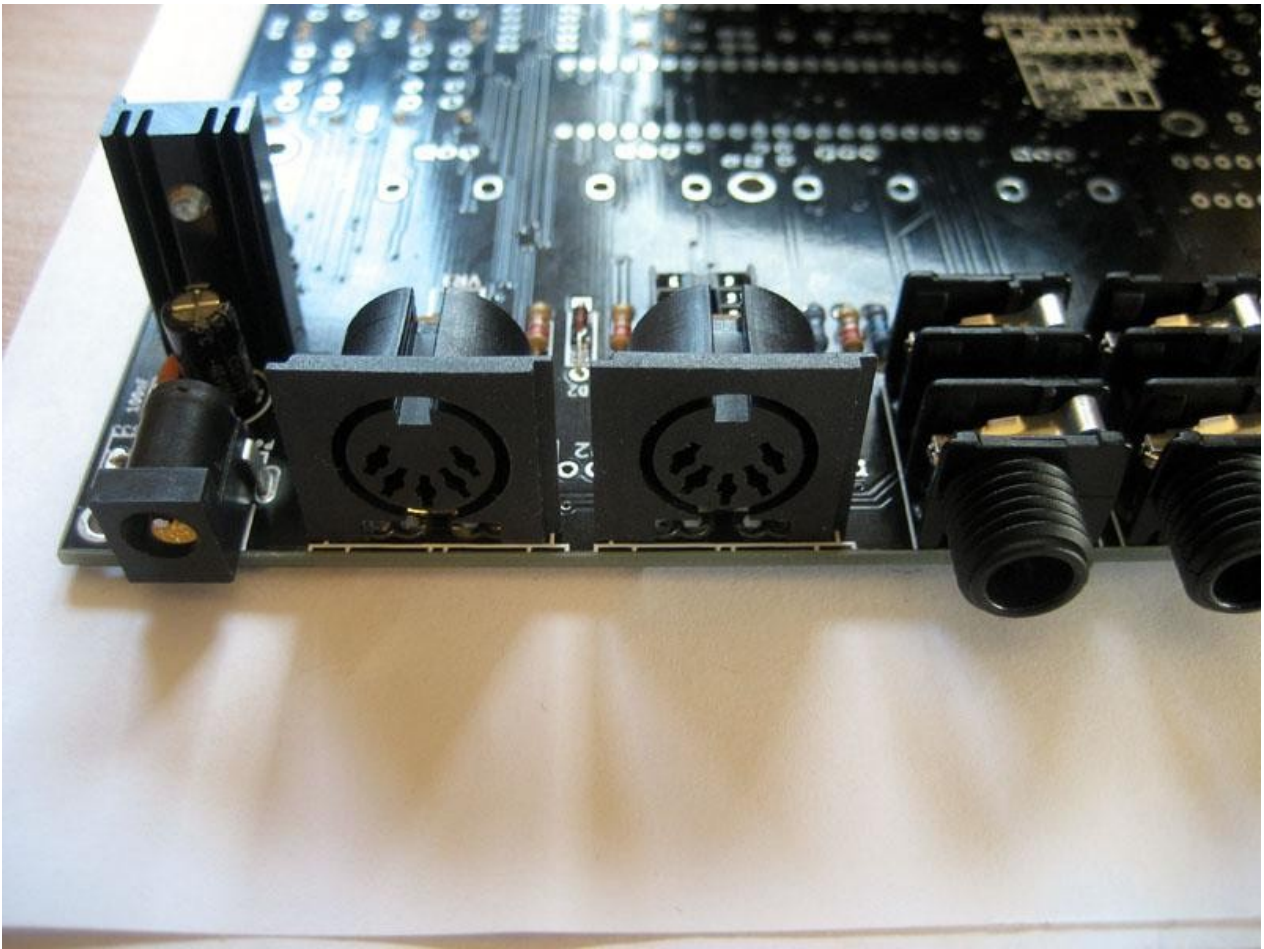
- solder the audio jacks





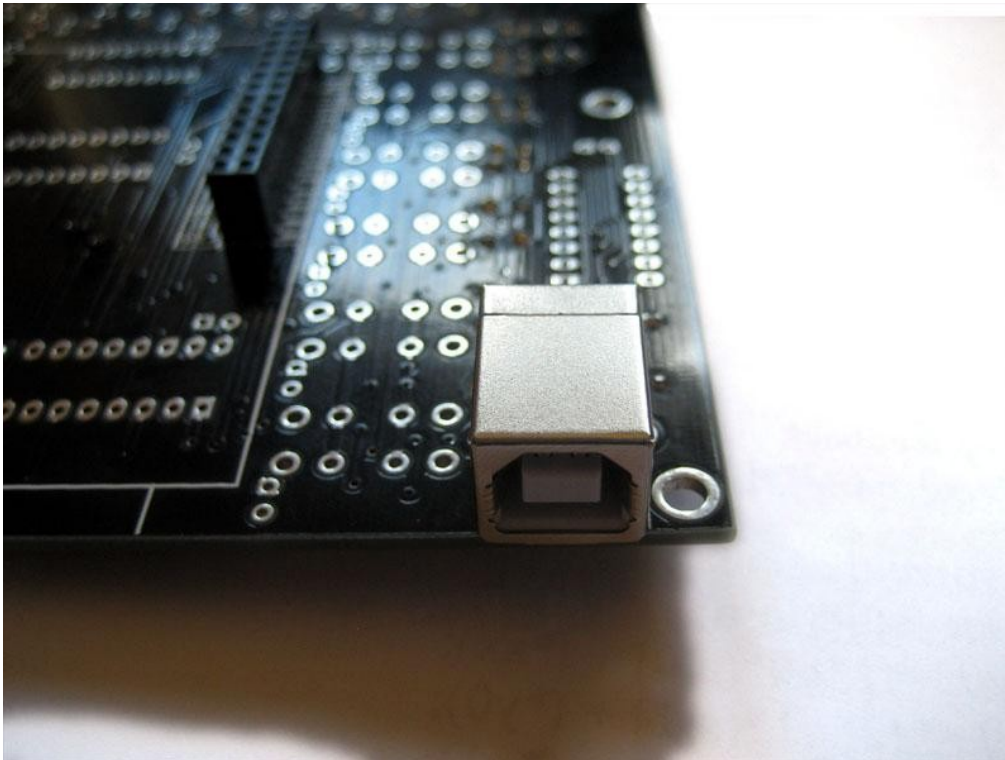
- Next are the MIDI jacks. Solder the center pin first, then align the jack flat on PCB and solder the rest.
- Cut away the pins on the top of the PCB as short as possible, so that later on in the build the display can lay flat on the other side of the PCB.





- Now solder the USB jack





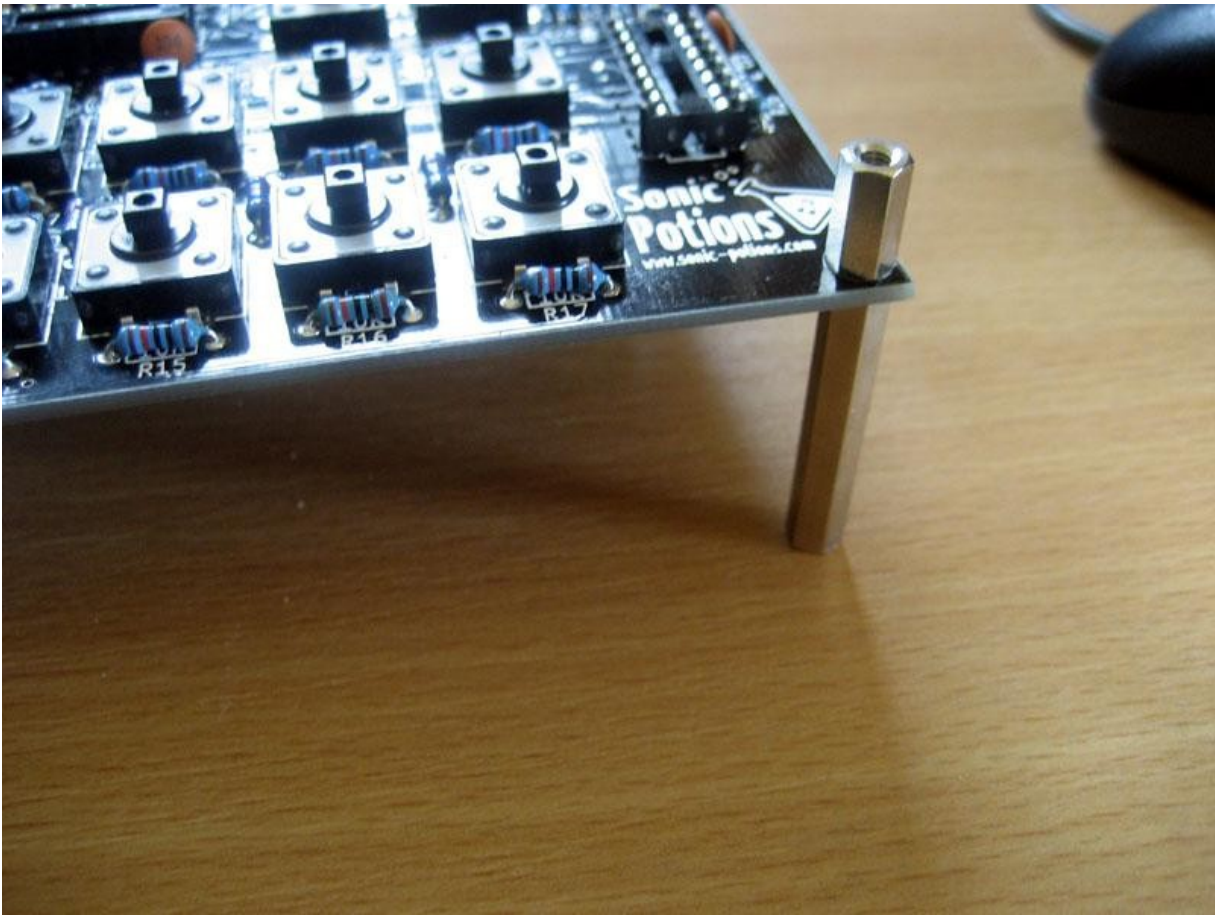
Top of the PCB

After finishing the bottom, we will now assemble the top of the PCB.



If you like you can already add the 4 metal stand offs to the 4 corners of the board.

This makes working on the board easier.



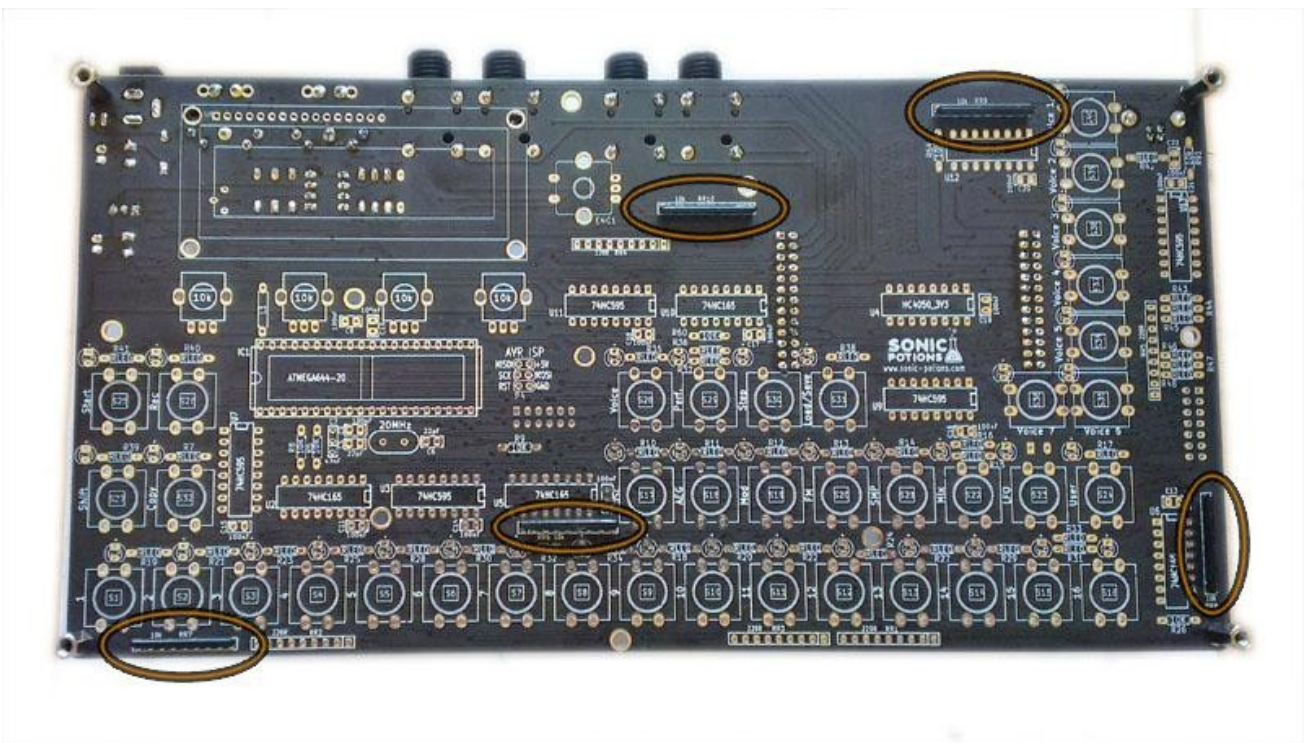
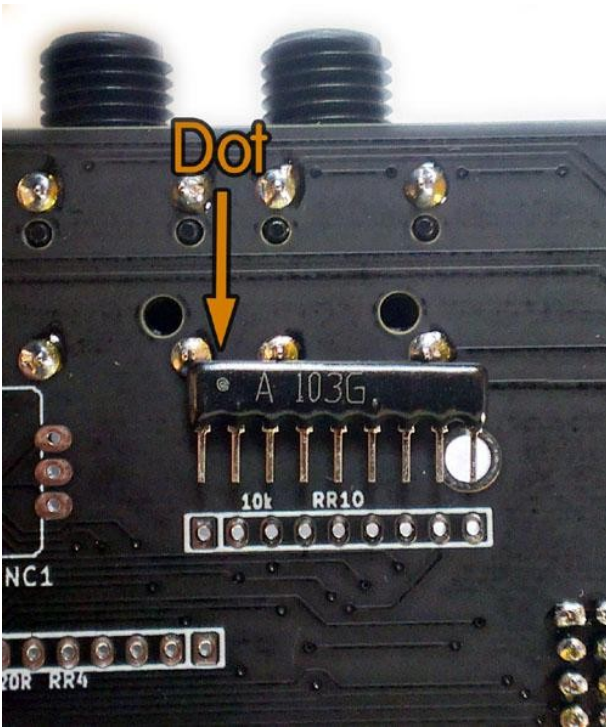


Step 7 – Resistor networks

Image	Description	Amount	Part nr.	Notes
	<p>10k resistor network 9 pin, common gnd</p> <p>Function: These are pull-down resistors to GND for the switches.</p>	5		
	<p>220ohm resistor network 9 pin, common gnd</p> <p>Function: These are current limiting resistor for the LEDs</p>	5		<p>These are current limiting resistor for the LEDs. If you are using other LEDs than provided with the kit, you have to change their value accordingly.</p>

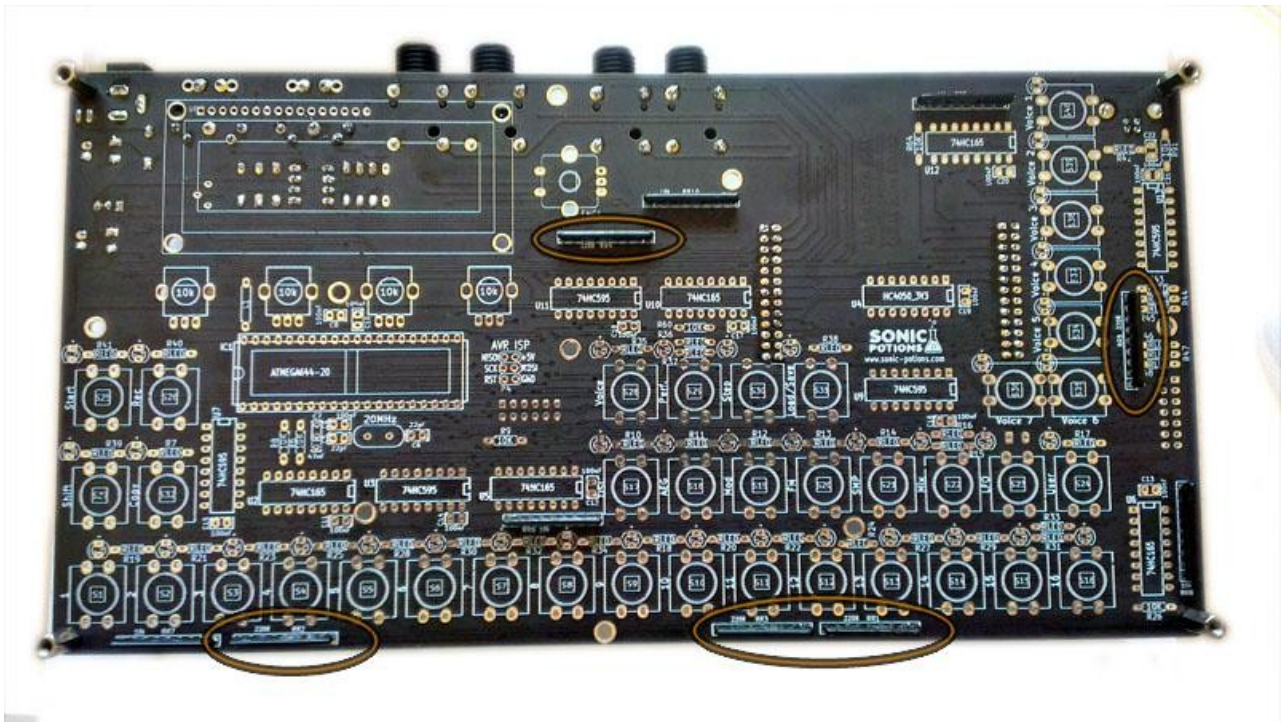
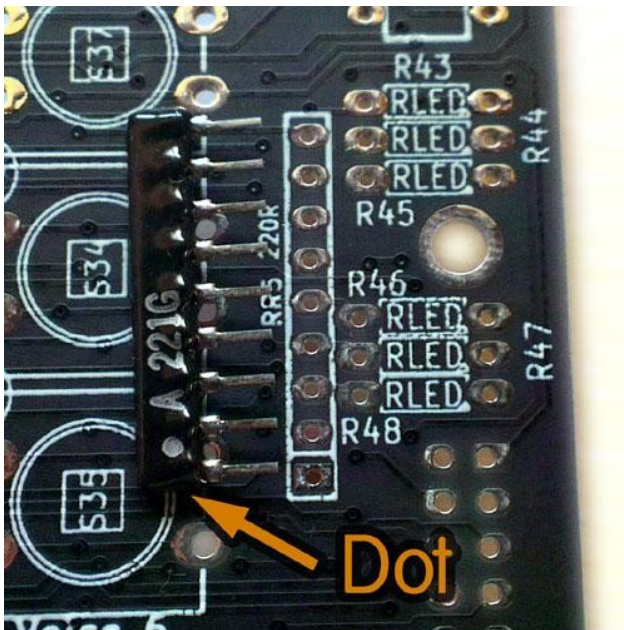
- Solder the 10k resistor networks. The GND pin is marked by a dot. The pin with the dot has to be soldered into the pad marked with a square around it.





- Next, solder the 220 ohm networks. As before, the dotted side goes to the square.





Step 8 – Capacitors

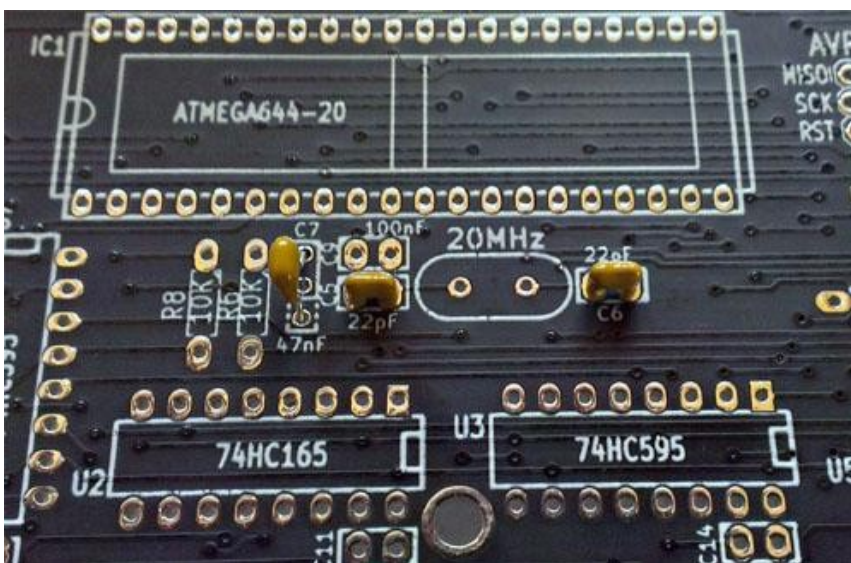
There are only ceramic type capacitors used on the top of the PCB. They are non polarized and can be inserted in either direction.

Image	Description	Amount	Part nr.	Notes
-------	-------------	--------	----------	-------

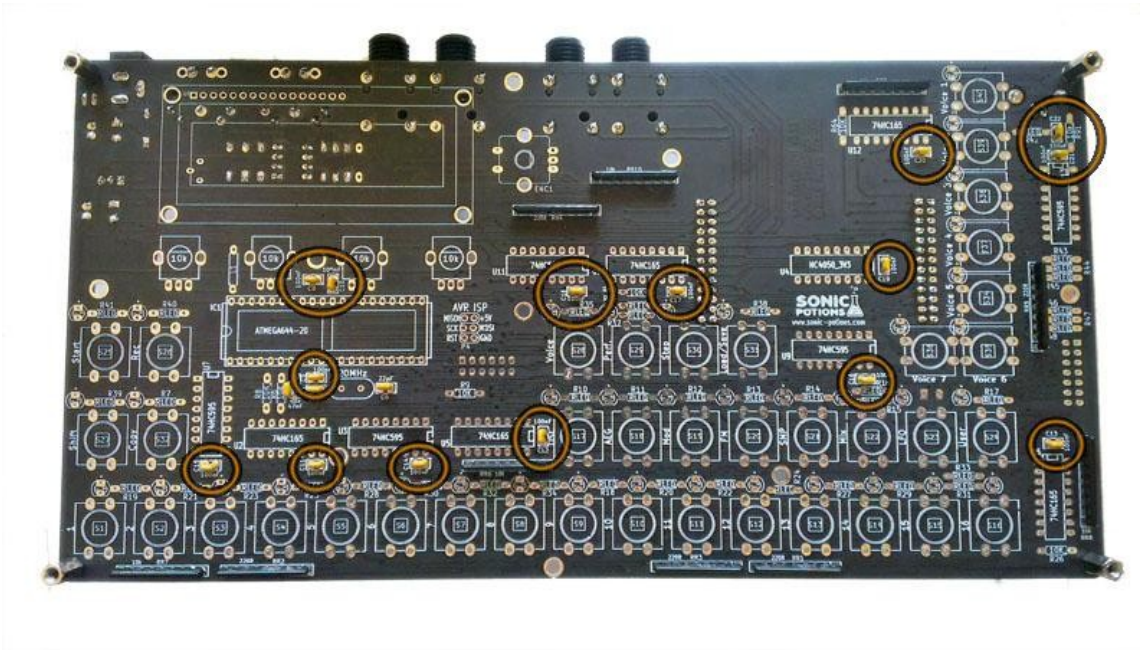


	<p>Ceramic 100nF</p> <p>Function: decoupling caps</p>	<p>15</p>	<p>C8-C22</p>	
	<p>Ceramic 47nF</p> <p>Function: Ensures the power supply rail has stabilized before booting the AVR</p>	<p>1</p>	<p>C7</p>	
	<p>Ceramic 22pF</p> <p>Function: part of the 20MHz quartz oscillator</p>	<p>2</p>	<p>C5, C6</p>	

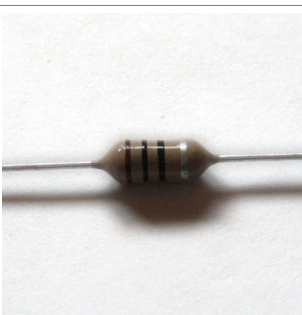
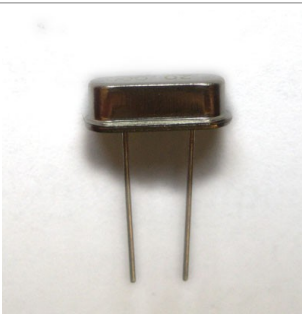
- Solder the 47nF and the 22pF capacitors next to the atmega644 footprint.



- The 100nF caps are next.

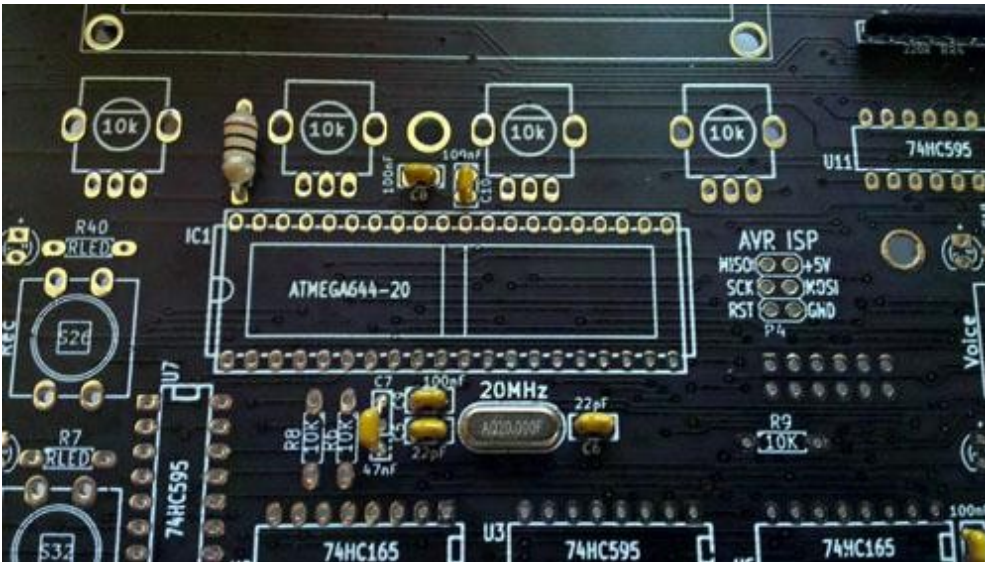


Step 9 – Quartz and Inductance

Image	Description	Amount	Part nr.	Notes
	Inductance 10uH Function: part of an LC-filter to cancel noise in the ADC for the potentiometers	1	L1	
	Quartz 20MHz Function: generates the clock for the AVR	1	X1	

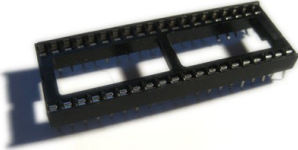
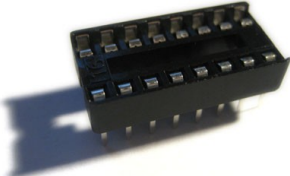
- Solder the quartz and the inductance. Both parts are non polarized.





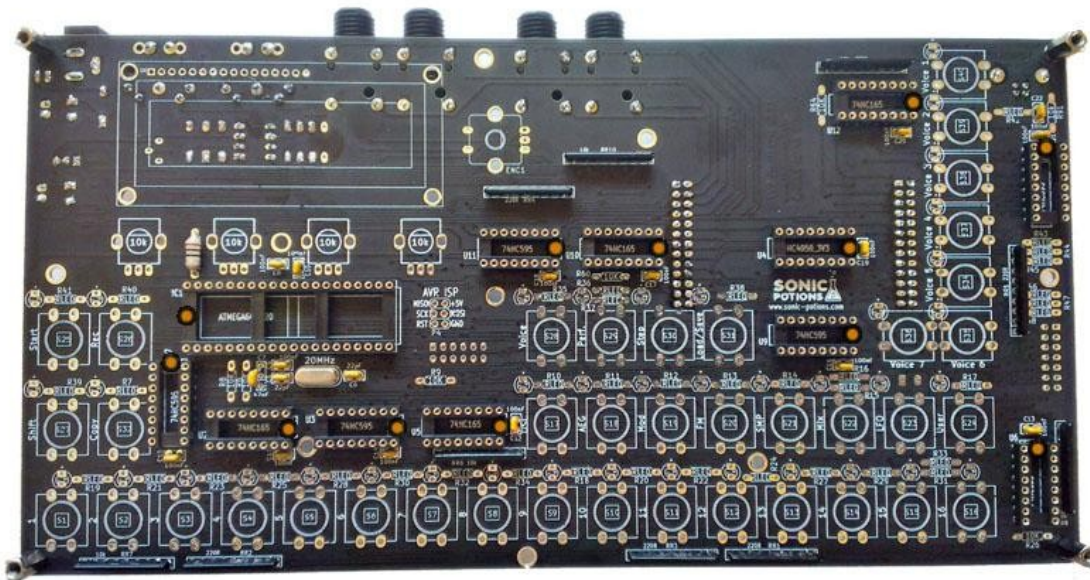
Step 10 – IC sockets

Now it's time to solder all IC sockets.

Image	Description	Amount	Part nr.	Notes
	IC socket DIP40	1	IC1	Notch has to be the same direction as the print on the PCB
	IC socket DIP16	11	U2-U7, U9-U13	Notch has to be the same direction as the print on the PCB

- **Attention!** The notch on the DIP40 socket points in a different direction than on the other sockets





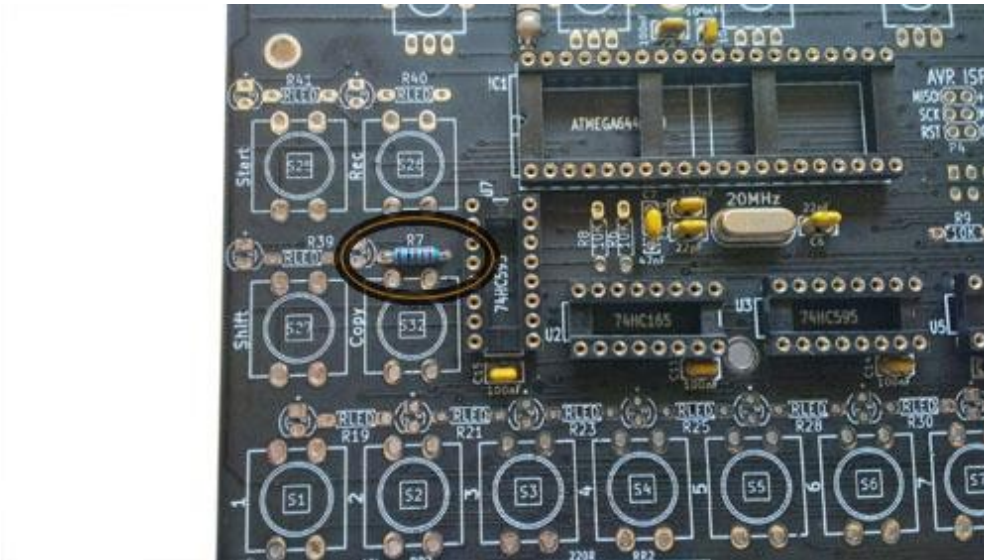
The notch direction is marked with a dot on this image

Step 11 – Resistors

Image	Description	Amount	Part nr.	Notes
<p>10k</p>	10k resistor	7	R6, R8, R9, R26, R60, R64, R91	
<p>221</p>	221 ohm LED resistor Function: current limiting resistor for the copy LED	1	R7	R7 is a current limiting resistor for the copy LED. If you are using other LEDs than provided with the kit, you have to change its value accordingly.

- Solder R7. It is marked as "R_LED" on the screenprint. The other "R_LED" positions are left blank.






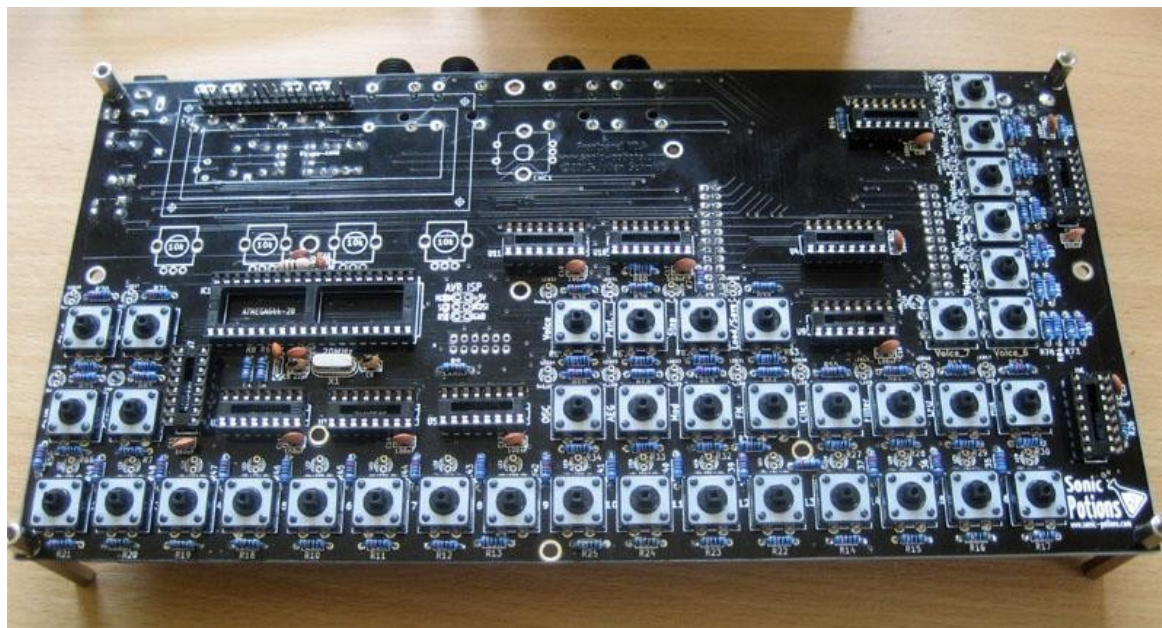
- Solder the 10k resistors



Step 12 – Switches

Image	Description	Amount	Part nr.	Notes
	TL1100 tactile switch	39	S1-S32, S34-S40	

- Put all the switches on the PCB. They should snap in and stay in place. Be sure that all switches lay flat on the PCB before soldering.



- Be sure to double check if you soldered all switches. It's very easy to forget some pins.



Step 13 – LEDs

!!! Please notice !!!

Starting 2020 we don't provide lightpipes with the metal cases per default anymore. LEDs will be mounted flush with the case. **Before soldering, please mount the PCB with the unsoldered LEDs inside the case, so you can level them with the front.** If you intend to get a case at a later date, separate lightpipe packs are available in the shop.

If you don't use the erica metal case you can solder the LEDs flat to the board.

Here is a picture how it should look:

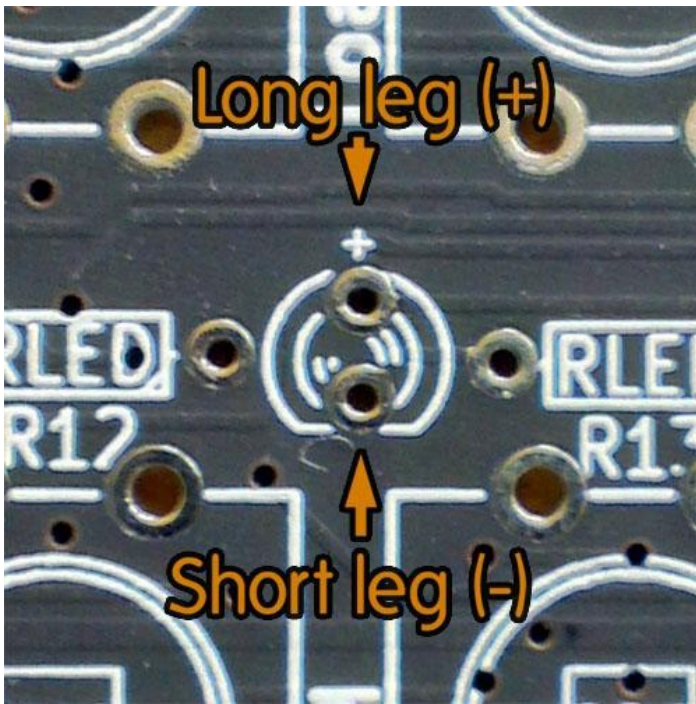


The LEDs are **polarized parts!** The longer leg is the positive (+), the shorter the negative (-) side.

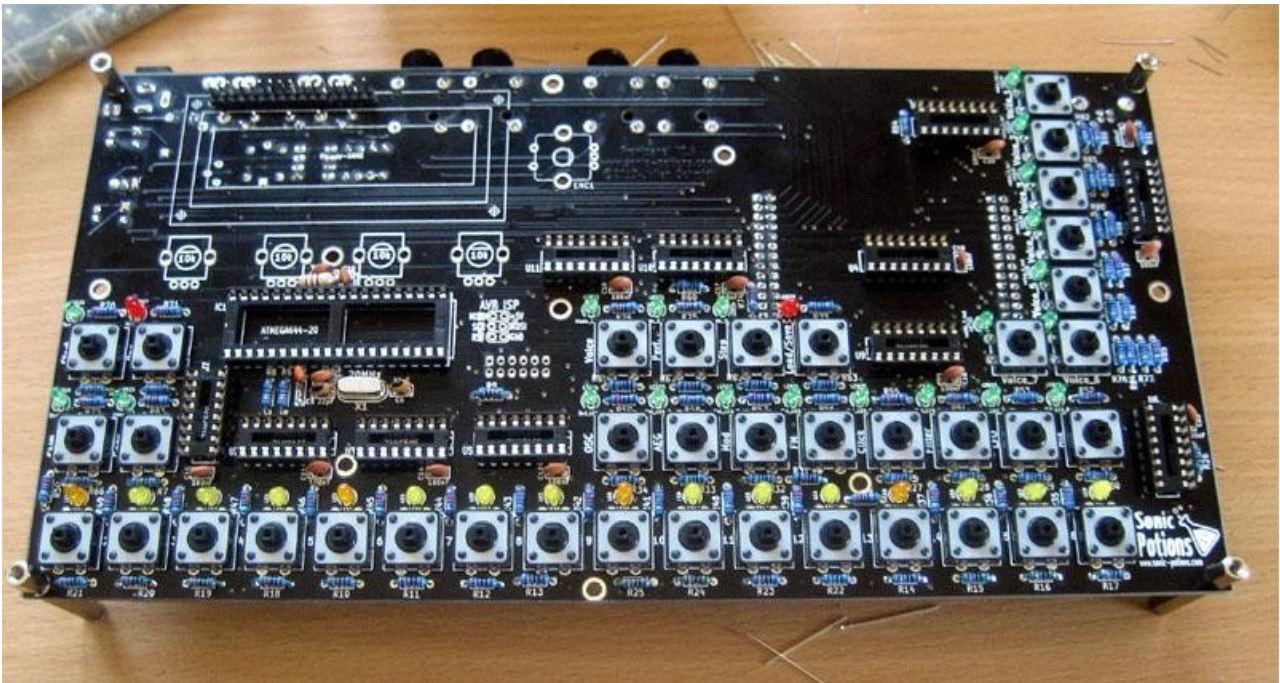
Image	Description	Amount	Part nr.	Notes
	Green LED	21		polarized
	Red LED	2		polarized
	Orange LED	4		polarized
	Yellow LED	12		polarized

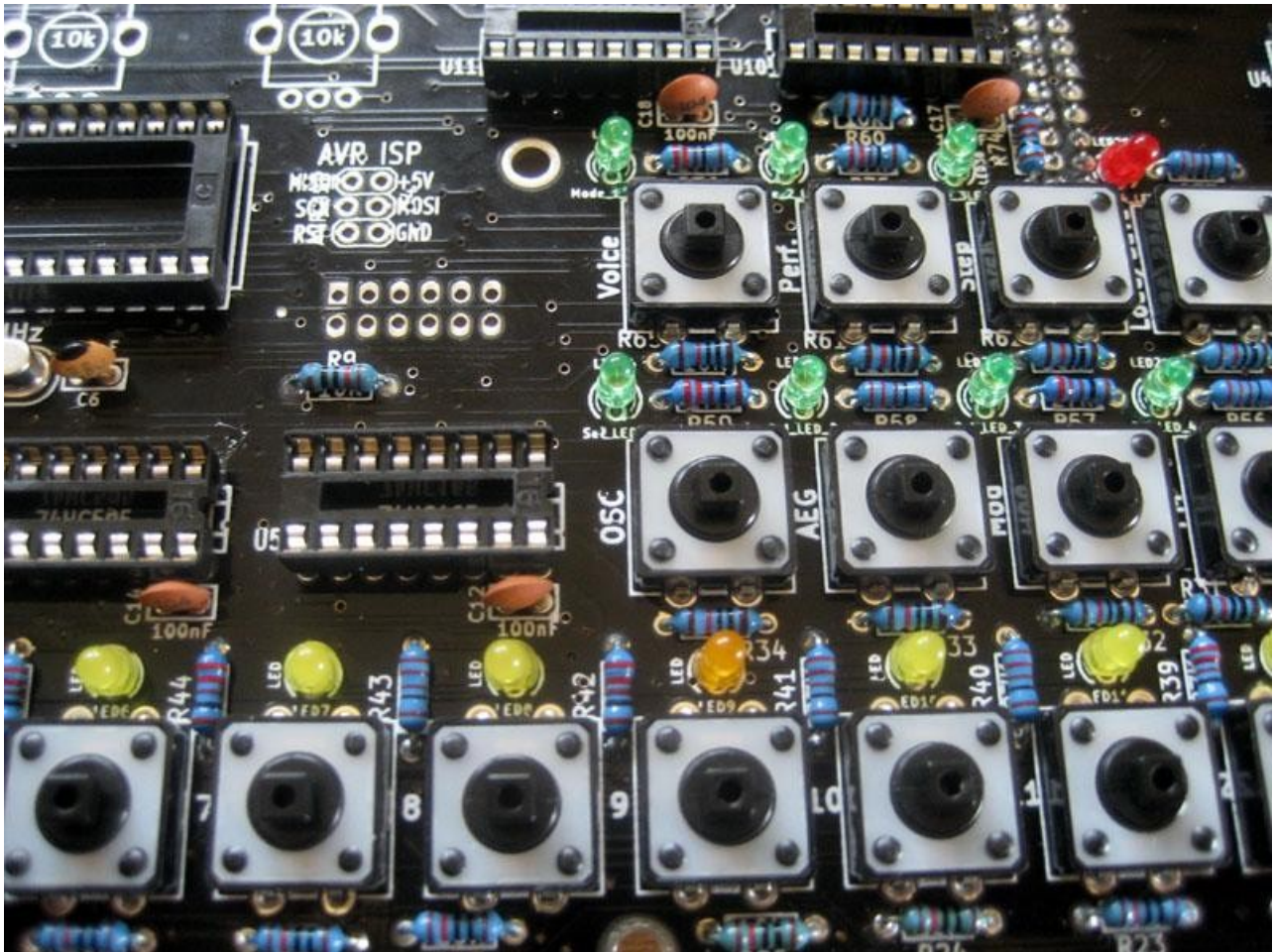
- For all LEDs on the board, the longer leg has to be inserted into the upper hole.





First put all LEDs into the PCB **but do not solder them yet!**





Then mount the PCB into the enclosure, turn it upside down, and put the LEDs into the holes in the enclosure.

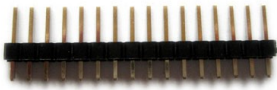
You can use a piece of cardboard or plastic to help get them flush with the case.

Now **solder only 1 leg per LED** to keep them in place. This way you can still adjust the height after soldering them. When all LEDs have the right height and are soldered in place, solder the 2nd leg.



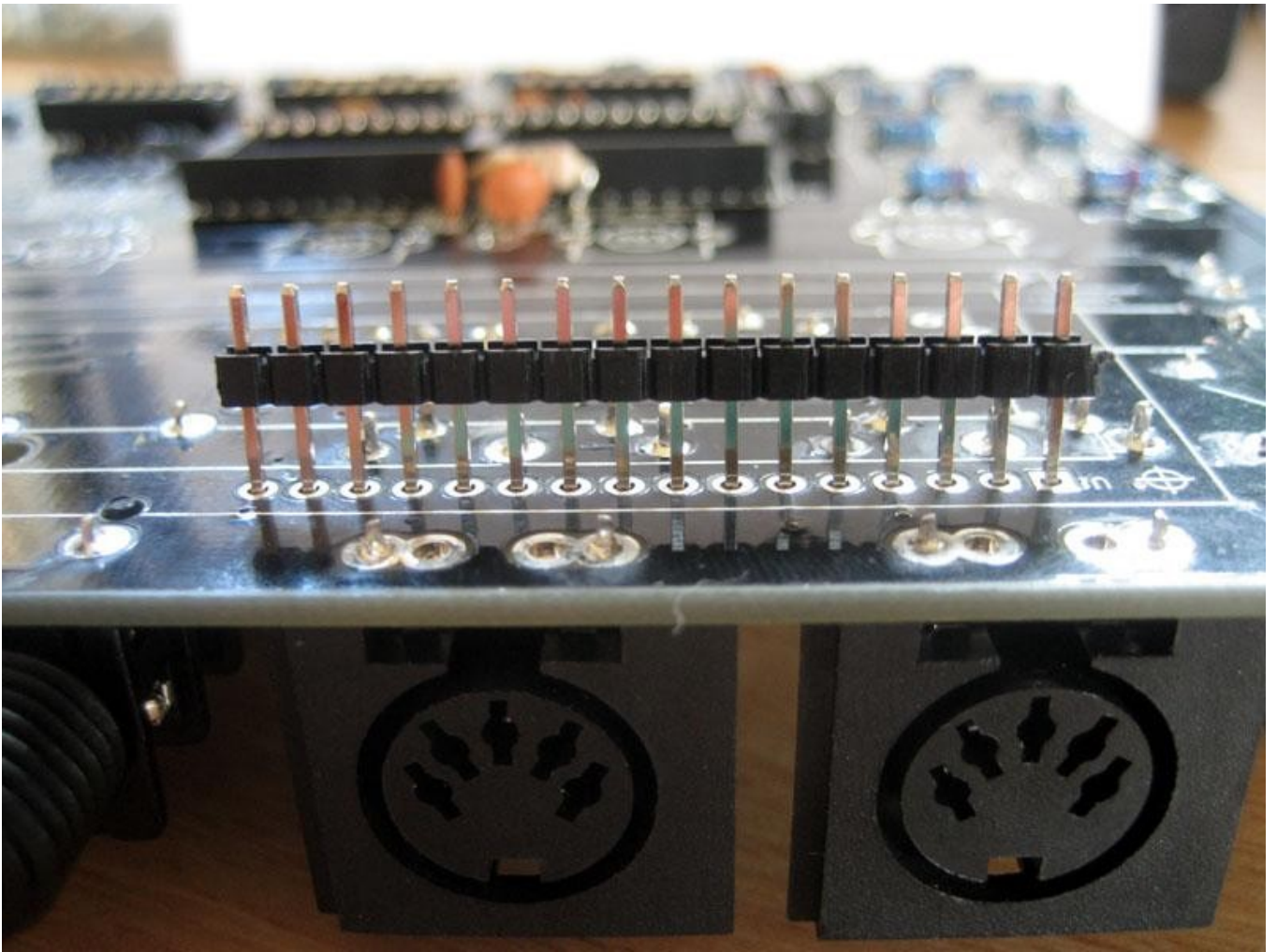
Step 14 – Display connector

This part is a little bit tricky. You already soldered the MIDI jacks to the bottom of the PCB in step 6. Since they are blocking the way, you can not solder the display connector like you normally would. You have to solder the pin array from the top.

Image	Description	Amount	Part nr.	Notes
	Pin header male, 1x16, 2.54mm	1	L1	

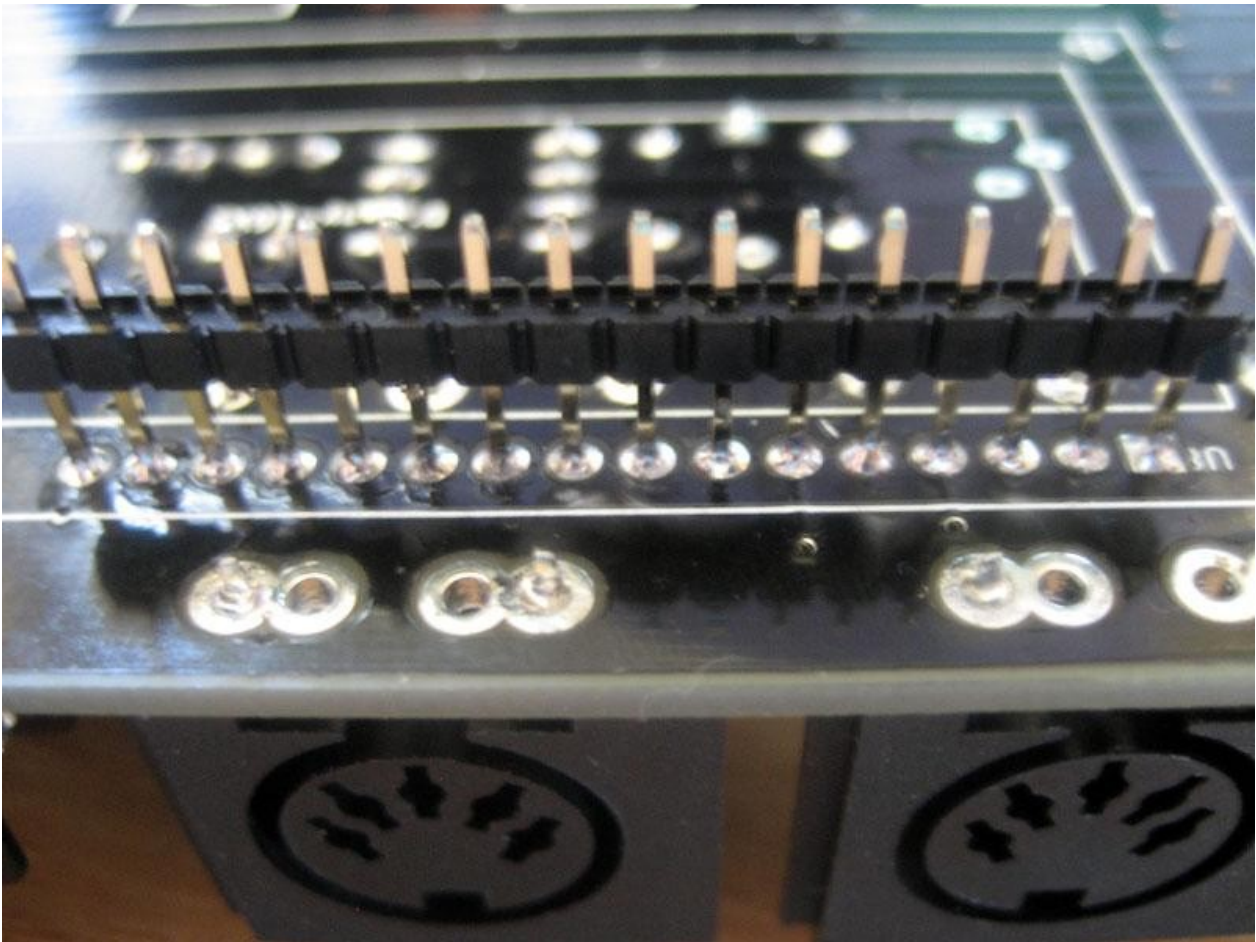
- Insert the pin array upside down. That means the longer side of the pins goes into the holes on the PCB. Since the MIDI jacks are right below, you won't be able to insert them very deep.





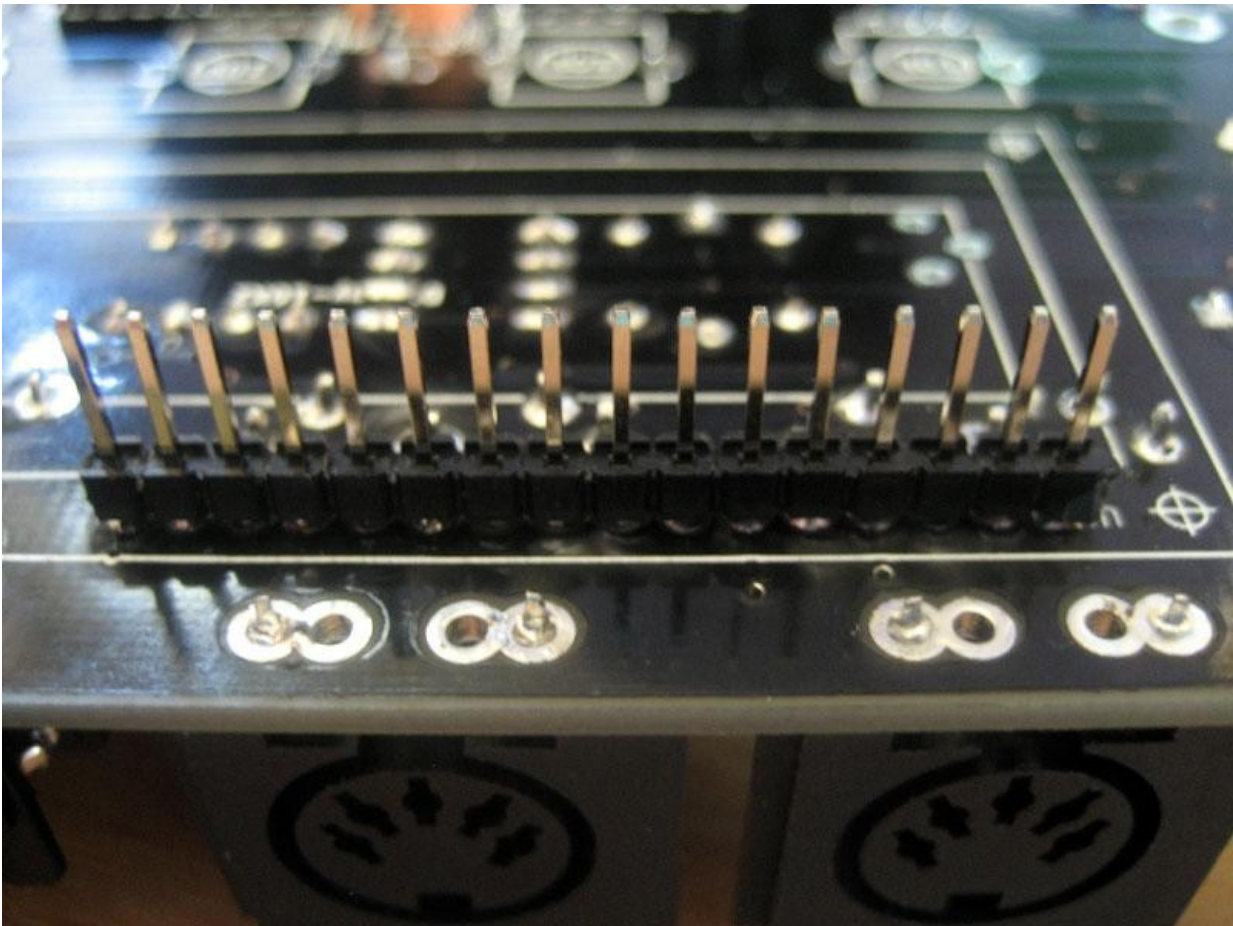
- Now you have a gap between the PCB and the black plastic thingy holding the pin array together. Here you need to apply your solder. Solder a single pin first, then adjust the pin array to point straight up and solder the rest of the pins with as little solder as possible.





- Take a screwdriver or similar and push the black plastic thingy down in the direction of the PCB. You cannot push it down in one go. You have to push alternating left and right to slowly move it down.







- If you took too much solder to push the plastic part down, either try to remove it using a solder sucker/wick, or remove the plastic part completely.

Step 15 – Encoder and Potentiometers

Add the 4 potentiometers and the encoder to the PCB.

Image	Description	Amount	Part nr.	Notes
	Encoder + switch 24 clicks, 25mm shaft	1	ENC1	



	10k potentiometer, linear	4		
---	---------------------------	---	--	--

- Solder one pin first and make sure the parts are flat on the PCB. Then solder the rest of the pins.



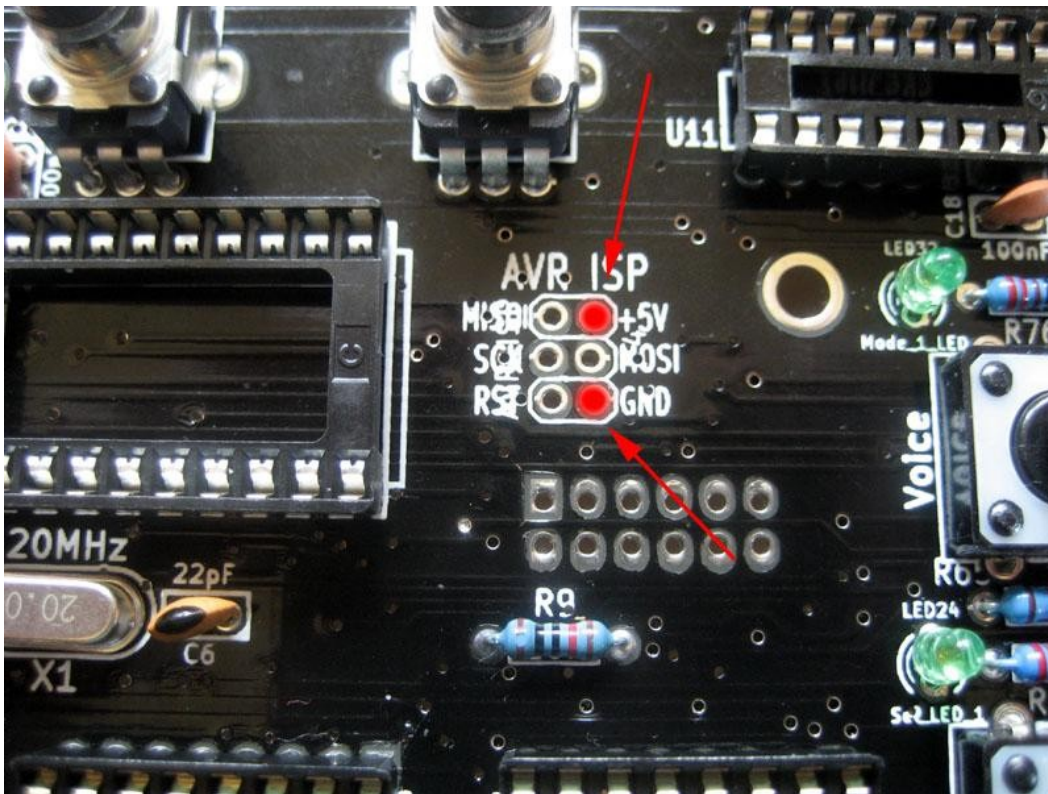
Step 16 – The end is near! A last voltage test

Before inserting all the ICs, we will make sure the voltage is correct and nothing is shorted on the power rails.

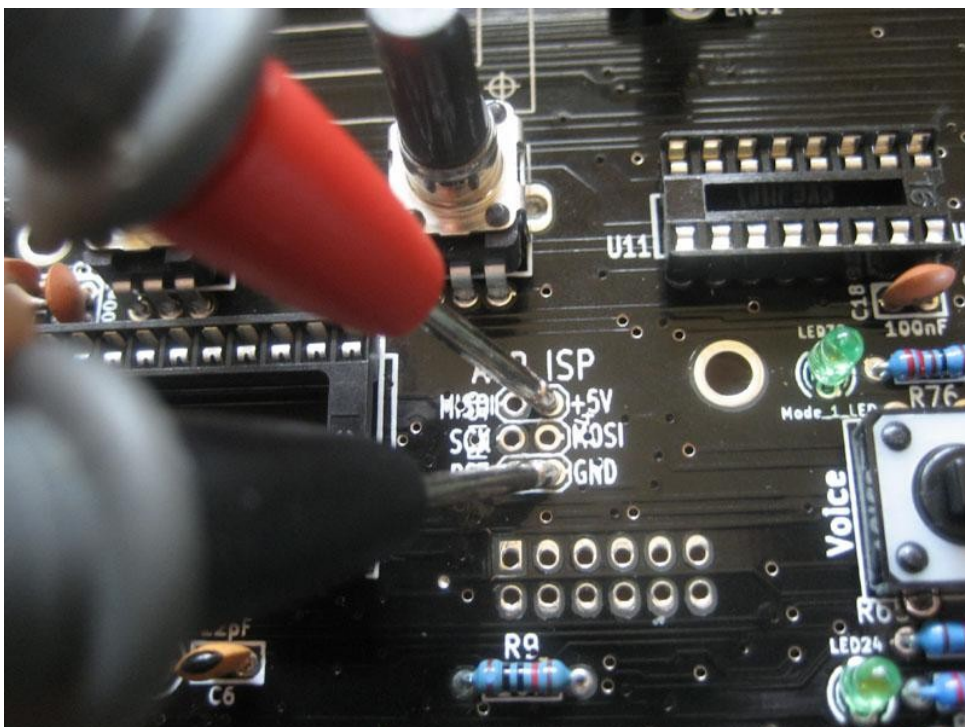
- If you have a multimeter with Continuity tester (it makes *beep* when the 2 probes touch) check that there is no continuity between the two marked points. A short beep in the beginning of the measurement is ok. That's caused by the charging capacitors.

If you have continuity between those points you have to recheck your soldering! It means you have a short between the 5V and the GND rails that could **damage your synth** and power supply when both are connected. **DO NOT CONTINUE UNTIL THERE IS NO CONTINUITY BETWEEN THOSE 2 PINS!**

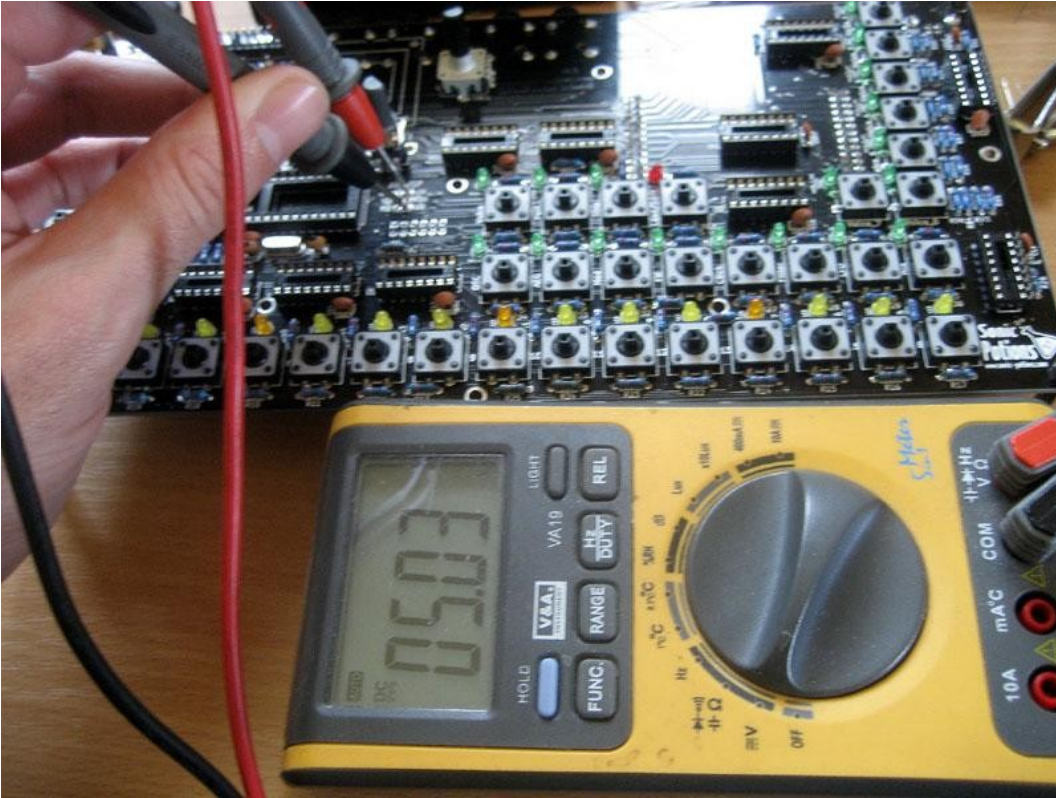




- Attach power supply to the DC jack
- Measure voltage between the 2 points




- You should read around 5V

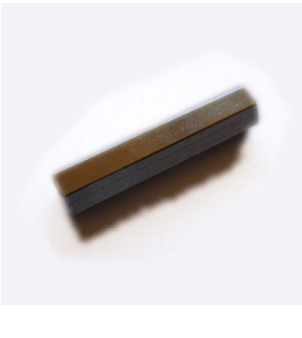


- Remove power supply

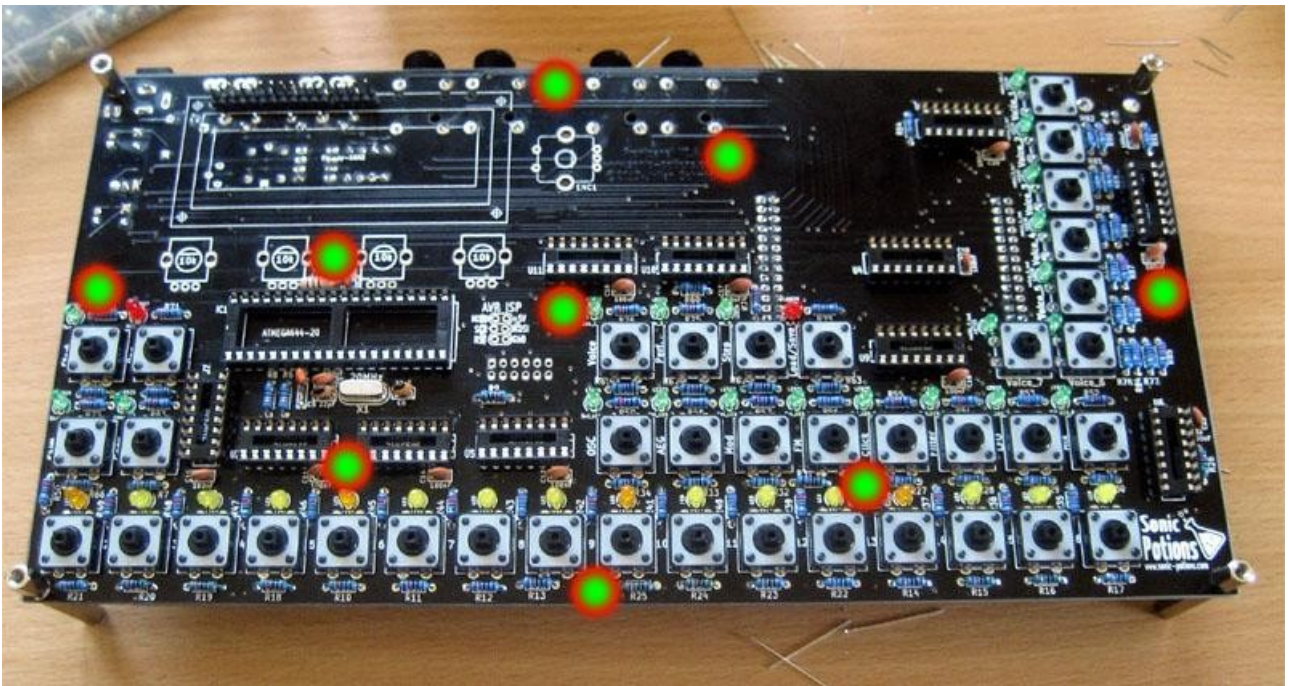
Step 17 – Attach the standoffs

Image	Description	Amount	Part nr.	Notes
	Standoff 10mm	4		



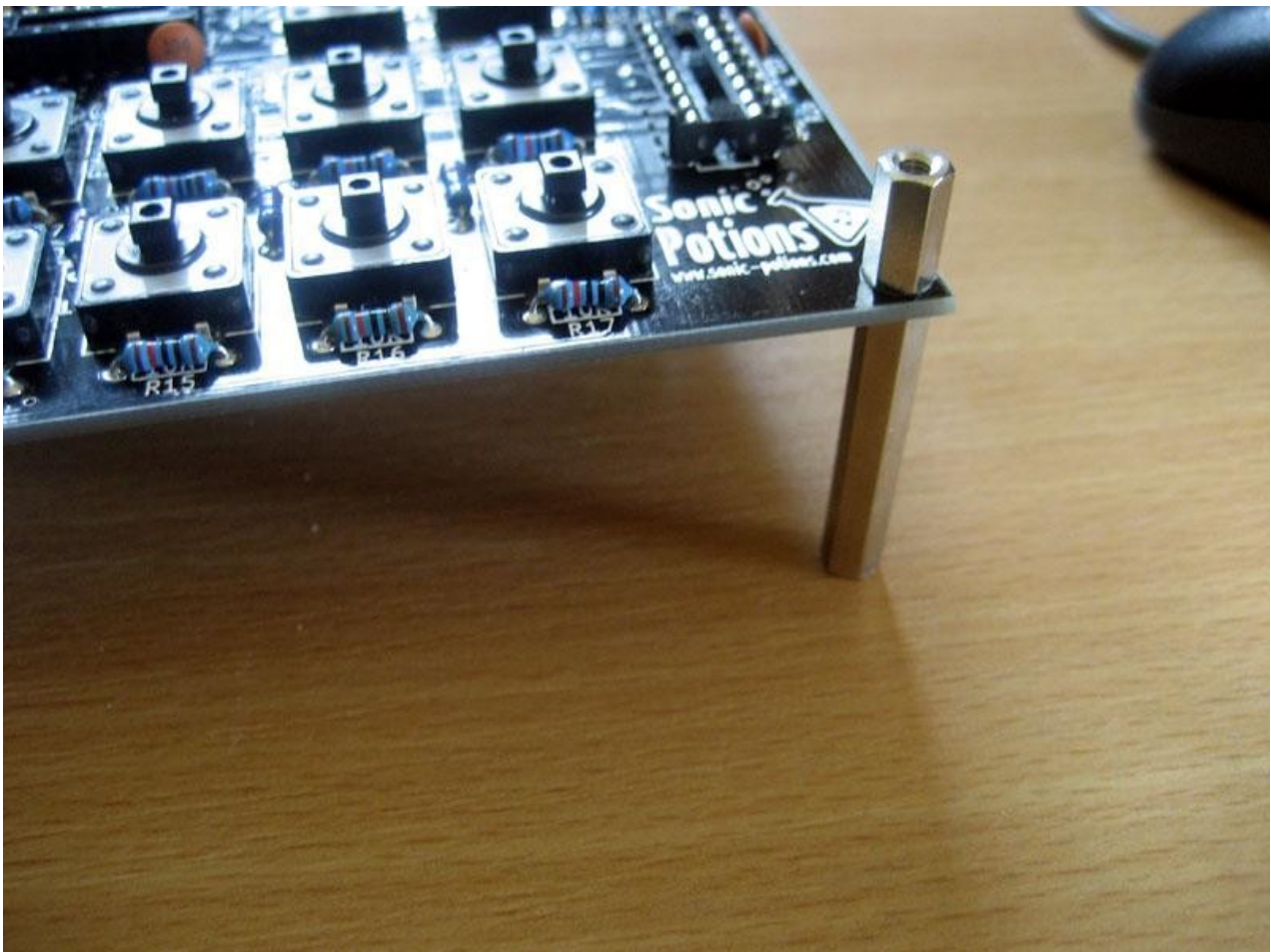
	Standoff 30mm	13		
	M3 screw	9		

- Add the standoffs to the board. A M3 screw on top of the board, a 30mm spacer on the bottom. The 9 positions are marked in the following picture.



- If you have not attached the 4 standoffs in the corners of the board, do it now. The 10mm spacers on top, the 30mm standoffs on the bottom.






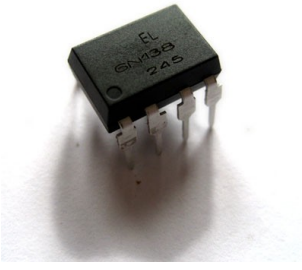
Step 18 – Insert all ICs

The ICs are polarized. One side of the ICs is marked with a notch. When inserting the ICs, line up the notch on the IC with the notch of the socket/silkscreen. On the upper side of the PCB the notch of all ICs is either pointed to the right side of the board or up. Only the atmega is faced to the left. On the bottom the notch/dot should face away from the MIDI jacks.

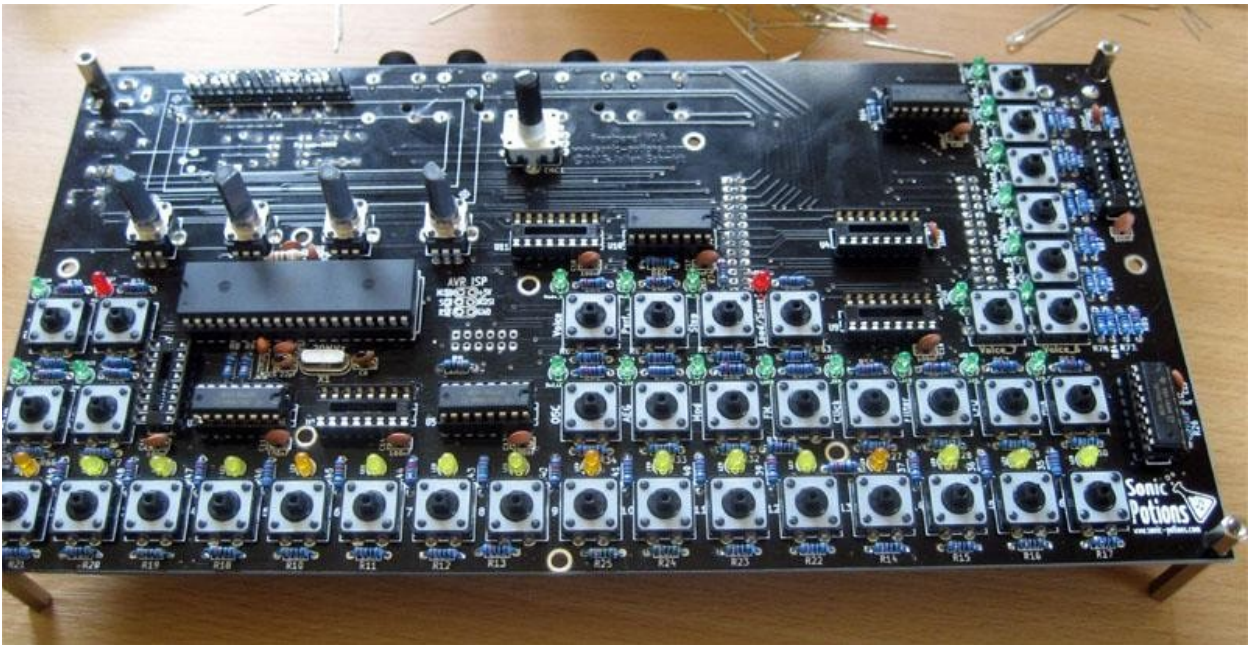


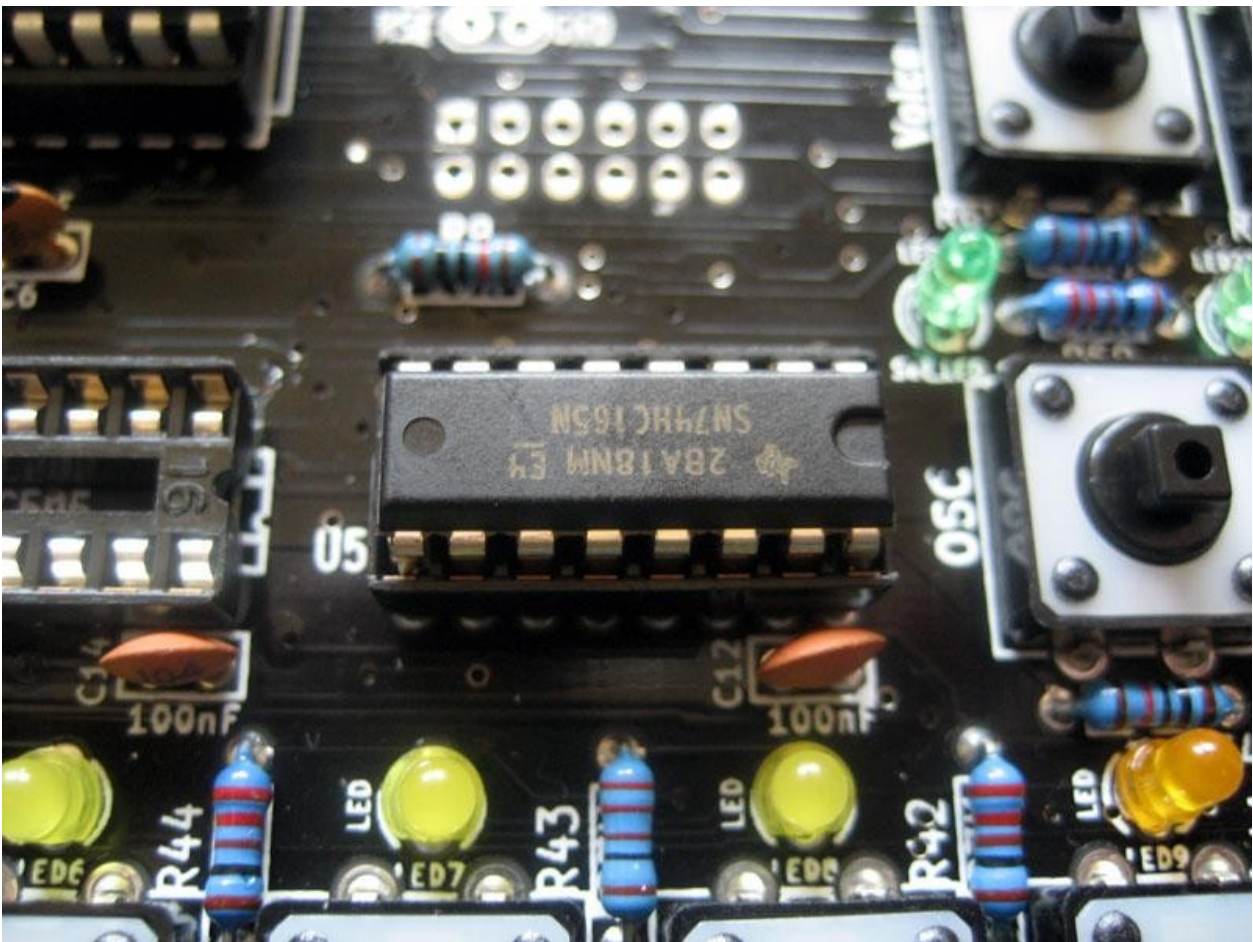
Image	Description	Amount	Part nr.	Notes
	<p>Atmega644p, 20MHz</p> <p>Function: microcontroller to run the user interface software</p>	1	IC1	Notch faces other direction than all other ICs!
	<p>74HC4050 Hex buffer</p> <p>Function: used as voltage converter to reduce the 5V signals from the AVR to 3V3 signals for the SD card.</p>	1	U4	Must be the HC version!
	<p>74HC595</p> <p>Function: serial-in/ parallell-out shift register. Used to adress all LEDs from just a few AVR pins</p>	5	U3, U7, U9, U11, U13	
	<p>74HC165</p> <p>Function: parallell- in/serial-out shift register. Used to read all buttons with just a few AVR pins</p>	5	U2, U5, U6, U10, U12	



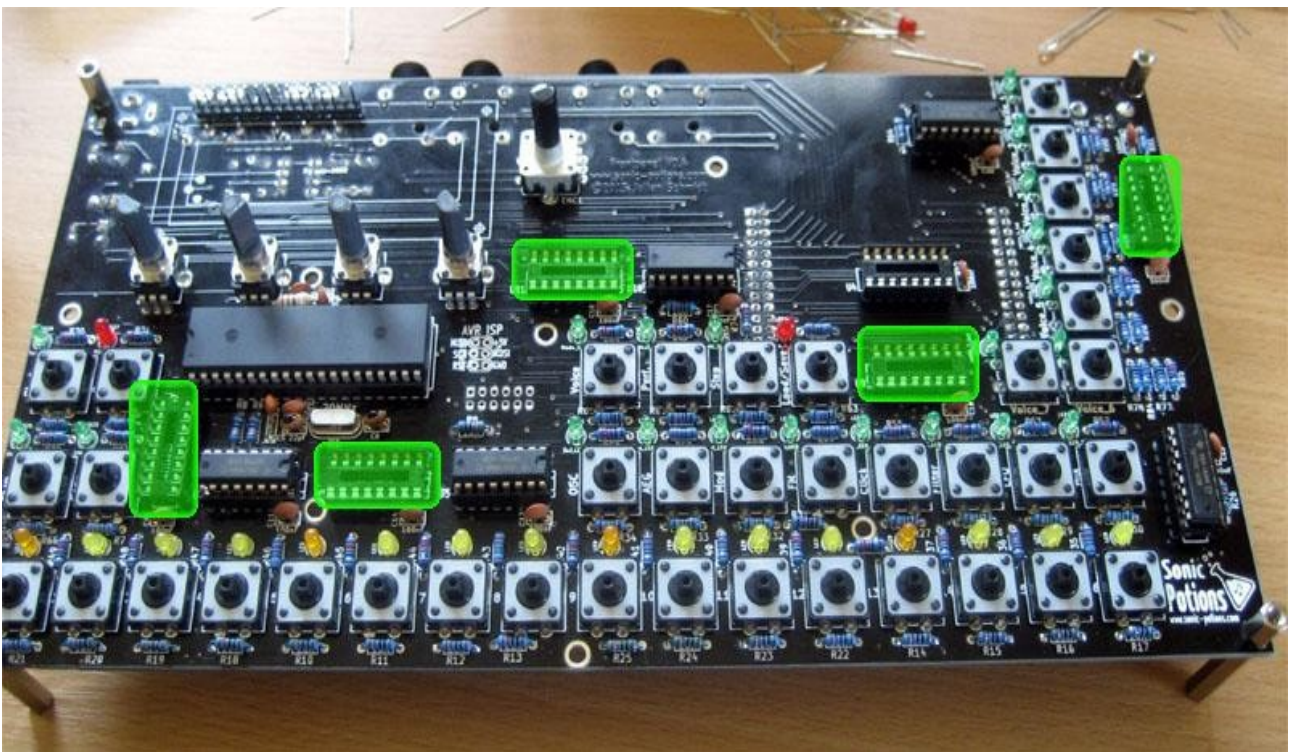
	<p>6N138</p> <p>Function: optocoupler for the MIDI input. Keeps the GND of the different MIDI devices separate.</p>	<p>1</p>	<p>OK1</p>	<p>Located on the bottom.</p> <p>Dot instead of notch.</p>
---	---	----------	------------	--

- Even if you have all standoffs attached, it is recommended to support the PCB from the back with your fingers while pushing the ICs in from the top. **Do not bend the PCB!**
- Insert the Atmega644p. Attention! The notch on this chip has to face to the left.
- Insert the 74hc165 ICs

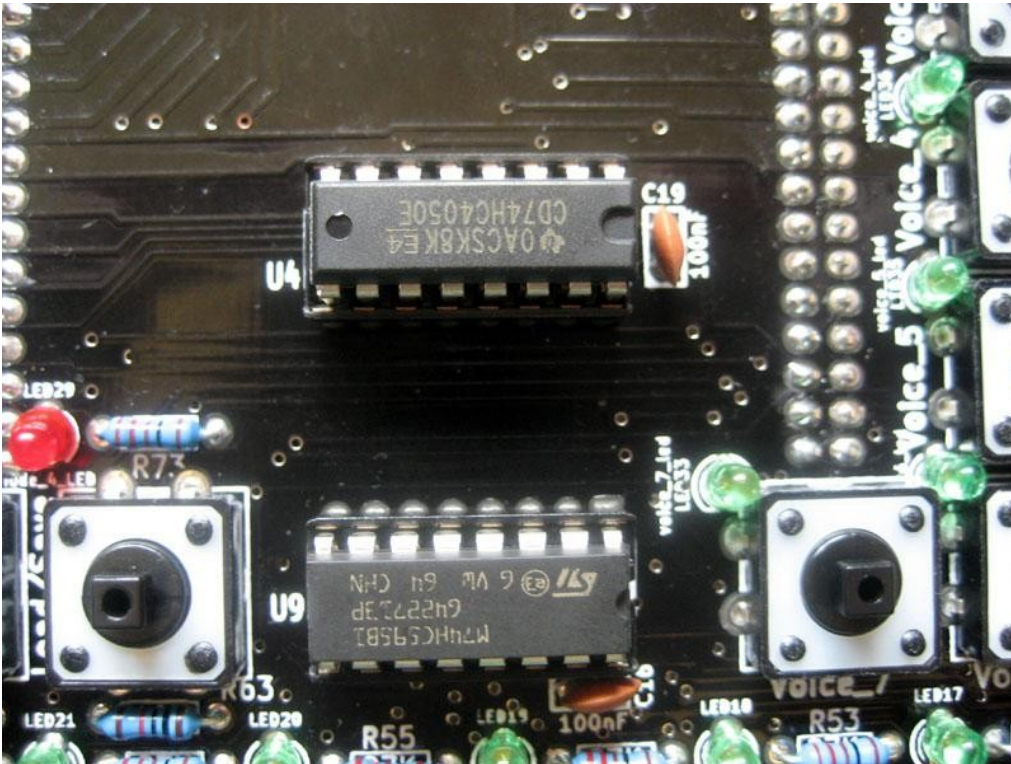




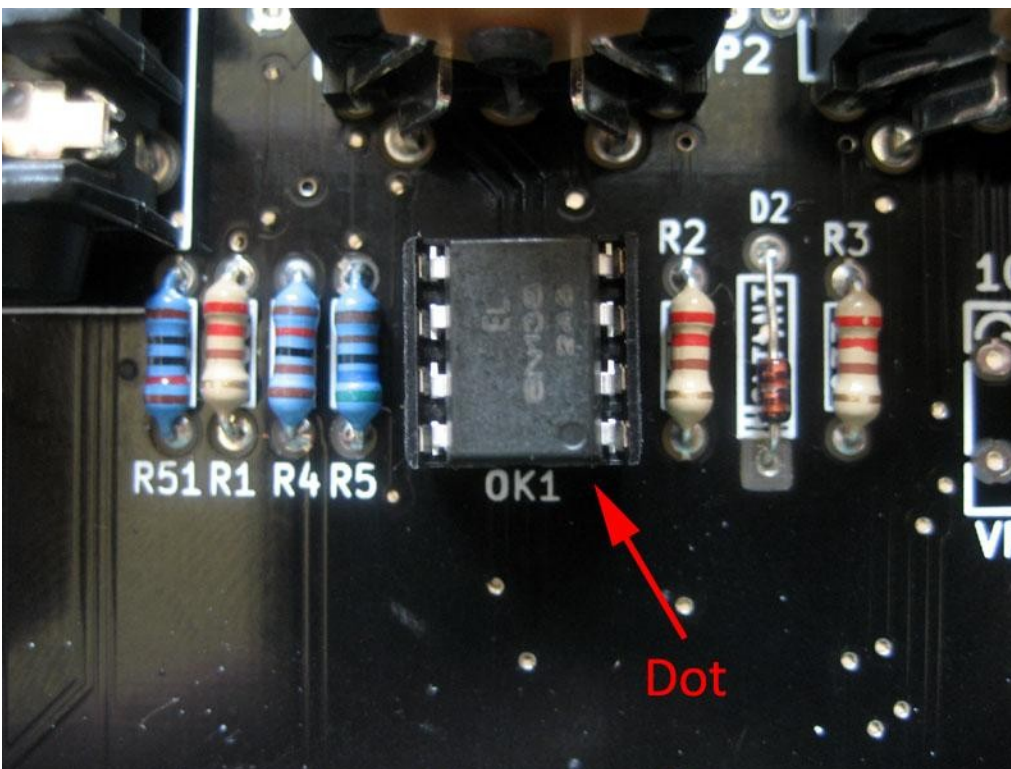
- Insert the 74hc595 chips



- Insert the 74HC4050



- Insert the 6n138 (bottom of the PCB)

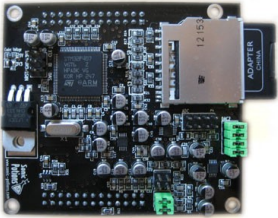


Step 19 – Powering up again

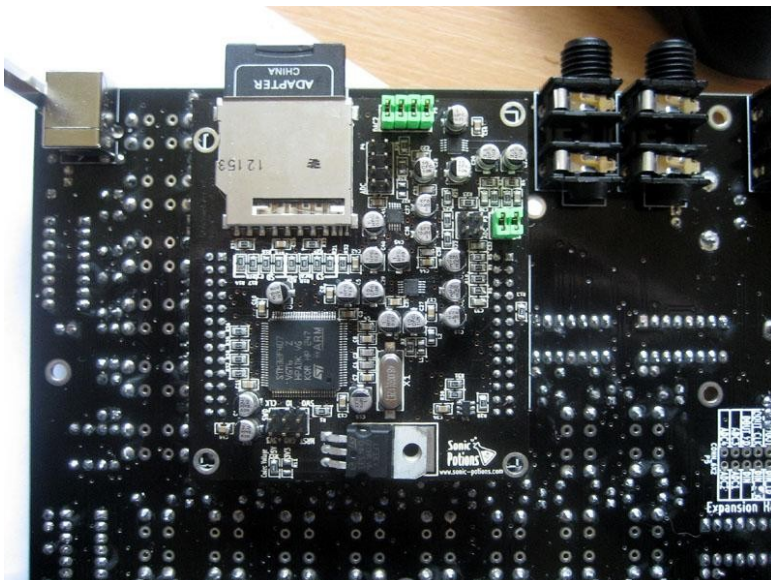
Just to be sure – power up the device after inserting all the ICs.
Touch the ICs with your fingers. None of them should get hot.
remove power supply.

Step 20 – Attach mainboard and test

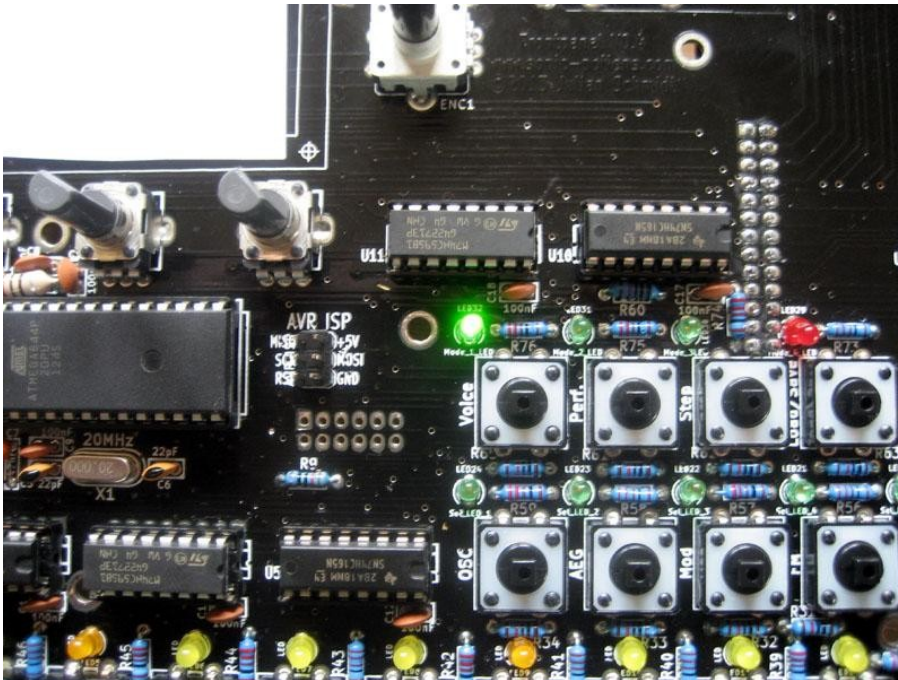
Now we are going to test if everything is working, so we can solder the display to the device.

Image	Description	Part nr.	Notes
	<p>STM32F4 audio mainboard</p> <p>Function: This board holds the STM32F4 ARM processor which runs all of the DSP code to generate the audio signals. There are also 2 audio codecs which provide the digital to analogue conversion.</p>		

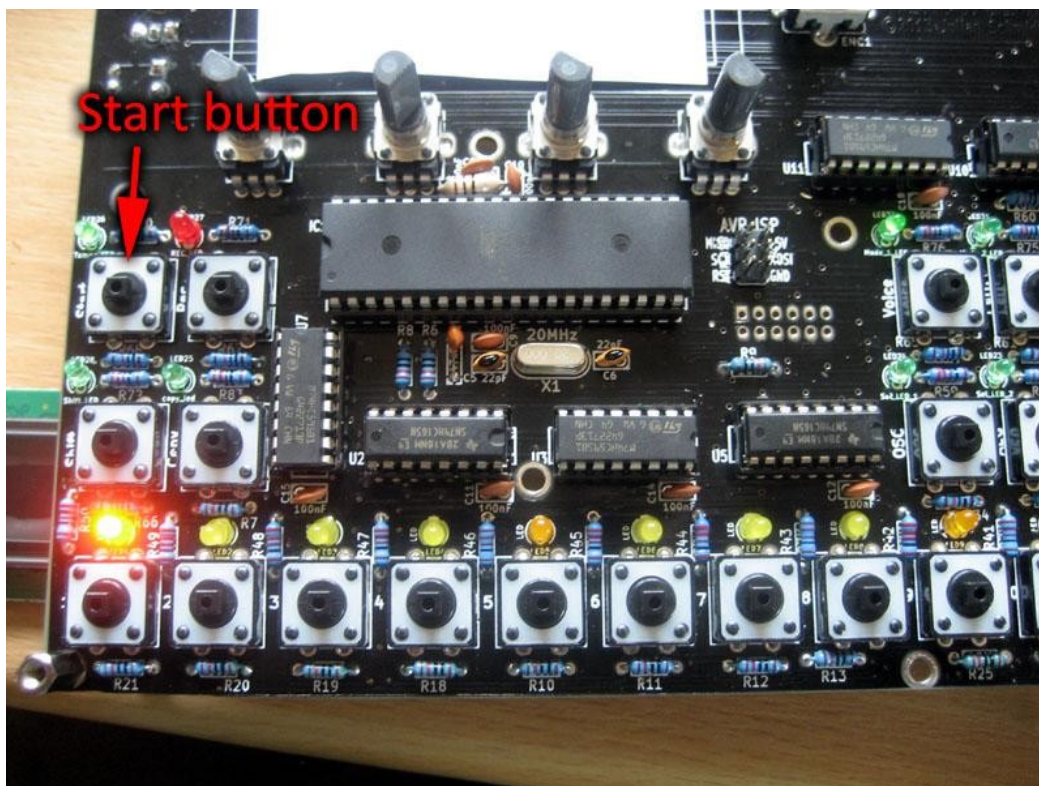
- Attach the mainboard to the 2 female connectors on the bottom of the PCB



- Power up the device
- Wait until the green 'voice mode' LED is lit.



- Press the 'start' button




- The LED next to the start button should start flashing
- The chaselight should start running

Troubleshooting: No blinkenlights :(

- Check chip orientation
- Do you have a programmed atmega? (you have if you bought a kit)
- Is the mainboard attached correctly

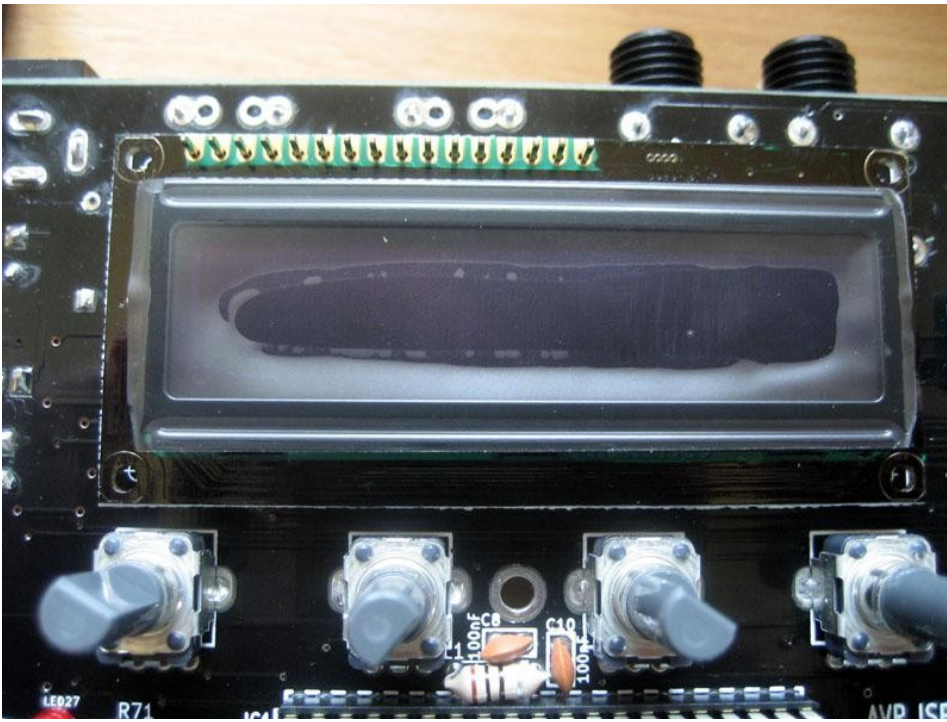
Step 21 – Solder Display

Now it's save to assume everything is ok and you can finally attach the display.

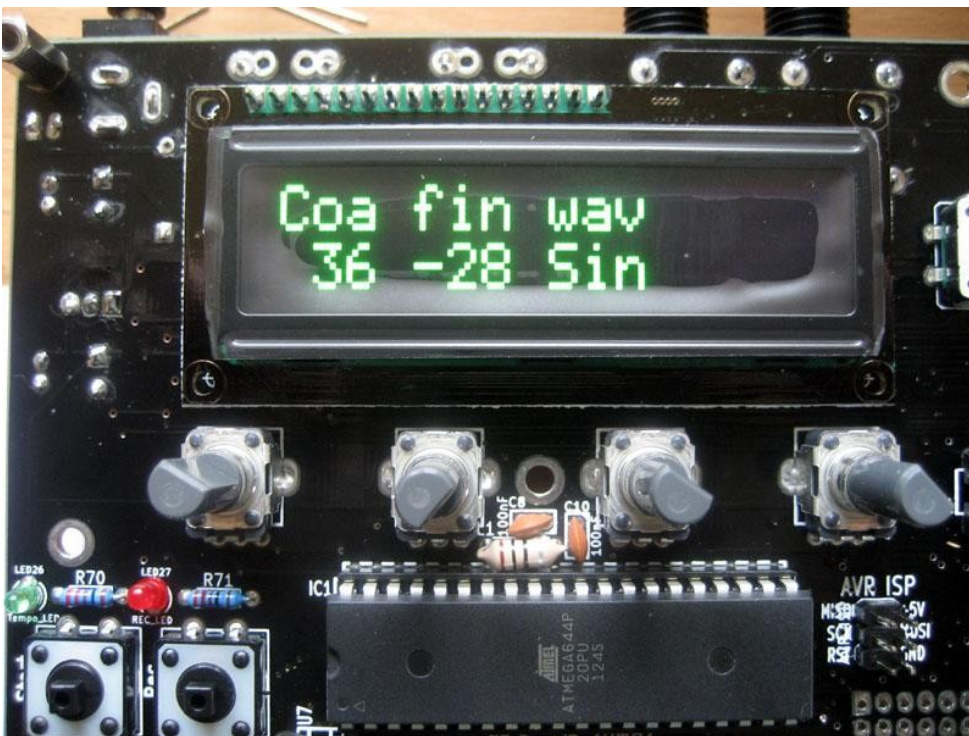
Image	Description	Part nr.	Notes
	2x16 OLED display		

- Put the display on the 1x16 pin header. Make sure to cut off the legs of all the parts below as short as possible, so that the display sits flat on the PCB. Then solder the display to the connector from the top.





- if you connect the power supply again, your display should show:



- **Only for LCD users:** adjust contrast trimmer if your display does not show anything



Step 22 – Button Caps

Image	Description	Part nr.	Notes
	19 x black cap		
	6 x grey cap		
	2 x red cap		



	<p>12 x white cap</p>		
	<p>1 x Plasticine glue (Blu Tack)</p> <p>(Added to the kit april 2014)</p>		

Make a thin roll out of the plasticine







Cut the roll into 39 little pieces (the so called "Huskerballs"), one for each button cap

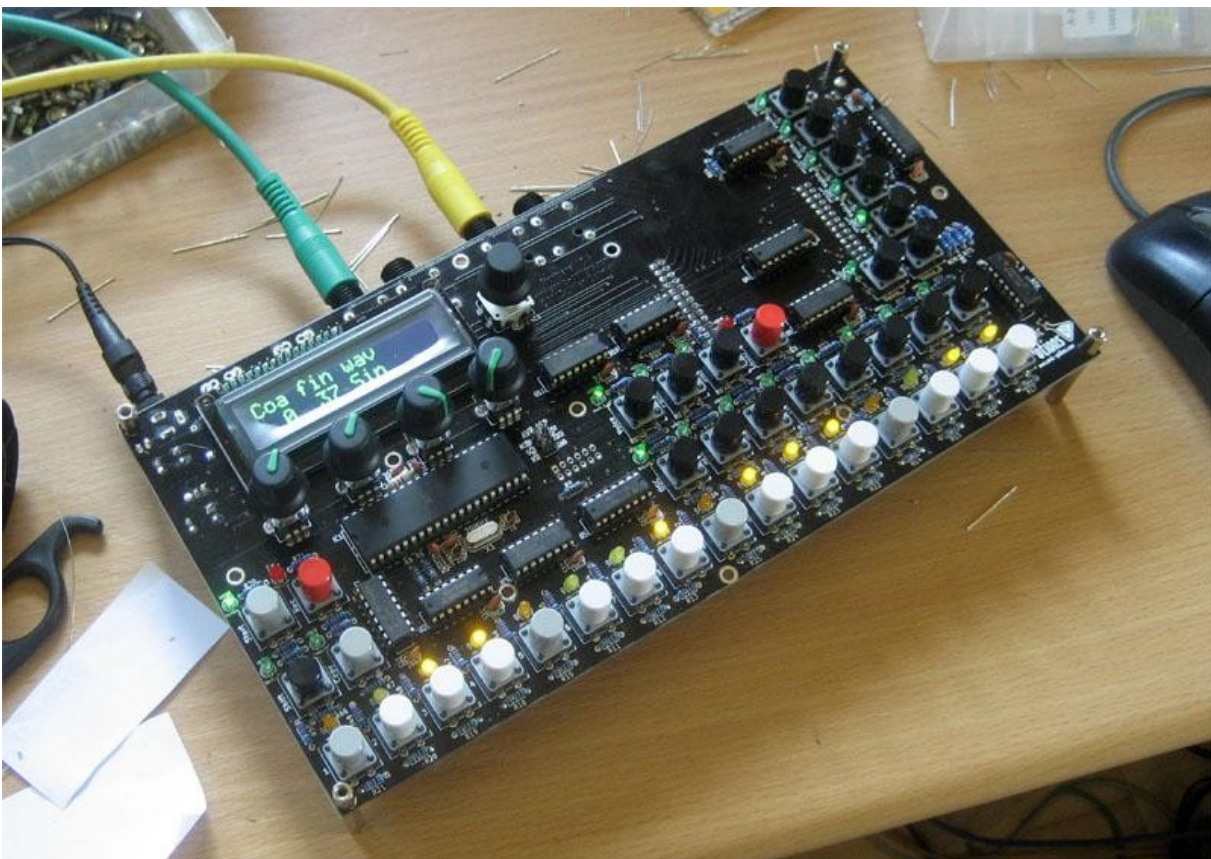


- Put one of the pieces on the bottom of every cap





- now put the caps on the buttons



Congratulations! You are now the proud owner of a Sonic Potions drumsynthesizer! I hope you enjoyed the build and have fun with your new synth!

Step 23 – SD Card Image

Before you start, please prepare your SD card.

- Download the factory SD card image from:
<http://sonic-potions.com/public/SdCardImage.zip>
- Unzip all files to the root folder of your (FAT32!) SD card

Quickstart Guide

The fastest way to get some beats out of your new box!

- Insert SD card and power up the synth.
- Press start button. The chaselight starts running.
- Use the 16 step buttons to activate some steps. Now you should hear something.
- Use the 7 voice select buttons to change the active track/voice.
- Again, use the 16 step buttons to activate some steps. The 2nd voice should play now as well.
- Press the red load/save mode button once. You are now in preset load mode.
- Use the encoder to select another preset. The sound should change as soon as you move the encoder.

