STEREOPING

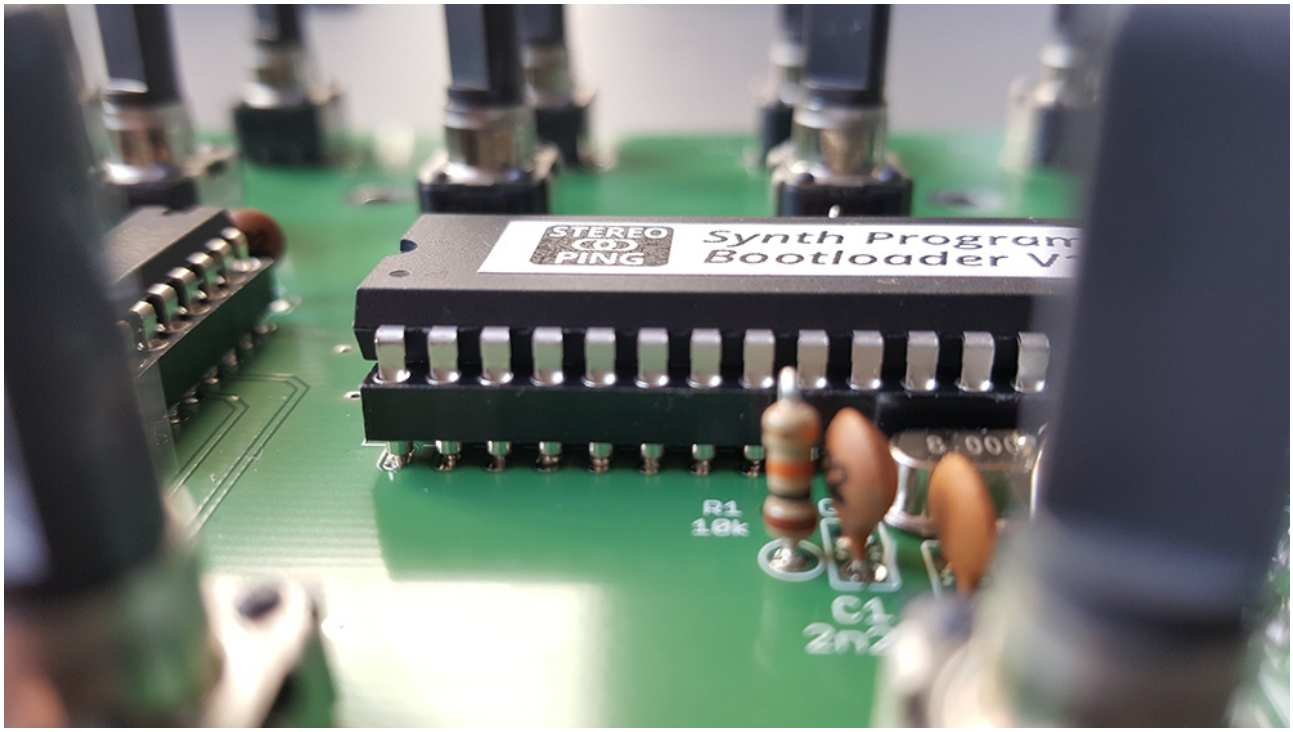
Synth Programmer
Building instructions V1.93

Power-Midi PCB V1.3

Main PCB V1.3

Display PCB V1.3

07/2022



1. Hello

Hi and thank you for having chosen the Stereoping Synth Programmer assembly kit. If you were looking for the user manual you are unfortunately wrong here, this is the building instructions for getting together the assembly kit. You can find the manual here
www.stereoping.com -> Products -> Synth Programmer-> Downloads.

If you experience any problems during building this kit, are coming up with a genious improvement feature, if you wish special photographs or simply are enoyed by something, even if you just want to write us that you LOVE our product - go ahead. We are looking forward to receiving your email to **hello@stereoping.com**

These building instructions came out quite large because we don't want you to make mistakes or experience any bad emotions while building the Synth Programmer. The kit and the instructions were developed very thoroughly and are verified multiple times. If you follow the instructions carefully, your Programmer will work perfect right from the beginning.

Please note: this DIY kit demands quite advances skills, we could not recommend it for beginners. Please have a thorough look at the chapters for the flatcable socket and the display on pages 14, 24 and 28. Both contain quite small solder pads which need to be soldered very proper. If you are unsure here, we'd recommend to go for the assembled programmer.

We can not be held responsible for any problems or damage caused by the building or usage of this assembly kit. Nor can we replace or repair PCBs, parts or complete kits, damaged or destoroyed by inappropriate building or usage.

2. Kit's content, hints and notes

The DIY kit contains everything to build a complete Synth Programmer:

- Case with already sticked faceplate in your edition
- Power-Midi circuit board (screwed with case bottom)
- Main PCB (screwed with case's upper half)
- Bag A with electronic parts for Power-Midi PCB and for case assembly
- Bag B with parts for main PCB
- Bag C with parts for the display PCB
- Bag D containing 45 pot knobs and 4 encoder knobsn, suitable for your edition
- Bag E with 45 pots
- Owner's manual

Tools you will need

For building the DIY kit you will need the following tools:

- Soldering iron or station, small tip (ca. 0.5 - 1mm)
- leaded or leadfree solder (works both)
- a side cutter to cut the wires from the parts after having them soldered onto the pcb
- philips screwdriver (PZ 1)
- normal screwdriver and 5 mm nut (simple pliers will also do) for display installation
- a multimeter to check the operation voltage before attaching the ICs

Before we start ...

Building the kit will take about 2-3 hours. Please take your time, enjoy the process of building your controller, concentrate on what you are doing. It happens quite fast to mix up two parts or solder one on the wrong side of the pcb. This manual describes the whole process very detailed. If you follow it thoroughly, your Programmer will work perfect immediately.

Important notes pointing to pitfalls are printed in red. If you are a professional and do not need this manual ... perfect. But please take a look at least onto these red passages.

If something goes wrong, is missing or unclear

... do not planic. Send us an email, we surely will help you.

Some hand soldering hints

Hand soldering is easy – if the parameters are fitting. Using a cheap 15W soldering iron, an oxygenized soldering tip and tin from the 70ies, the best professional can only produce junk. Before we start, some hopefully helpful hints about soldering:

- Solder marks are more often screwed up by to less heat than to much. If tin does not want to flow, it's most likely due to one (or a mix) of 3 reasons: a) not enough heat or b) not enough flux or c) bad tin.
  - Heat 'can' be archived by holding a weak soldering iron on the solder pad for some minutes. As a consequence it starts to stink, the PCB gets black and the electronic component you try to solder is dying by heat – whereas the pad still is not soldered. It would be better to use a strong soldering iron or – even better – a soldering station. A good soldermark does not take much longer than 3 seconds.
  - Flux is part of the soldering tin, makes the hot tin flow into the smallest corners and helps connecting the different parts (PCB, component, tin) to a homogenous, clean solder mark. Only fresh soldering tin contains flux as most of it fumes away when the tin is heated up. Therefore your solderings get much better by permanently providing (small amounts of) fresh tin during soldering. If you get to much tin on the solder mark, take some part away with a pump or desolder braid and add new tin again.
  - Soldering tin is a highly complex chemical product with lots of ingredients. It's the same as with other products: there is crap, there is quality and there is a lot in between – do not take the cheapest. For DIY it is legal to solder with leaded tin – that's what we also would advice to go for. Leaded tin is much easier to handle. If you want to solder lead free, try to get tin with some percent of silver (Ag for Argentum). It's a bit more expensive but you will definately have much more fun. All Photos in

this instructions show soldering with leadfree tin containing 3,8% silver.

- After at least 10 solder marks it's time cleaning the solder tip by dipping it into steel wool or clean it with a humid sponge. Afterwards the tip should be tinned again immediately. The tin protects the tip from getting oxygenized. Otherwize it will become black and can not fulfil it's function any more.
- Even though we said a good solder mark hardly takes longer than 3 seconds – take your time. Solder must flow. To be precise: it must flow to the other side of the PCB. Solder tin only flows on surfaces being hotter than it's melting point. If you remove the tip to early, the tin might get cold again before it was pulled thru the via onto the solder pad on the other side of the PCB. The chapter *Power-Midi PCB - Midi jacks / power jack* shows some good photos where you can see how beautiful the tin crawled onto the other side – leadfree tin by the way.
- A solder mark must not look like a squere – in other words: as soon as you think you got the idea how soldering works, try to start using tin economically. The profile of a good soldermark looks more like a triangle, maybe even with concave sides.
- ... if you got further questionbs you are welcome to drop us an email.

Health advice

- While soldering, toxic smoke will come up which should not be breathed in, the smoke is also not good for your eyes. First action is to open a window. Sound funny but: if you do not have a solder smoke extraction, try to blow slowly in the direction where you solder. The toxic smoke logically will be blown away from you and you can not breathe it in in high concentration. Breathing in is only allowed in soldering breaks :-)
- Solder tin contains lead which is toxic. Even leadfree solder contains highly concentrated chemical ingredients. It is highly adviced not to eat or dring while soldering. After your solder session, the hands should be washed thoroughly.

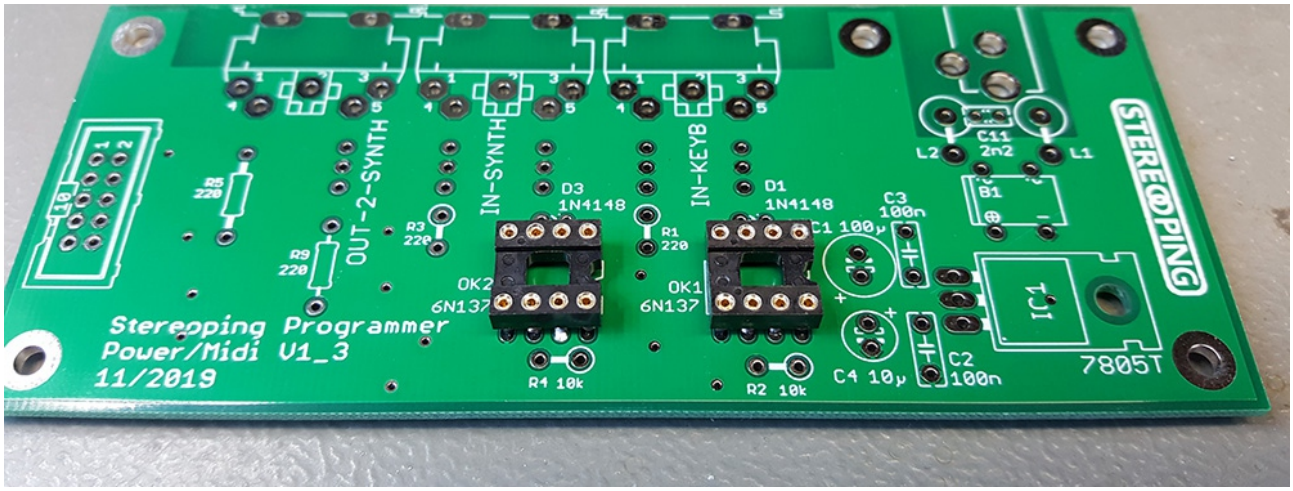
3. Power-Midi board – Bag A

As a warm up we start with the small Power-Midi PCB. All parts for this board are contained in bag A. At the end of this chapter there is a nice photo of the finished PCB which you can compare to your progress.

IC-sockets

Note: if you are skilled soldering veteran you can as well leave out the sockets and solder the ICs directly onto the PCB. The optocouplers practically never fail and the electrical conductance as well as EMI properties of soldered parts are better than using sockets. If you are unsure, go for the sockets. The electronic component will not be exposed any heat and if you manage to place it the wrong way it's easy to turn it around.

We start with the sockets because they are the flatest parts on the board. All IC sockets got a little notch on one side. This must of course fit the silkscreen on the PCB:

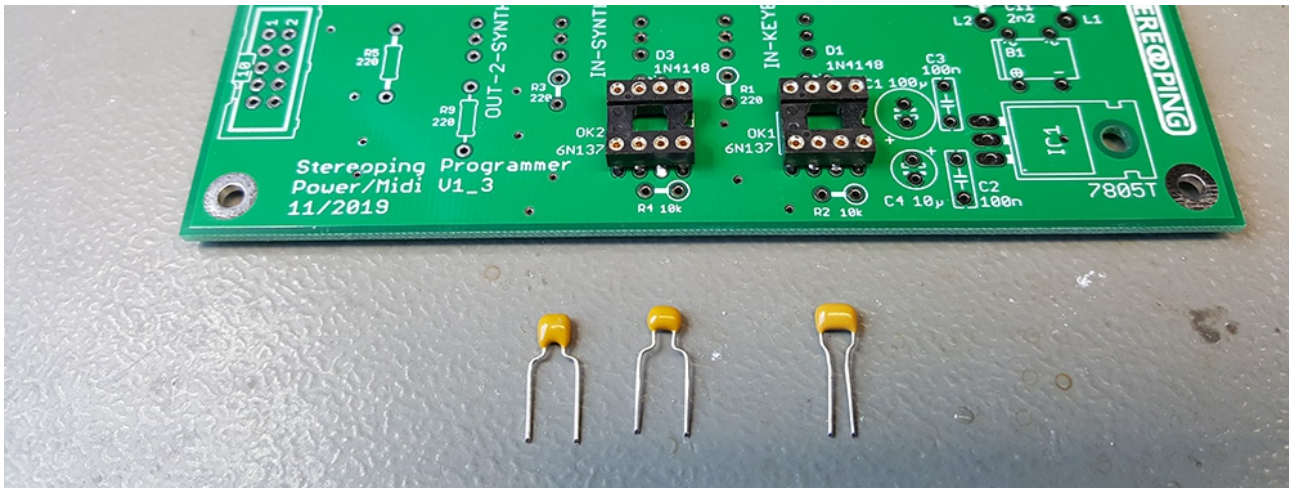


The notch indicates where to put leg Nr. 1 of the IC – leg 1 is indicated by a notch in the case or a point printed together with the device name. Putting an IC into the socket the wrong way sometimes (not always) can damage the part. There is no reason to worry here for the optocouplers.

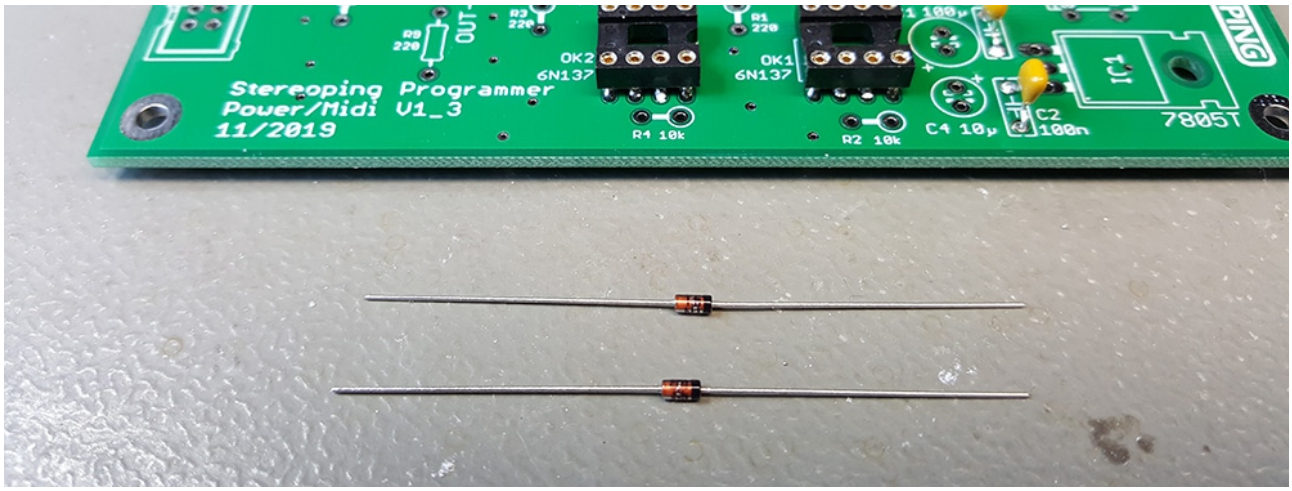
Now place one of the sockets and solder <u>just one leg first</u>. This allows you to check if the socket <u>really</u> lies flat on the PCB from all sides. If it does, solder the rest of the legs.

Small capacitors

We continue with 3 small caps which do not provide polarity, the direction does not matter. The caps labeled '104' ('100 nano farad') are for positions C2 and C3 near the voltage regulator 7805. The third is C11 with 2n2 (Wert '2,2 nF') and the label '222' . The 100n got a spacing of 5 mm, the 2n2 cap got 2,54 mm. All 3 should fit into their positions perfectly, you can not mix them up.



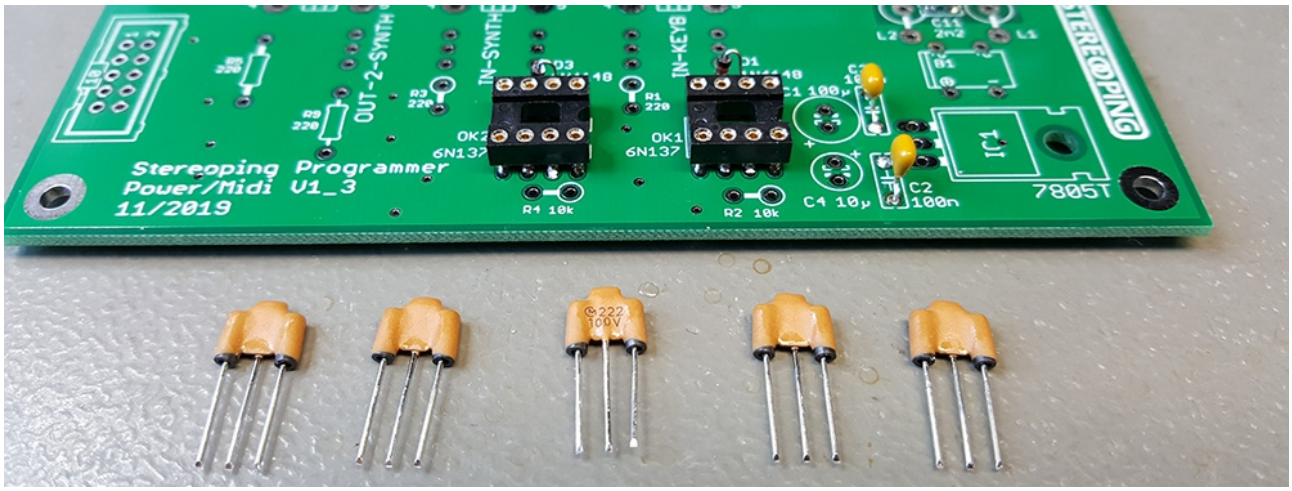
Diodes



Now on to the two diodes. They have a black mark on one side. This mark can also be found on the PCB and should match the mark on the diode. Well, the mark for the red diodes unfortunately can hardly be seen on the silkscreen. Looking onto the PCB with the type 'Stereoping Programmer, Power-Midi V1_3' to the left bottom, the mark of the red diodes must point to the RIGHT – like they are already layed out on the picture above.

LCL-parts

Next we go for the 5 LCL-parts. In short their task is to filter out unwanted high frequency noise and signals catched by long cables – the part improves signal quality to avoid midi data errors. The part has no polarity, place it the way you like.

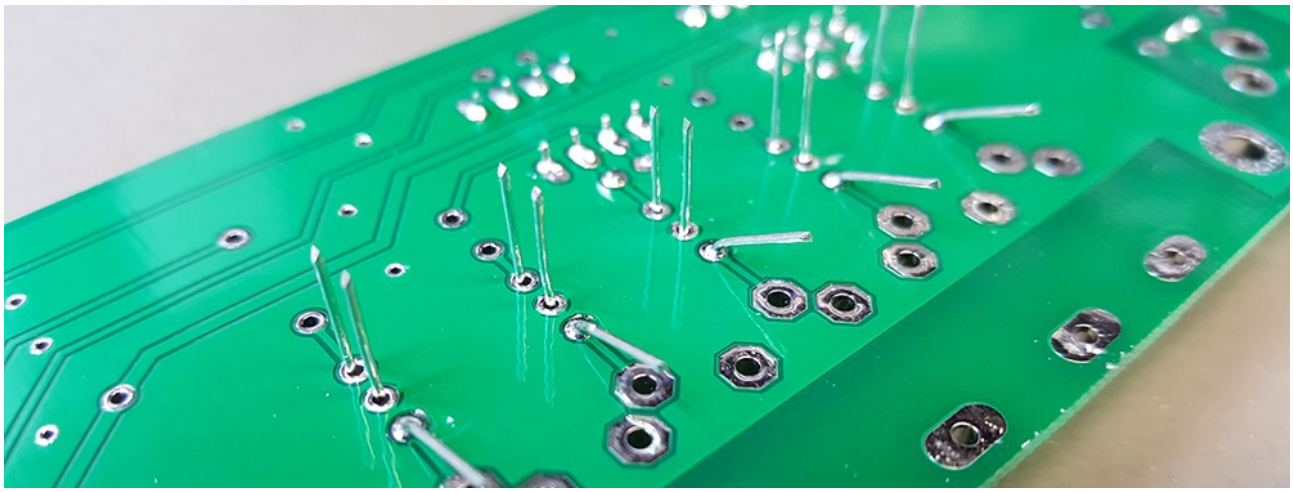


We do it like this:

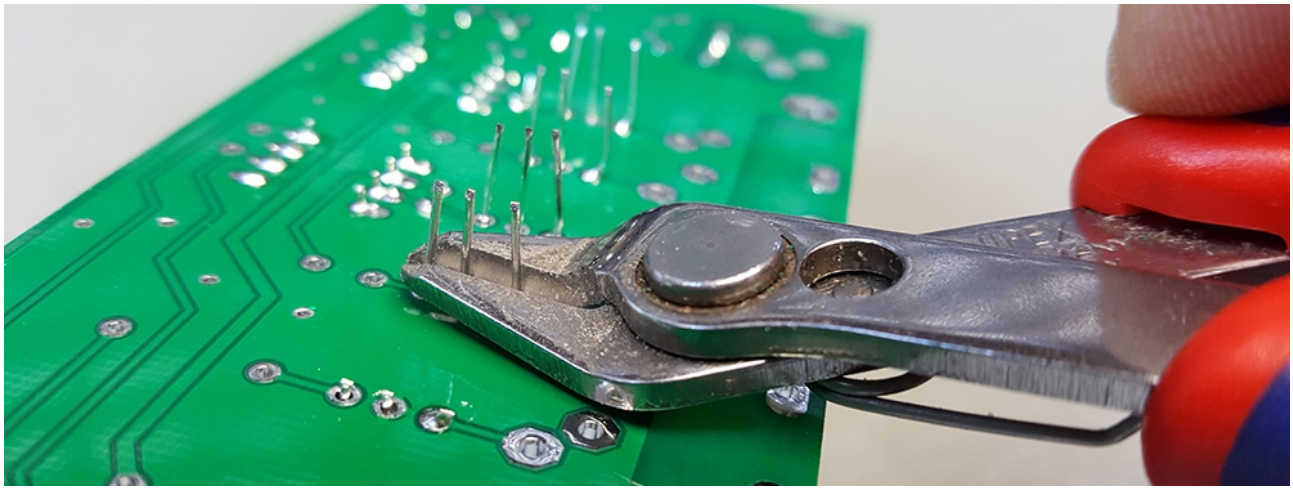
- Put all 5 parts into their position, bend one (or both) outer leg(s) away from the middle leg to keep the part in position. It should be as narrow to the PCB as possible.

- Solder just <u>one</u> leg first. If it happened to slip away from the PCB you can heat the single leg and push it back into the right position. Solder pads NOT connected to the groundplane (without the 4 connection fillets like on the middle pin pads of this part) are easier to be soldered as they do not deviate so much heat to the surrounding groundplane.

This is how it looks from the backside:



Now cut off all 3 legs in one turn, not to short, leave about 1mm.



Then solder the rest of the legs.

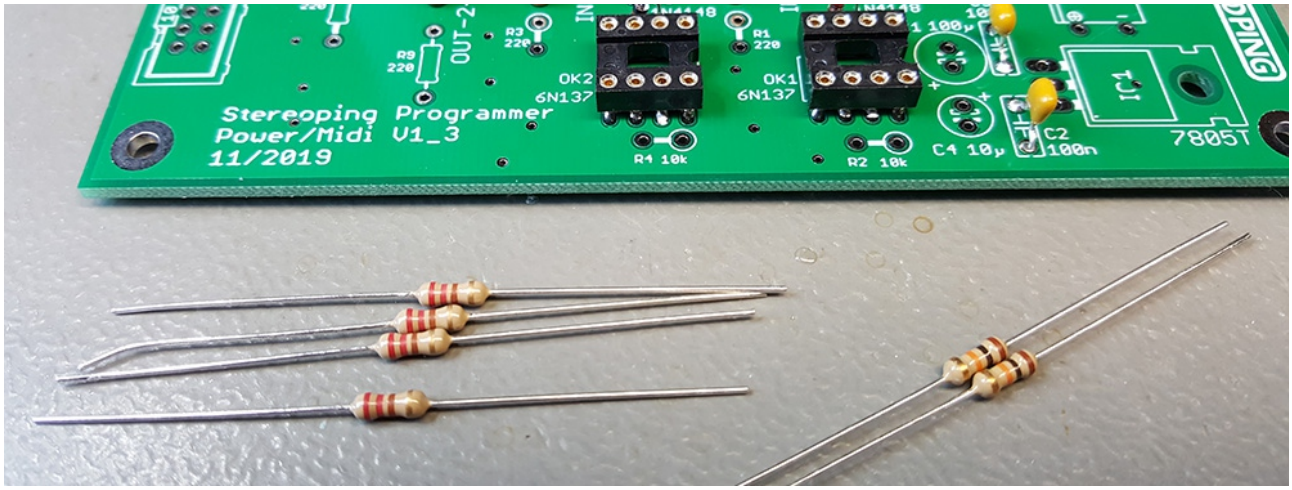
Why cutting off the legs before soldering? It's easier to solder because the shortened wire does not conduct so much heat away from the solder pad into the long metal wire.

Resistors

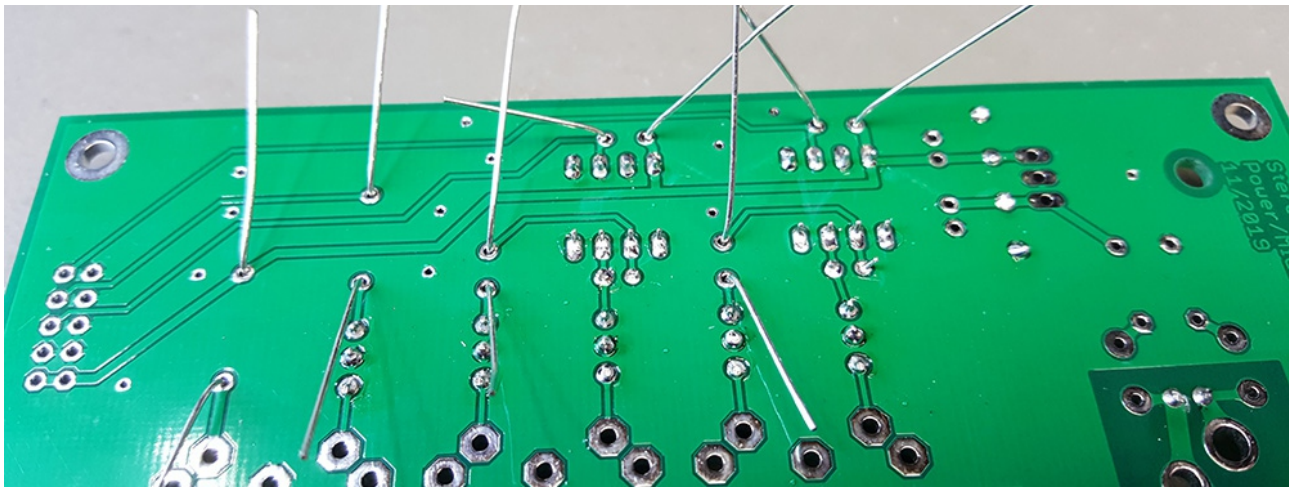
The resistors are marked with an 'R' (e.g. 'R5'), the value (e.g. '220' for 220 Ohm) can be found right besides the name.

There are two values:

- 220 - having the colors Red, Red, Brown, Gold
- 10k – Brown, Black, Orange, Gold



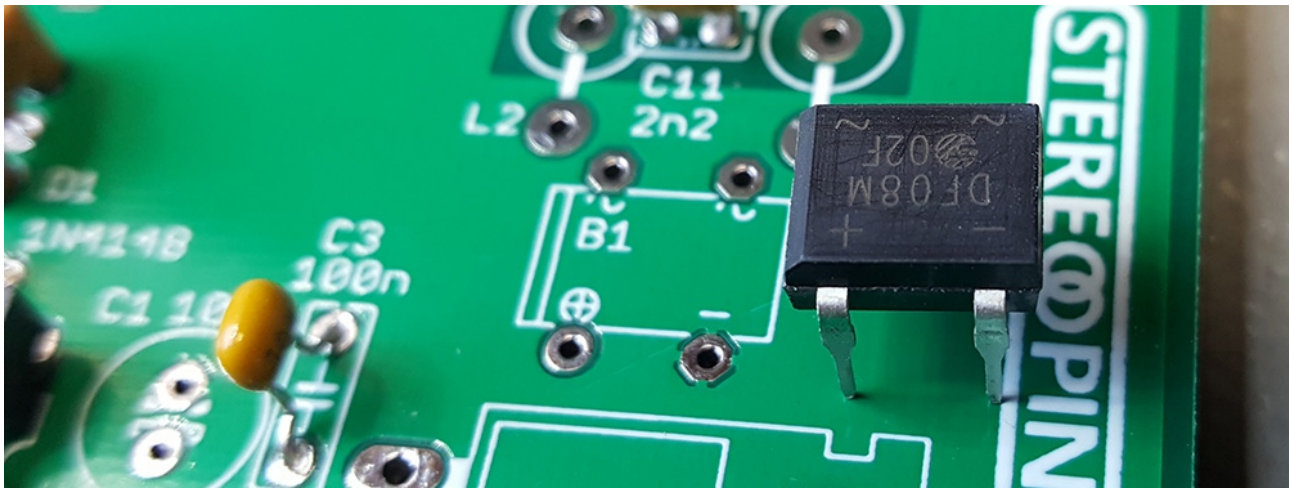
Resistors also do not have any polarity, place them as you like. As all parts, they should be soldered as narrow to the PCB as possible. To fit their solder pins one leg must be bent 180 degrees. On the solderside bend the legs away from each other to prevent them slipping out of their holes when turning the PCBs. After soldering cut off the legs and we're done here.



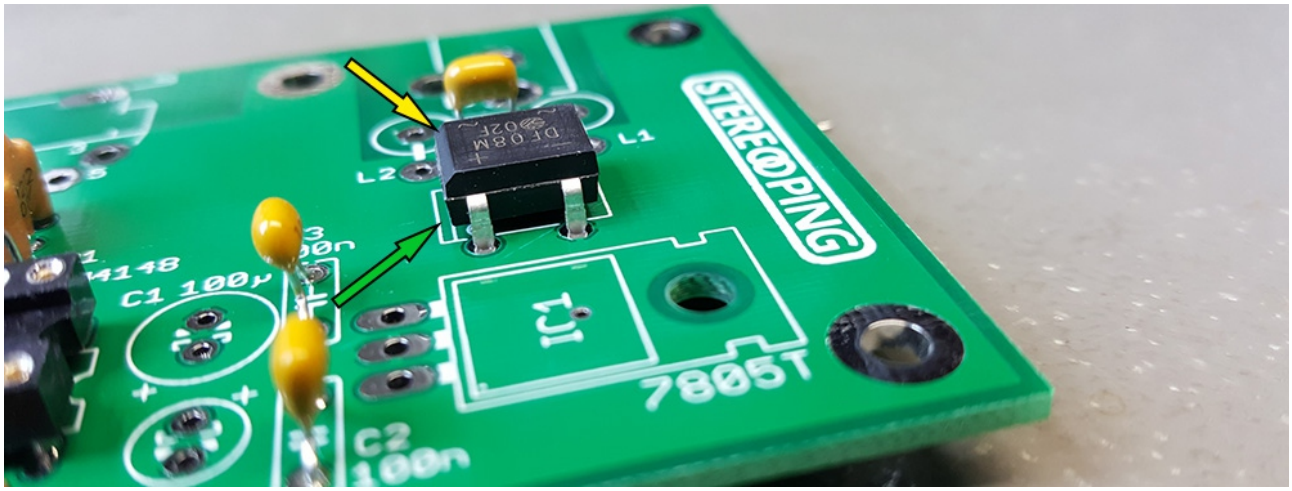
Rectifier

The rectifier does not have a socket provided, it will be soldered directly onto the board. It enables your programmer to be supplied with ANY power supply, no matter of which polarity! Even AC wallwarts will be working perfect. As soon as the wallwart does not provide much more than 12 V.

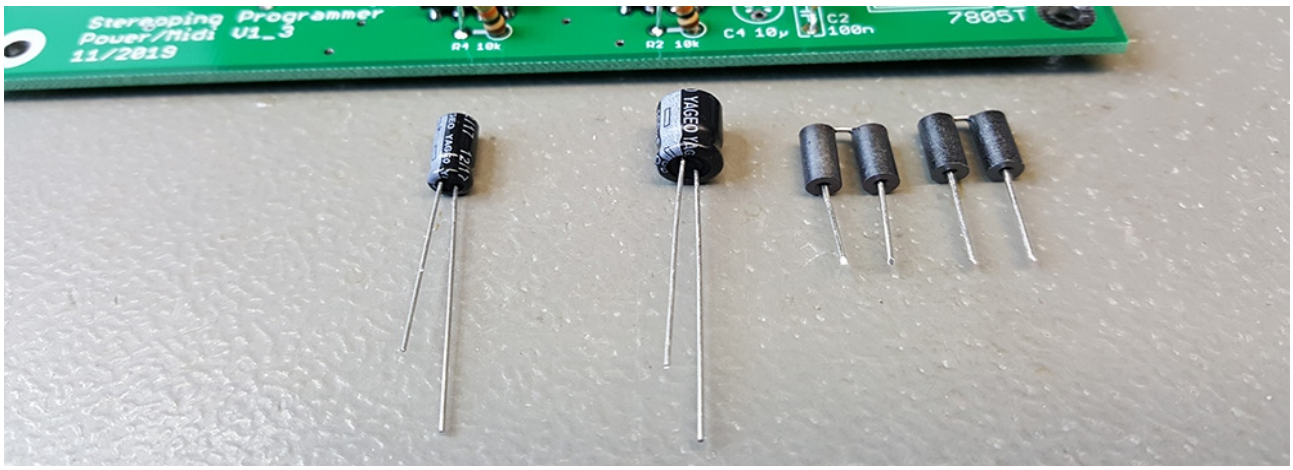
Important! The rectifier MUST be populated in the right direction, which is not that difficult. On the rectifier you can spot 4 icons; plus, minus and two sinewaves. The PCB's footprint shows them as well, just allign the rectifier so they will fit.



There is a second aid: one edge of the rectifier is sloped (yellow arrow), the PCB got as double line on this side (green arrow).



Capacitors with polarity and ferrites

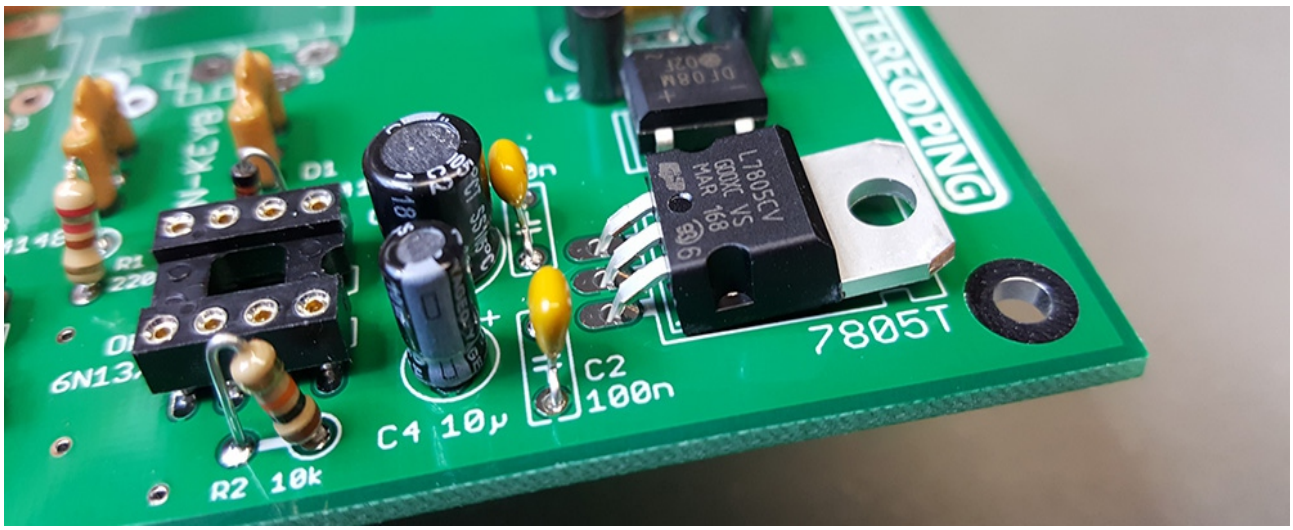


Next are 2 electrolytic caps where it is important to solder them the right way as they got Plus and Minus legs. They got a grey stripe with printed minus signs. The leg on this side is – YEAH! - Minus. The other leg is a bit longer and of course Plus. The PCB's silkscreen shows a little PLUS-sign printed near the leg where the longer caps-wire should come in. The thick 100µ Cap is C1 which got a big, perfectly fitting circle in the silkscreen, the taller Cap with 10µ is for position C4. You can use the same trick we already learnt for the LCL-parts: first solder both wires not connected to the groundplane (without 4 fillets), cut all 4 legs and solder the 2 other wires connected to the groundplane - easier now as more heat goes into the solder pad and not in the long wire.

The ferrite beads will go into positions L1 and L2 near the power supply jack.

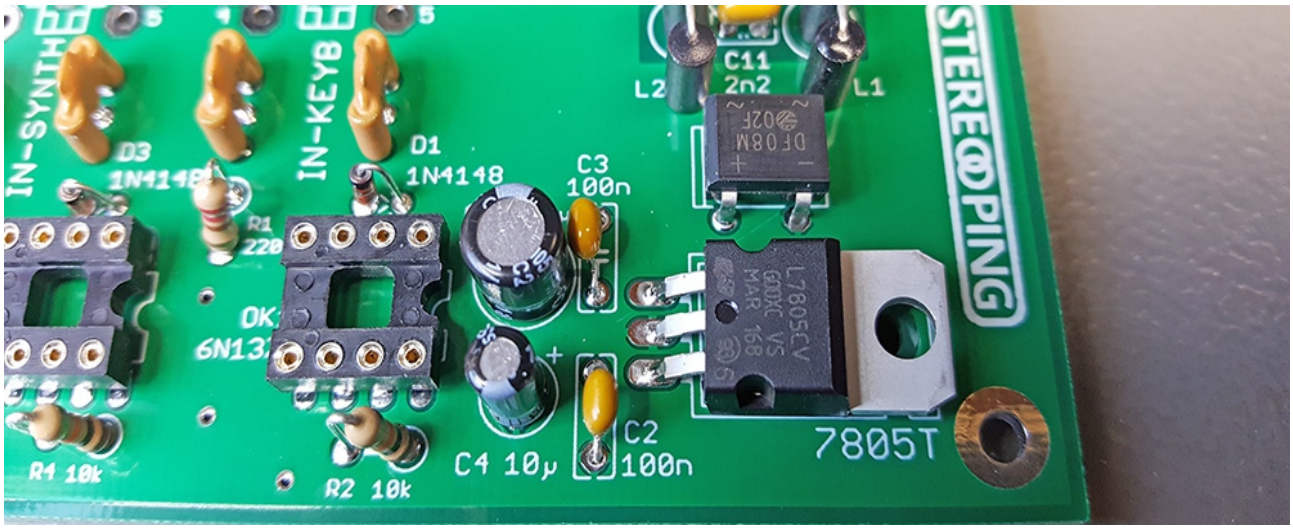
Voltage regulator

Put the 7805 into position with the type facing into the direction of the IC sockets. Push it down onto the PCB and bend it over until it's lying flat onto the PCB.



The regulator pulls the legs a bit out of the PCB automatically, that's ok. Now you can solder it from the backside. Take care to make clean soldering marks. Especially the middle pin is important. The goal is not to use much solder tin, but to make a clean and proper soldermark.

In normal operation it will not even get lukewarm, there is no need to screw it with the PCB or use any heat sinks. It's metal head is internally connected to ground, as well as the screw holes. Means: it is perfectly ok if the screw near the regulator will touch the metal part of the 7805 later, both got the same electrical potential.

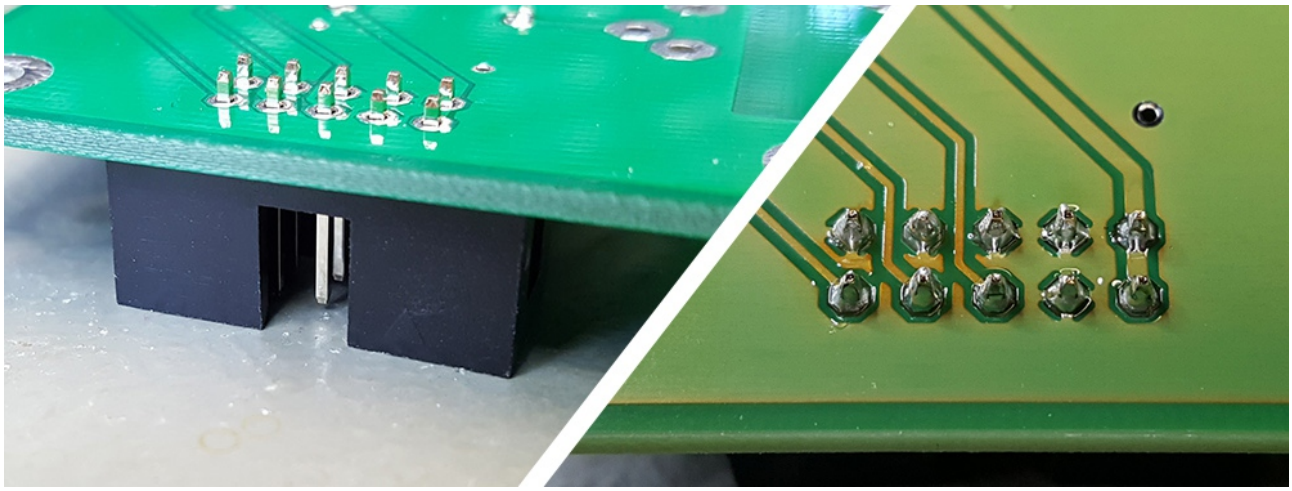


Connector for flat cable

Note: it might be the case, this connector is already soldered onto the PCB to avoid problems.

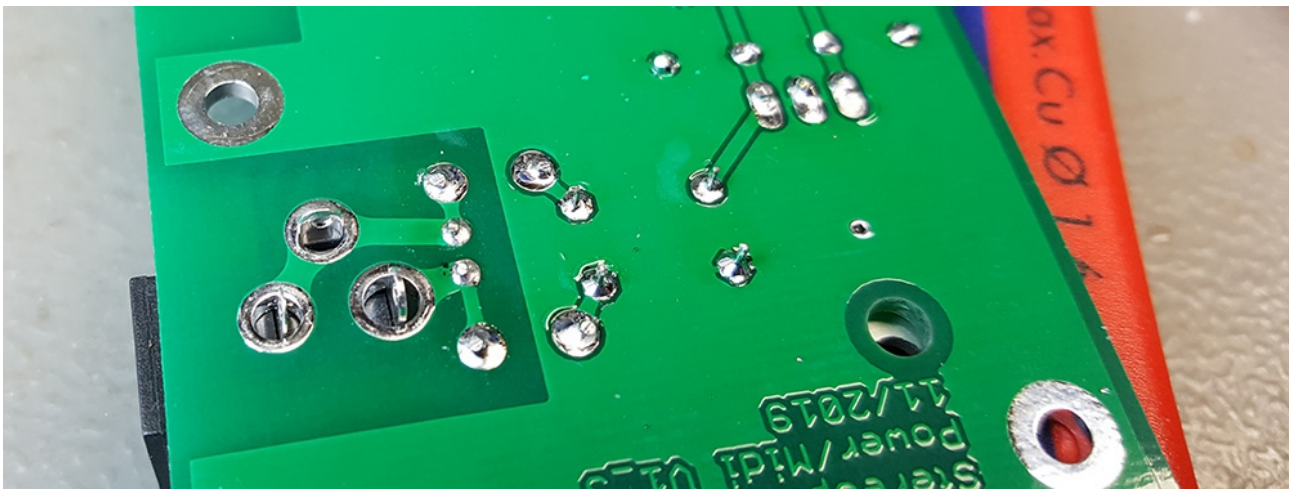
Ok, take a deep breath, clean your solder tip and we can dare to continue with the 10 pin connector. The pins are quite near together. This is the cable where Midi data gets in and out as well as the power supply – this is the aorta of your programmer and should be soldered thoroughy.

Place the connector from above – **take care for the right direction of the cavity!** It should be on the same side as the silkscreen shows. We recommend soldering one leg first and check again for the connector to sit flat on the PCB. Then the other legs can be soldered. Take your time (4-5 seconds per leg) to allow the tin to flow through the via onto the other side of the pad.

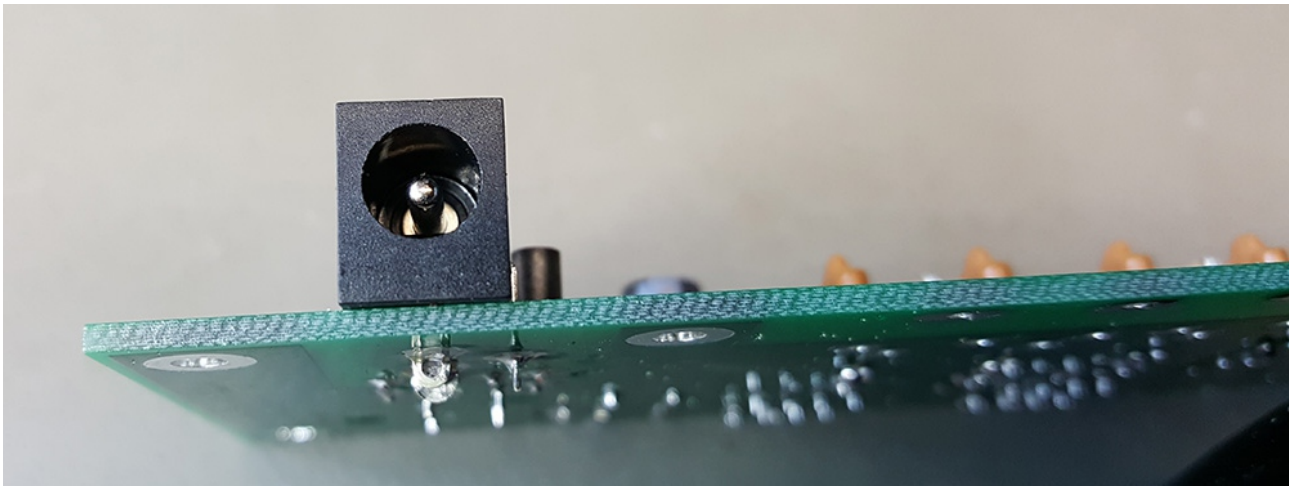


Power supply jack

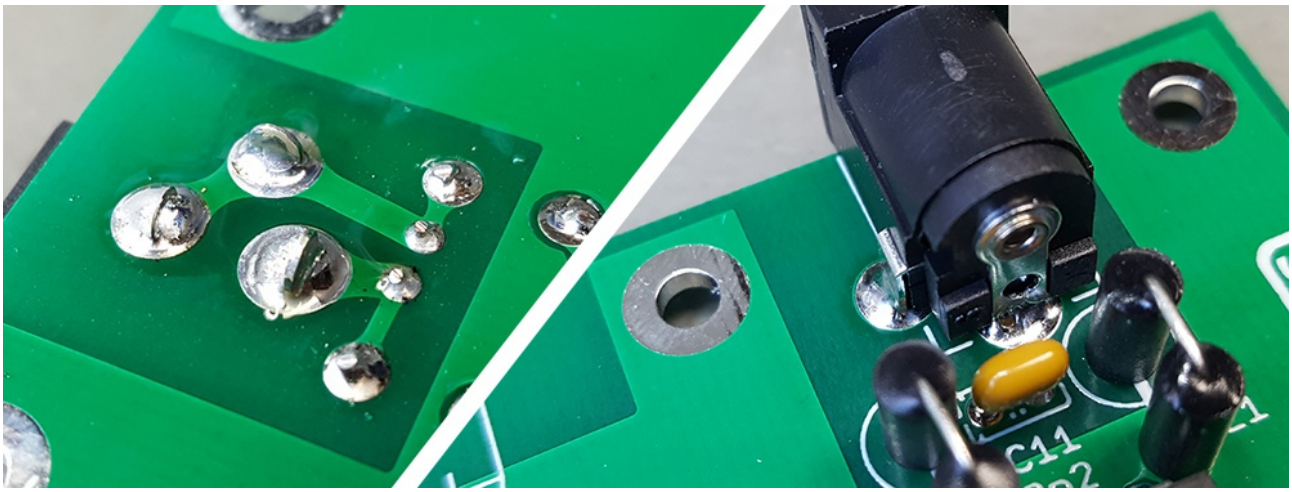
To get the power supply jack into the right position it's easiest to lay the PCB with the jack directly onto the table. You might want to put something beneath the opposite side for the PCB to be parallel to the table.



As always: we would solder one leg and take a look if the jack is sitting perfectly flat:



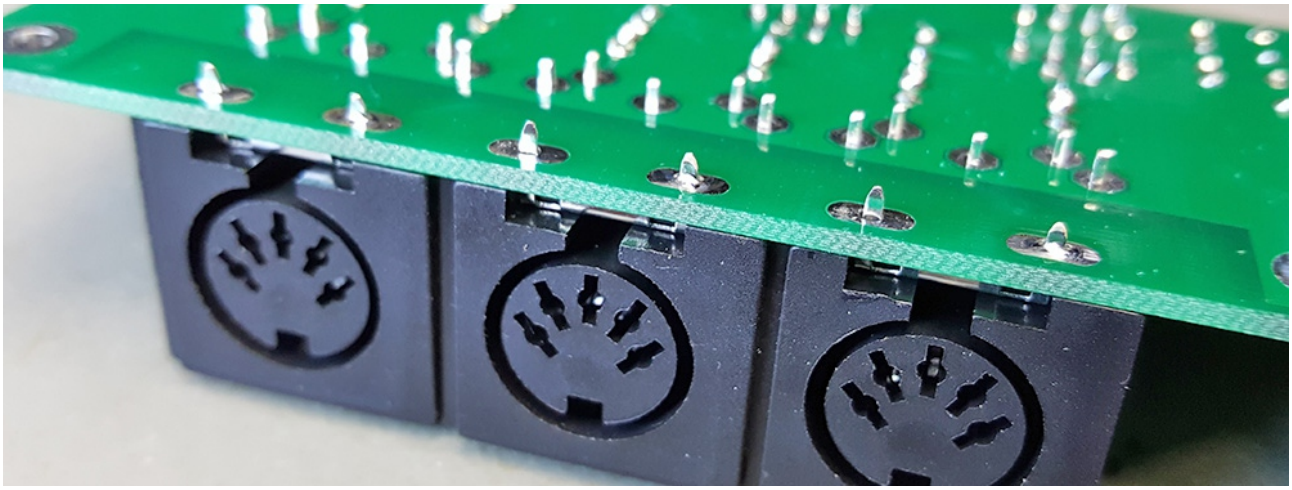
Then got for the other two contacts. In the end it should look like this:



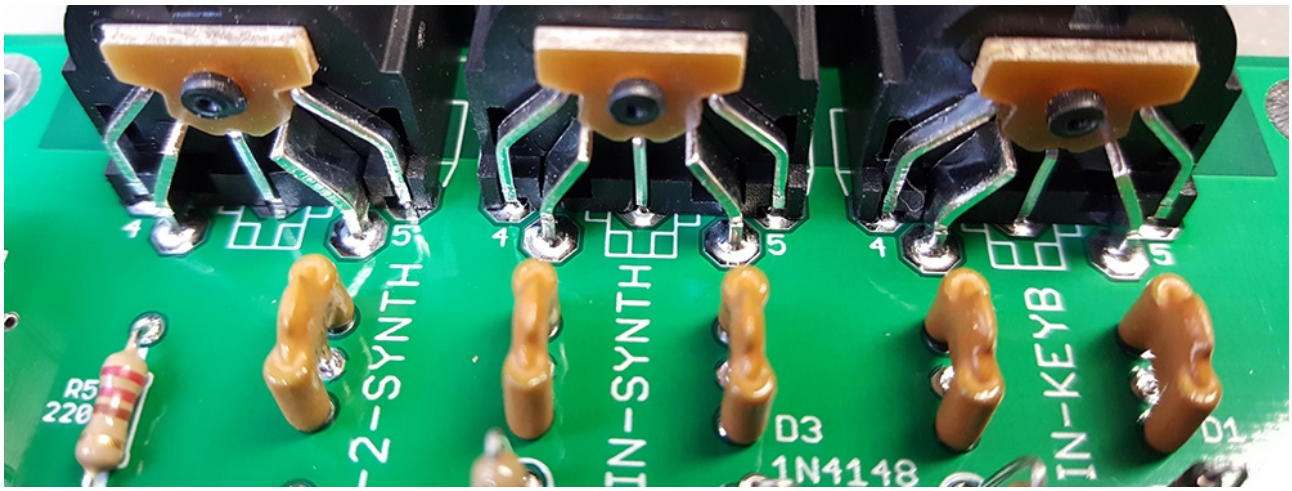
Midi jacks

For the power supply and Midi jacks it is important they are perfectly flat on the PCB. Otherwize your powercord or midi cables will stand crooked – that's not what you want.

Put the PCB with midi jacks upside down and relaxed onto the table. Solder one of the front pins of each jack. Now look from the other side for all the jack's edges to sit perfectly flat on the PCB.

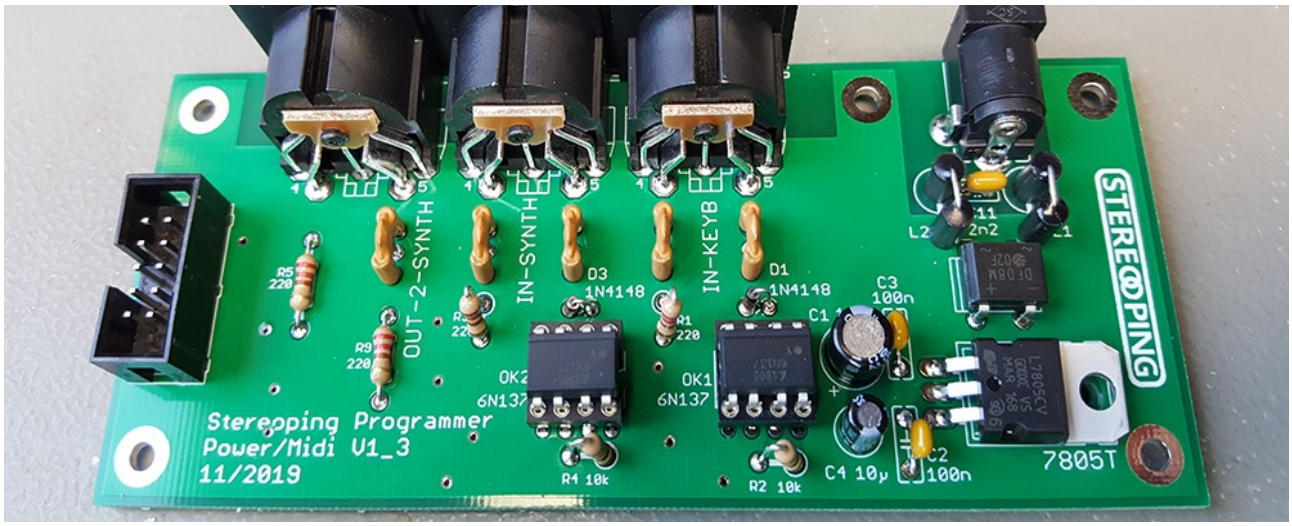


Preferably there should not be any space between jack and PCB. Otherwize take PCB and jack between thumb and trigger finger and heat up the single solder mark again. The jack will snap into optimal position.

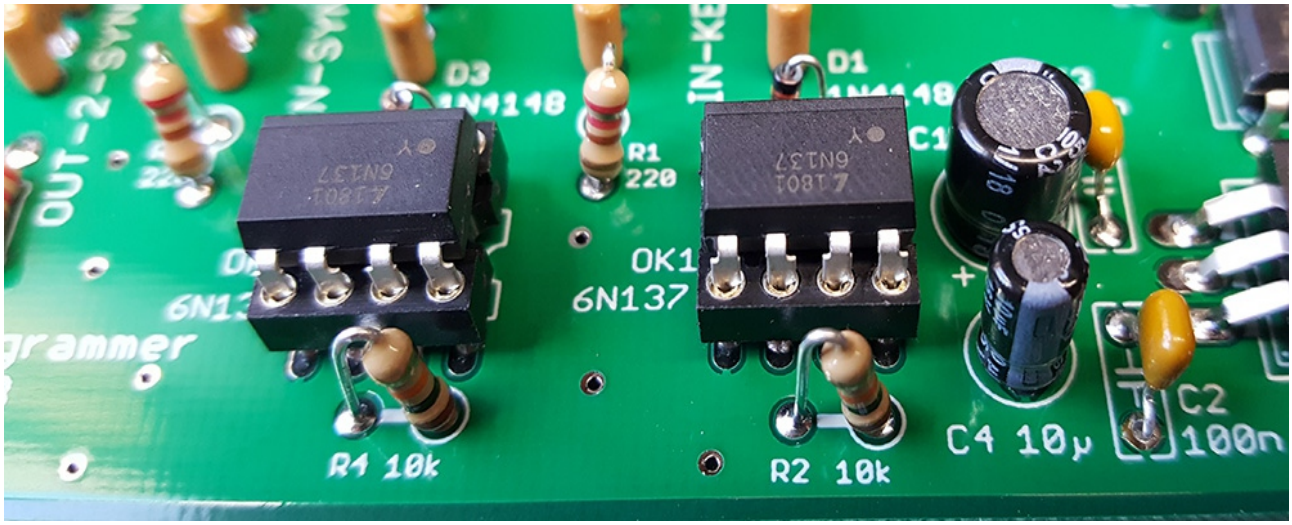


Solder all remaining pins and do not be stingy with tin and time. If the pin and the leg are heated up properly and you give the tin a bit time to flow, it will crawl through the via onto the other side of the PCB as the picture above shows quite well. That's what we want - the jack will be stressed mechanically later by midi cabled being pushed in and ripped out.

That's it, you got it! The rest of bag A's content is for case assembly which we will need later.
The Power-Midi board is finished and hopefully it looks even better than the one on the picture:



One little word: if you got the PCB laying in front of you like on the picture above, the optocouplers will be plugged into the sockets upside down. Therefore another detail picture:



4. Main PCB – Bag B

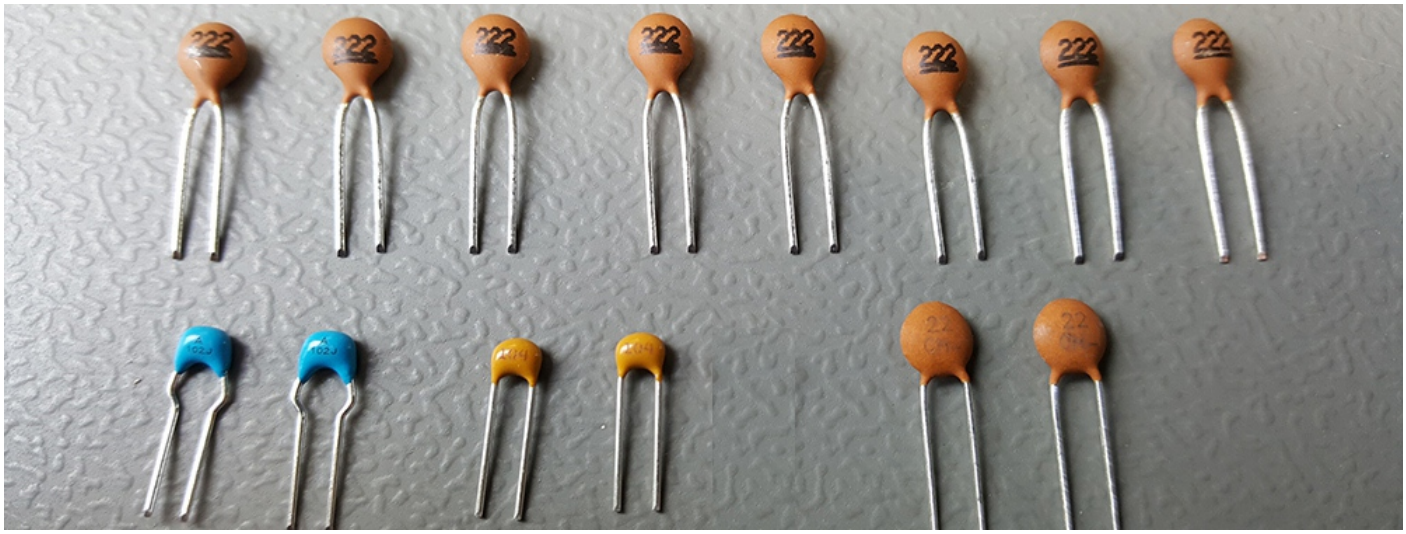
Note: some pictures show an outdated PCB of the mainboard.

IC-sockets

Note: if you are a skilled solder-pro you can leave out the 7 sockets with 16 pins and solder the ICs directly. But it's quite advisable to solder the 40 pin for the main processor in any way.

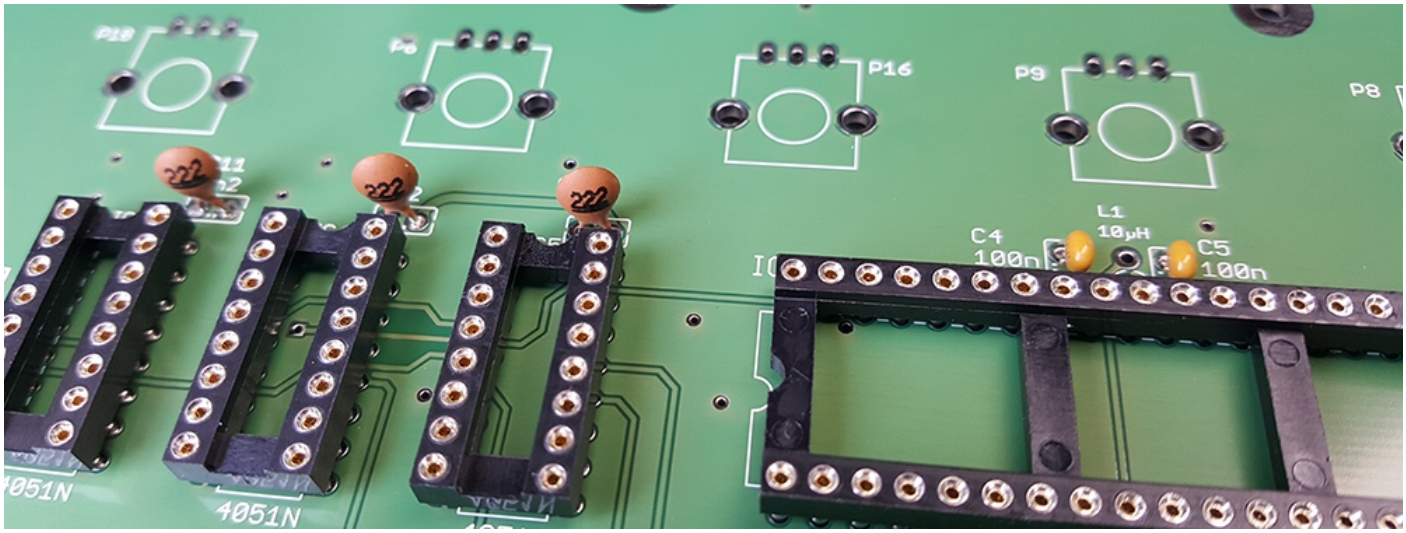
The same here as with the Power Midi PCB: place the sockets with their mark fitting the silkscreen. Solder one leg first to go sure it's flat on the PCB.

Capacitors



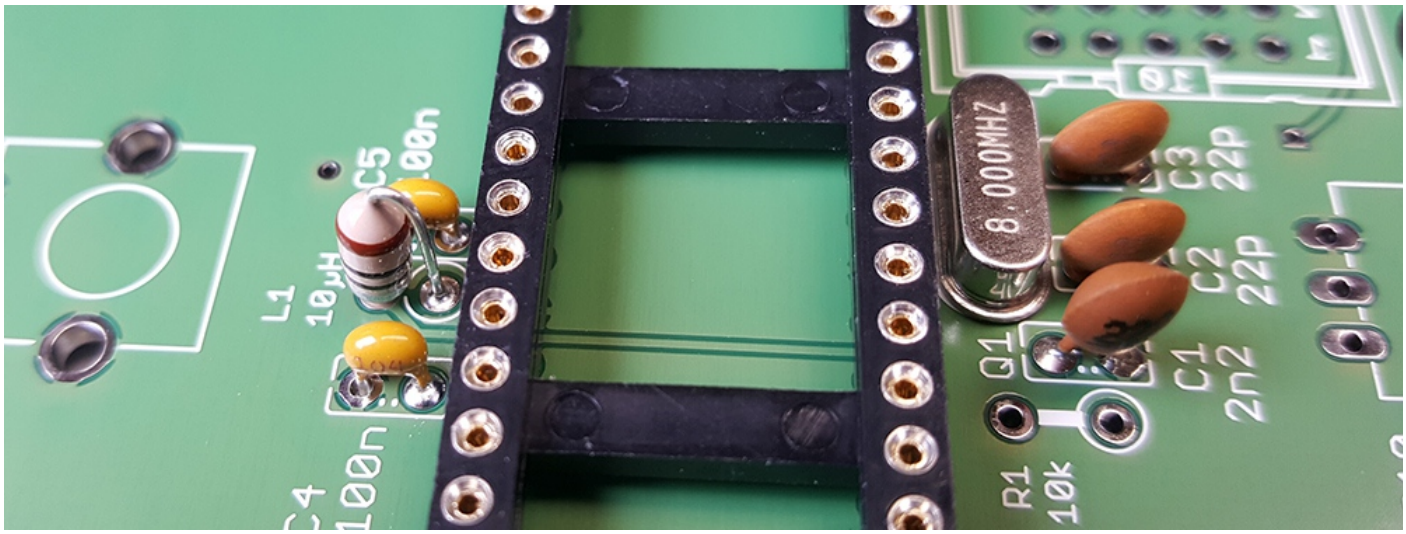
Ceramic capacitors do not have polarity. Place them as you like, bend the legs a bit, solder them and cut off the wires. Their real color and look might be different than shown on the picture above! There are 4 different values:

- 2 x 22p, bright brown with label '22' for positions C2 and C3
- 2 x 1n with label '102' for positions C10 and C14
- 8 x 2,2n with label '222' for positions C11, C12, C13, C1, C9, C15, C16 and C17
- 2 x 100n with label '104' for positions C4 and C5



Crystal and inductor L1

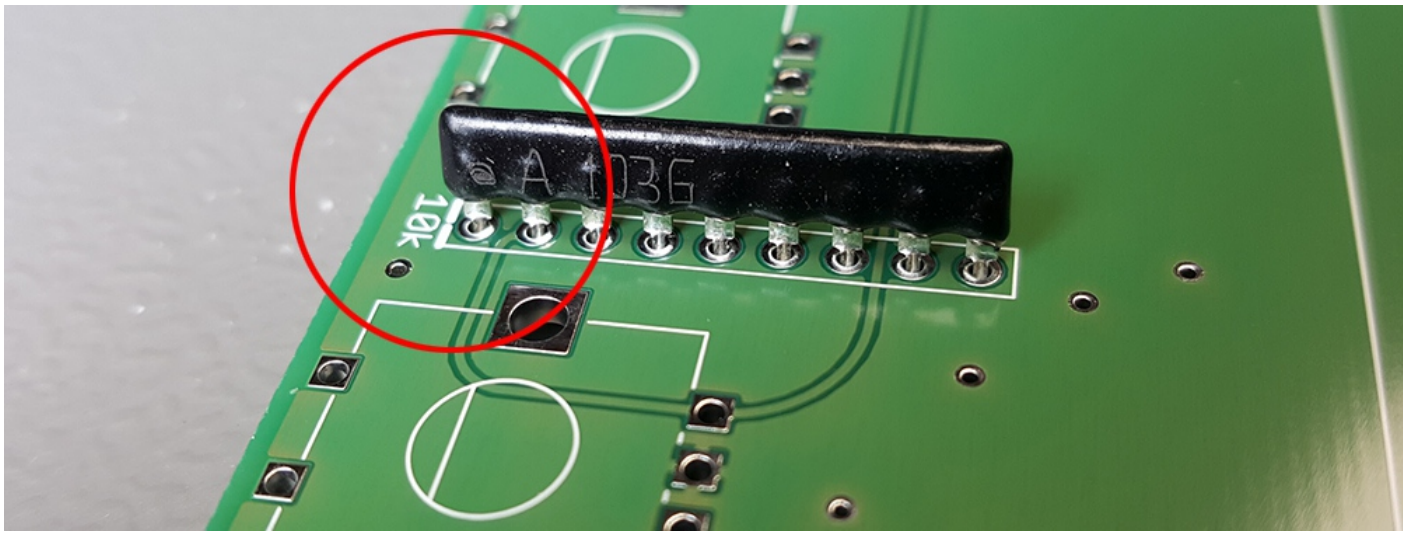
The crystal is below the 40 pin socket, L1 is above (in the picture to the right and left of the socket). Both can be soldered either way, no polarity. For the crystal it is especially important to sit flat on the PCB.



Resistor array

Pitfall! Please pay extra attention here to place the resistor array the right way! There is a printed point on the part. This is the side which must fit the fat line of the silkscreen's rectangle where '10k' is printed.

A picture hopefully clears this up:



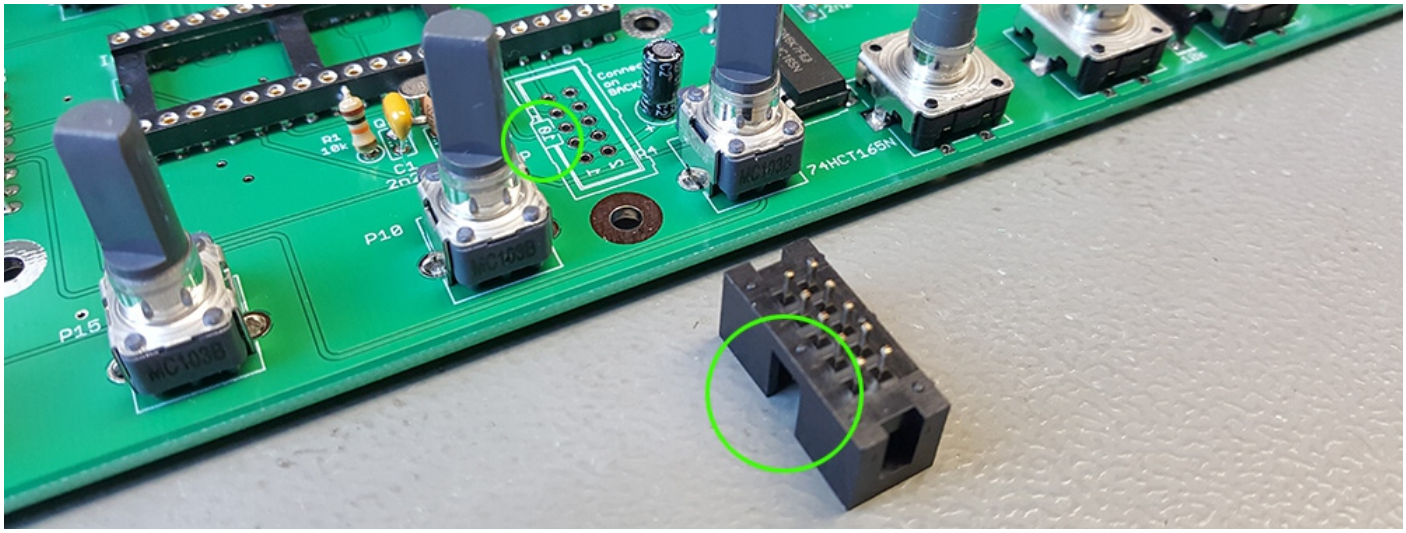
Connectors for flat cable

Please dont get confused if you see the pots already populated in the (outdated) pictures. It is easier to solder the connectors before you start with the pots. Just ignore the pots in the pictures for now.

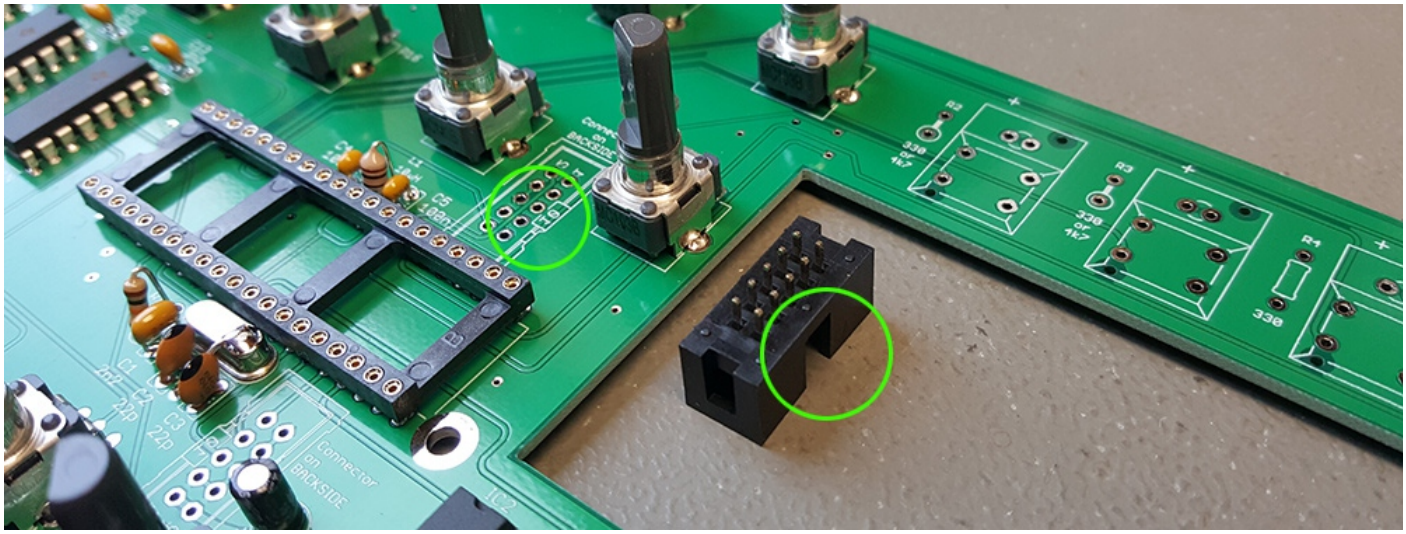
This step should be made extra thoroughly, we need to place two connectors on the backside of the PCB and in the right direction.

Attention, pitfall! Both connectors go to the <u>BACKSIDE</u> of the PCB, indicated by the mirrored print on the silkscreen.

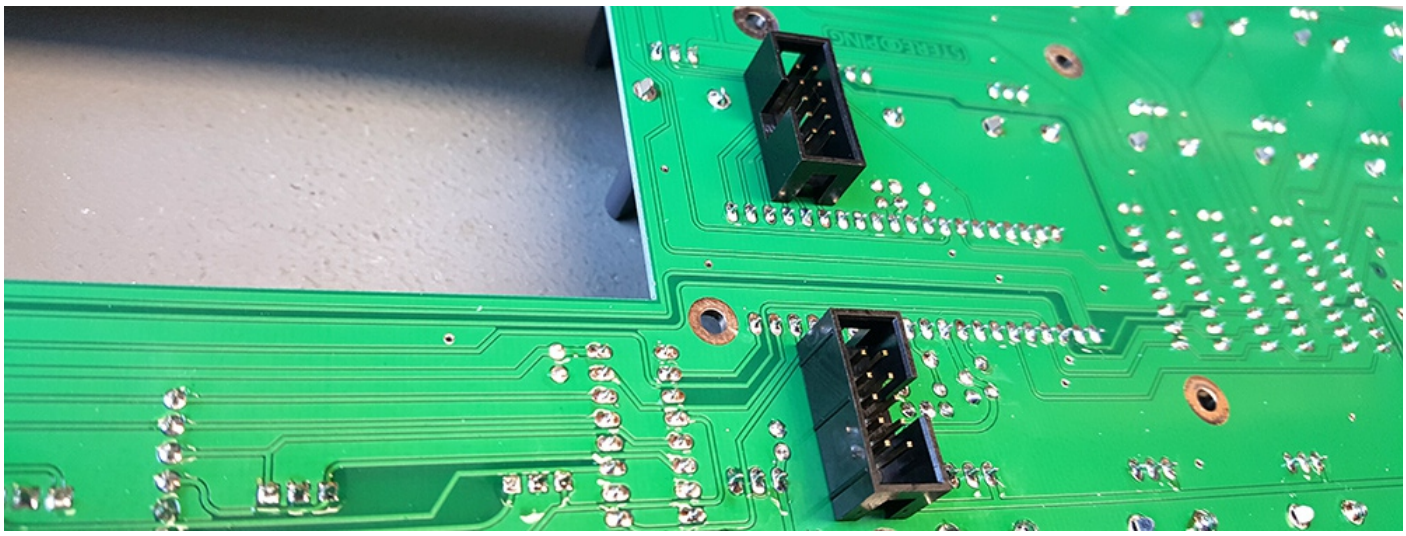
Seen from above and front : the cavity for the connector to the power/midi board is pointing to the left side.



The connector for the display is pointing to the right side:



After you put them into their places it should look like on the following picture. If it does, solder the pins.



Potentiometers

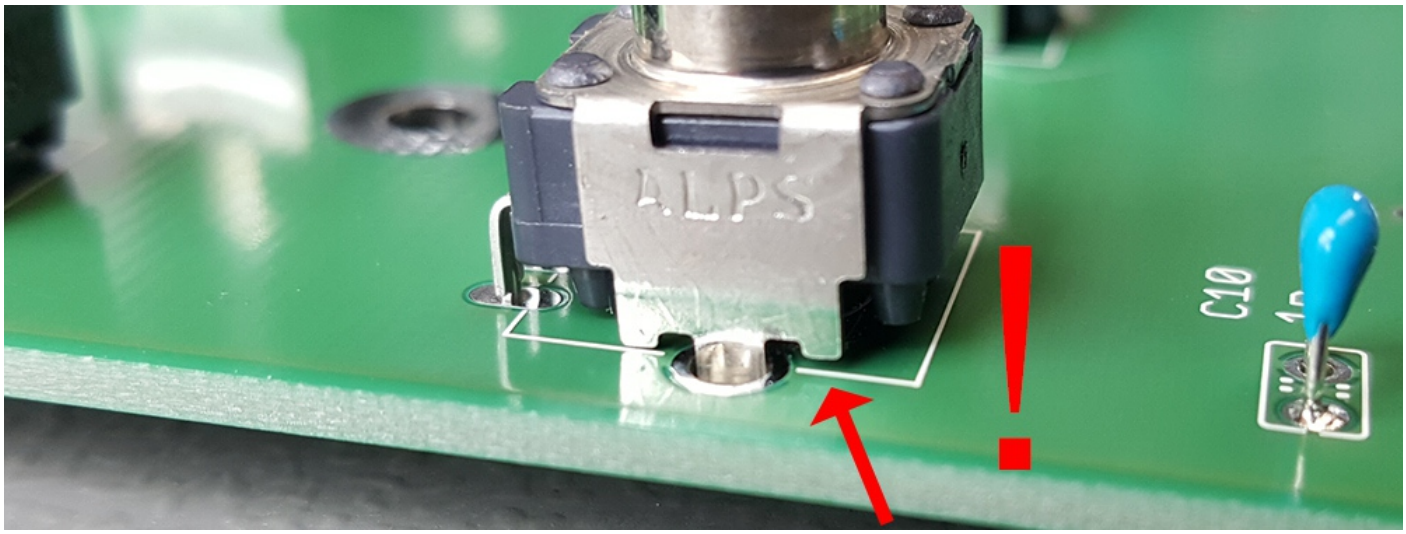
Now is the time for the 45 potentiometers. It's easier to put in all 45 pots alltogether as the PCB sits much better on the table then.

Attention: there are some pitfalls here. To get a crooked pot out again is not much fun. Please do not solder them before having read the following!

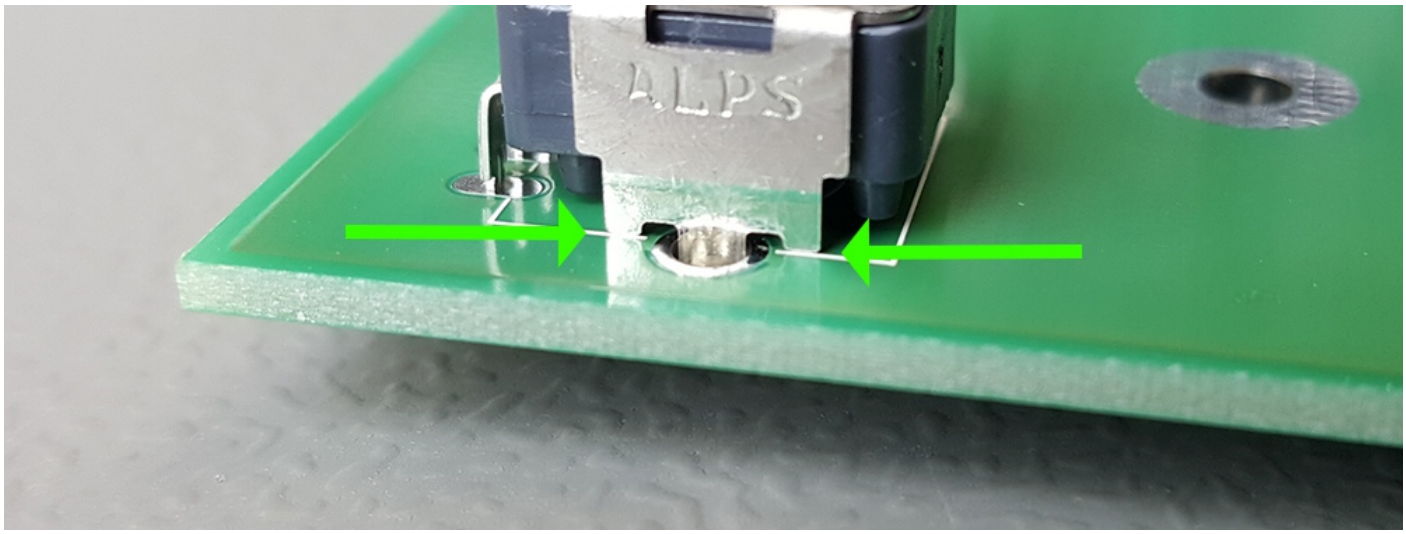
Pitfall 1: when putting them into position go sure the 3 contact pins find their holes. The legs can be slightly bent from transport. It sometimes happens when pushing a pot with a slightly bent leg into position, the leg gets bent even more when missing the drilling hole.

Pitfall 2: Even if you got the impression all pots are snapped in with their metal brackets, they still can tilt a bit. It's not unlikely the one or other pot is not perfectly aligned. You should check every single one of the 45 pots from both sides. We found it easiest to examine the end of the metal pins to touch the PCB (see photos). The pin and it's mirror on the PCB should be in touch.

NOT this way please:

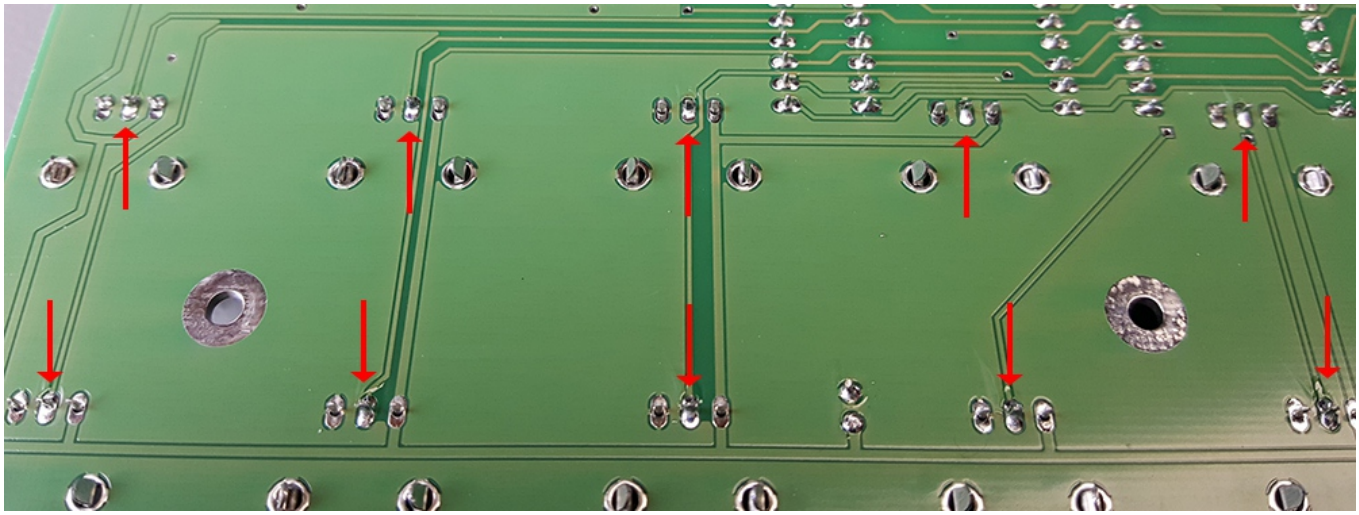


Yeah, looks much better:



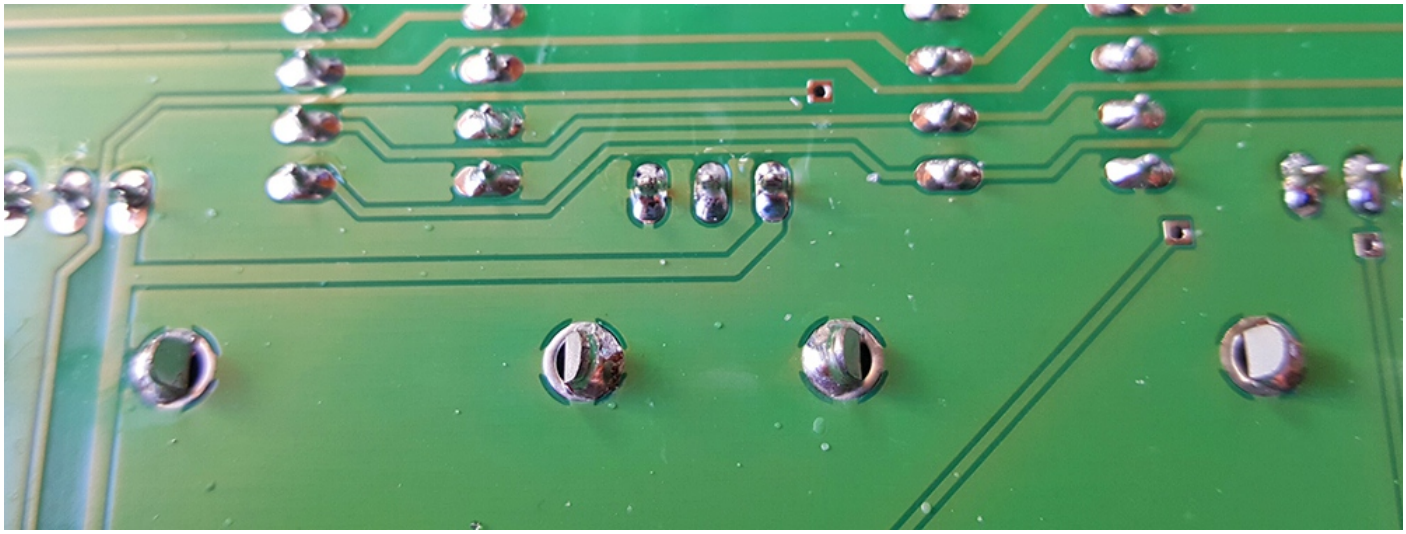
Best practice:

- align all pots as described above
- turn over the PCB and carefully put it on the table
- solder the middle pin for all pots:



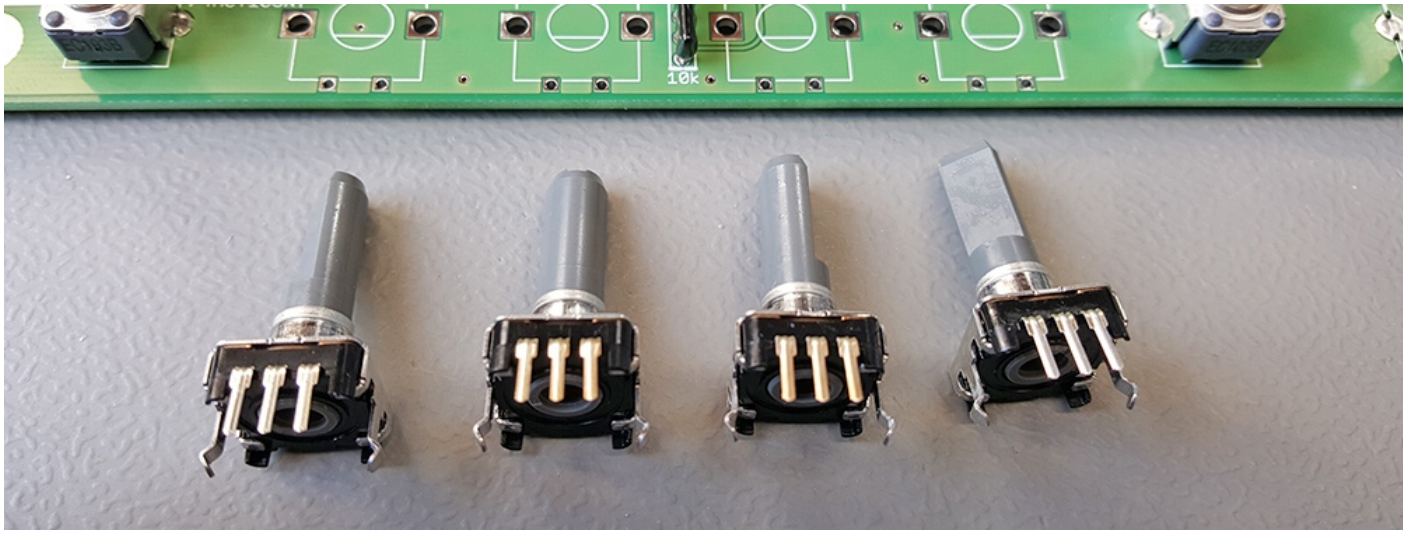
  Now they are kind of fixed althoug they still can tilt a bit. Now you can examine all pots once again, maybe one got tilted by putting the PCB on the table. If one must be corrected you simply can do that by just heating up this middle pin.
- We then solder the metal bracket
- At last both remaining pins

It's not necessary to flood the whole opening on the metal brackets with lots of tin. If the inside gets flooded like shown in the picture that is perfectly enough:

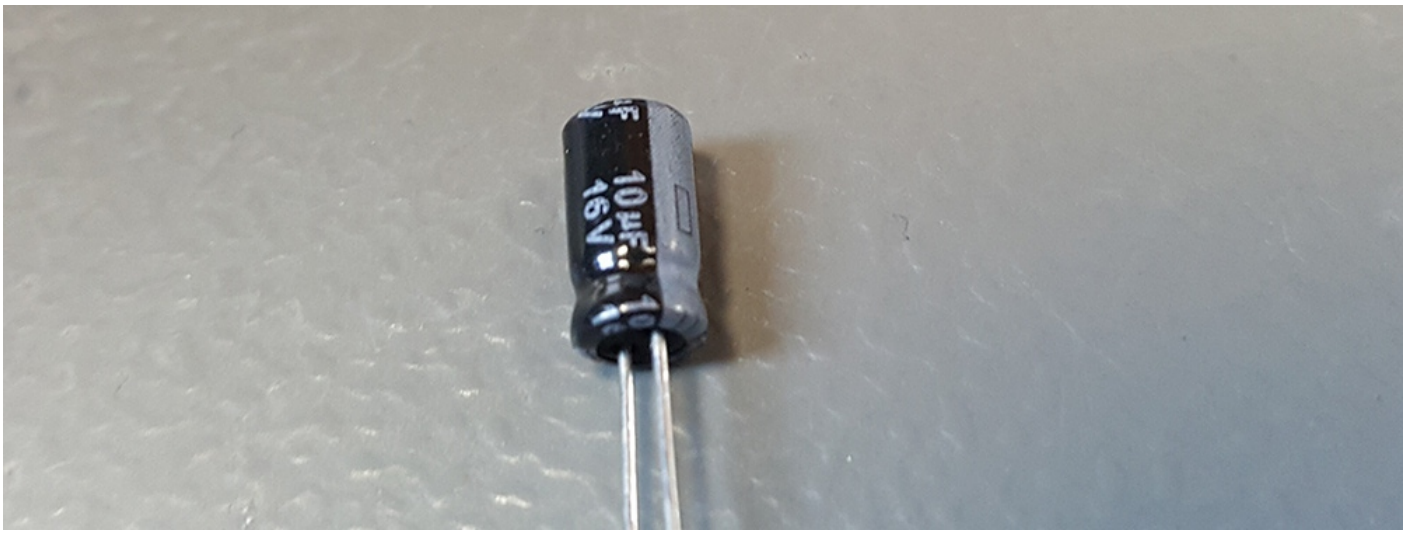


Encoder

Not much to say. Snap them in, check their position to be straight and solder all legs. The solder pins are quite small here.



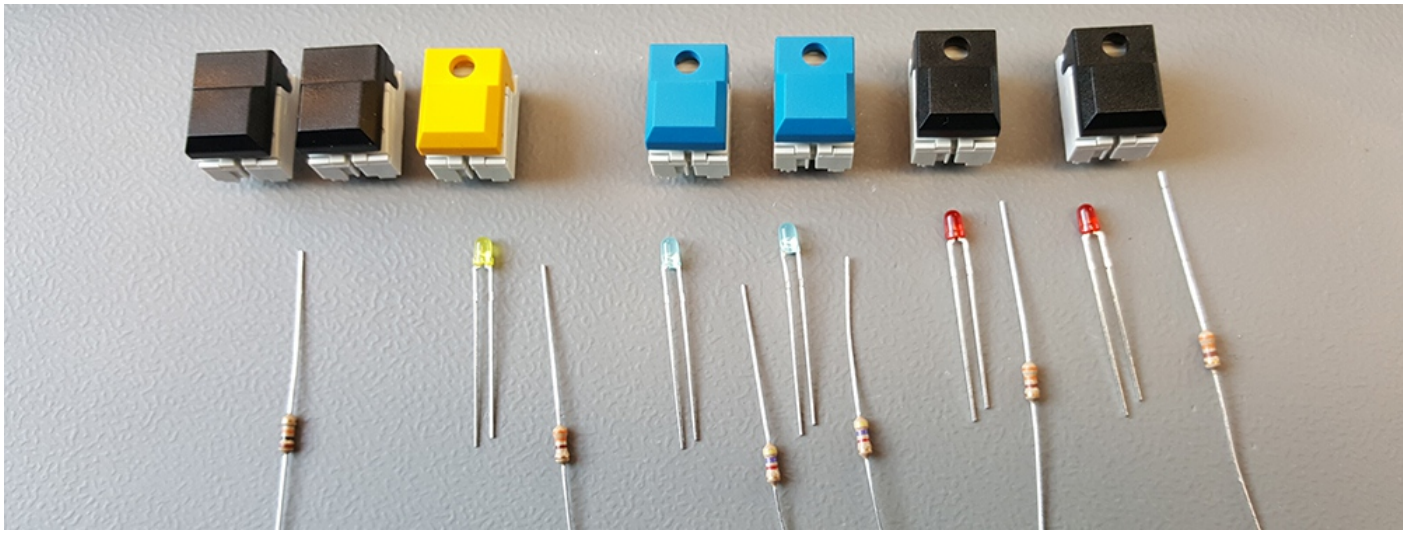
Electrolytic capacitor 10µ



You already know the smaller 10µF cap from the power midi board. This is for position C6, mind the polarity! Long leg into the pad where the small + is in the silkscreen.

Buttons, LEDs, resistors

Each edition got different button- and LED colors, as a consequence the resistors also are sometimes different.



10k resistor

You know this part already, ring colors Brown, Black, Orange, Gold. It comes into position R1 near the 40 pin socket.

LED colors and their resistor values

The Synth Programmer's Editions all in all have 3 LED colors: red, blue and yellow. Red and yellow LEDs always get a 330 Ohm resistor – colorcode Orange, Orange, Brown, Gold. The blue LEDs have a 4,7k resistor – Yellow, magenta, red, gold. In the following we provide button and LED colors for all editions as well as the resistors:

Matrix

- Buttons 1&2 left side – Button black without LED
- Button 3 left side – Button red, LED red, R7=330 Ohm
- Button 1 upper row – Button blue, LED blue, R2=4,7 kOhm
- Buttons 2-4 upper row – Button black, LED red, R3/R4/R5=330 Ohm

Microwave

- Buttons 1&2 left side – Button red without LED
- Button 3 left side – Button red, LED red, R7=330 Ohm
- Button 1 upper row – Button blue, LED blue, R2=4,7 kOhm
- Buttons 2-4 upper row – Button black, LED red, R3/R4/R5=330 Ohm

80

- Buttons 1&2 left side – Button black without LED
- Button 3 left side – Button yellow, LED yellow, R7=330 Ohm
- Button 1 upper row – Button blue, LED blue, R2=4,7 kOhm
- Button 2 upper row – Button blue, LED blue, **R3=4,7 kOhm**
- Buttons 3&4 upper row – Button black, LED red, 330 Ohm

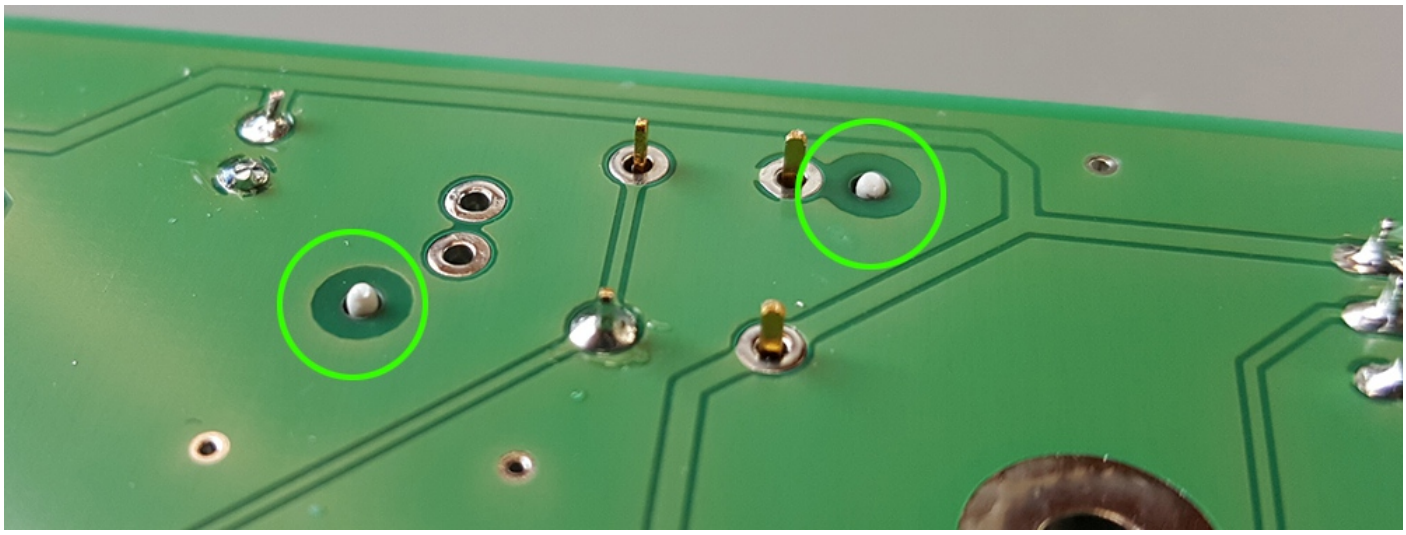
Pulse
- Buttons 1&2 left side – Button black without LED
- Button 3 left side – Button black, LED red, R7=330 Ohm
- Button 1 upper row – Button blue, LED blue, R2=4,7 kOhm
- Button 2 upper row – Button black, LED red, 330 Ohm
- Buttons 3&4 upper row – Button red, LED red, 330 Ohm

Chroma
- Buttons 1&2 left side – Button black without LED
- All other buttons black, LED red, resistors all 330 Ohm

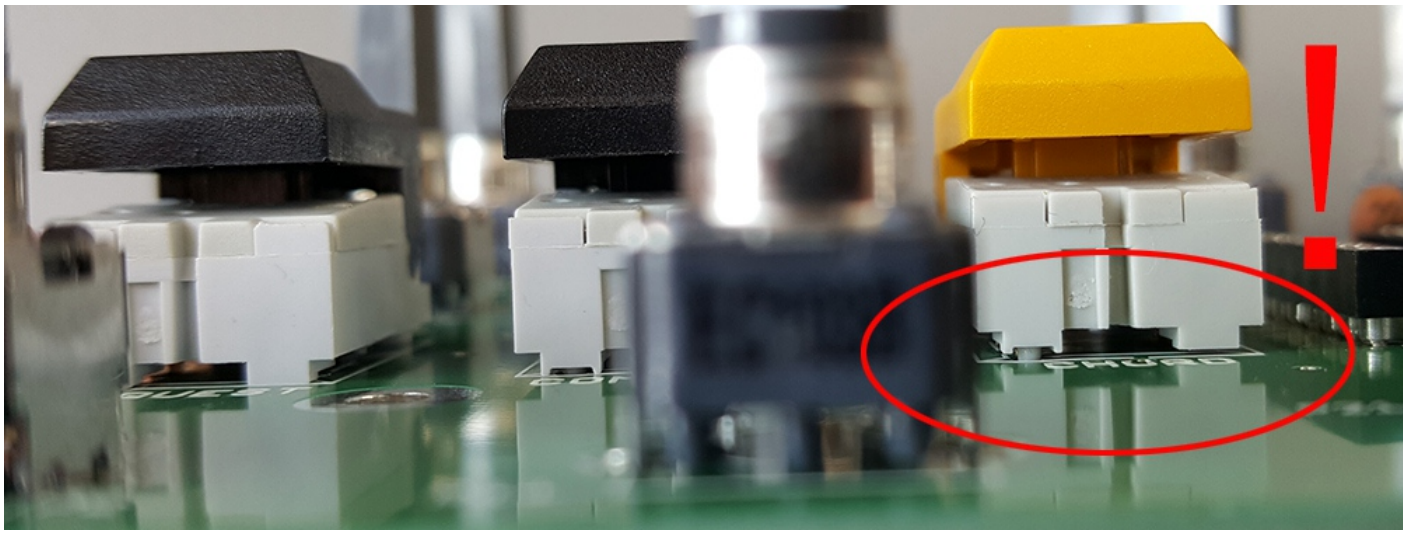
Prophet VS
- Buttons 1&2 left side – Button black without LED
- Button 3 left side – Button blue, LED blue, **R7=4,7 kOhm**
- Buttons 1-4 upper row – Button black, LED red, R2/R3/R4/R5=330 Ohm

The resistors should not make any problems. Place part, bend legs, solder wires, cut wires.

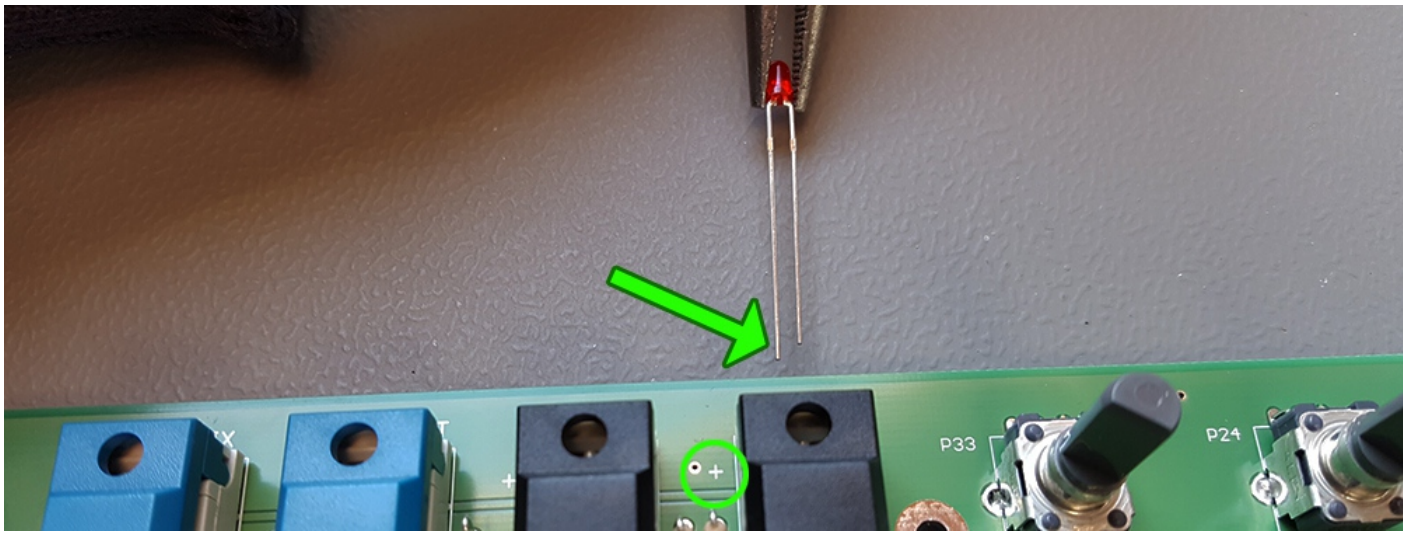
Now on to the buttons. We solder the buttons without LEDs, the LEDs can be pushed into the buttons easily after they are soldered on the PCB.
It's the same as with the pots. The buttons tend to sit in their positions not perfectly flat. The old trick also works here: solder just one leg, turn PCB and examine all edges. Correct position if necessary by heating up the solder mark. Then solder the rest of the legs. There is another help: the buttons got 2 plastik pins going through PCB drills. They help to evaluate from the solder side if the button sit right or not.



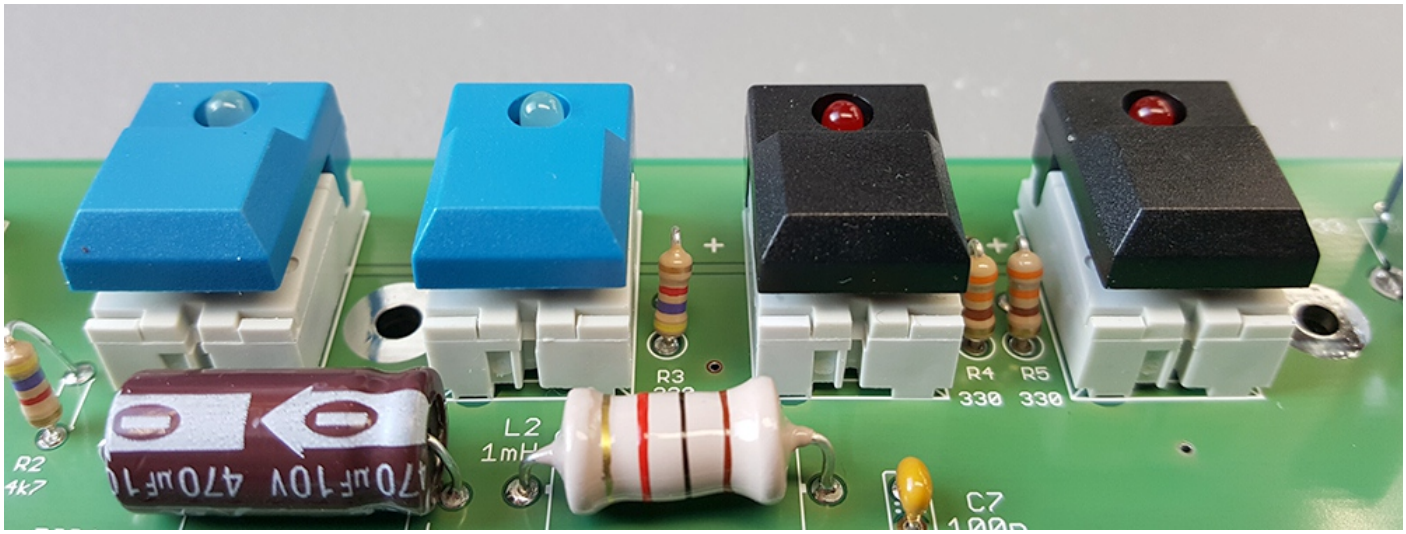
It's never wrong to double check. An example of a button sitting bad is shown in the photo:



After the buttons you can push the LEDs into position. **Attention – mind the polarity!**
To the left of the button near the LED you find a little '+', this is the side for the longer LED leg to go.

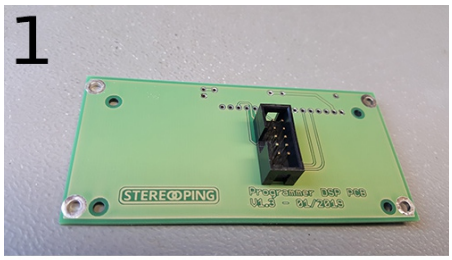


Note: the blue LEDs are a bit longer than the red ones, it's perfectly normal they stand out ½ mm more.(Picture shows outdated PCB version)

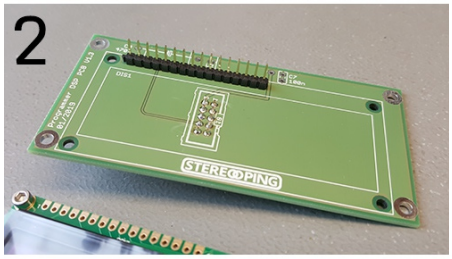


5. Display PCB

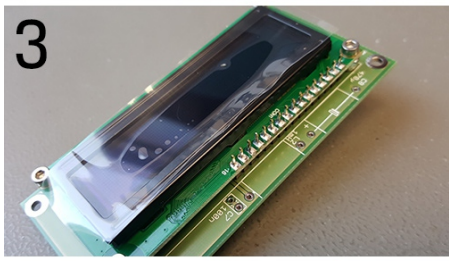
The last step. The display comes last as it is the most sensitive part. It would be a pity if by accident you would scratch the screen with the soldering iron or a pliers. Please handle the display with care and grab it on the edges if possible. You dont want to scratch off some of the exposed SMD parts on the backside of the display.



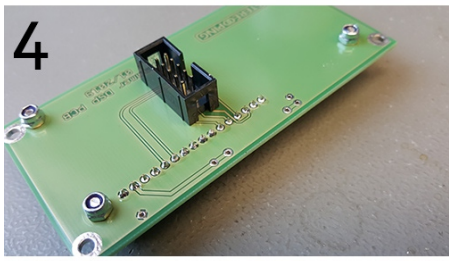
1. The connector is placed ! on the backside ! again, indentation pointing toward the Stereoping logo. Solder it in place. This step MUST be done before soldering the display itself otherwise the solder pins for the connector would be blocked.



2. turn around the PCB, put the pinheader into it's place (long legs pointing away from the PCB), attach the display and secure it with 4 safety nuts from the other side. Not TO tight – as the securing nut touches the PCB and you feel a little resistance it's enough. Turn the PB for the display to look towards you, the contact row should fall on the PCB by gravity.



3. Now you can solder the contact row of the display.

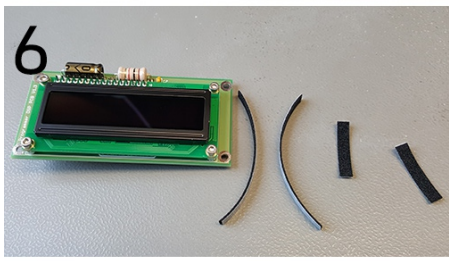


4. Solder the other side of the pinheader.

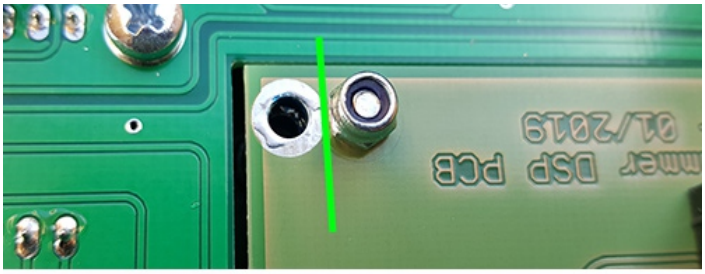
5. add the remaining parts. Mind the polarity of the big 470µ Elko! The inductivity works in both directions.

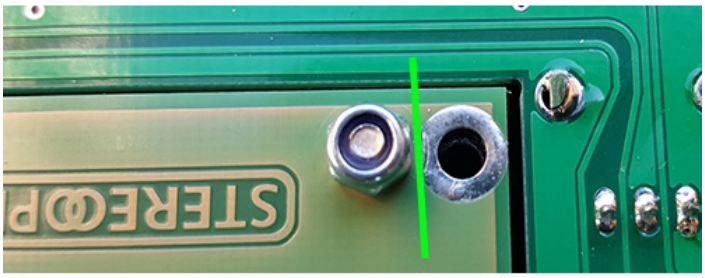
6. cut the self adhesive foam strip into 4 suitable parts. You need to snip it in half along the long side.

7. remove the protection foil of display and stripes and glue the 4 stripes onto the metal part of the display. Keep distance to the display glass of about ½ to 1mm. It does not matter if parts of the foam strip are hanging over outside the metalframe.



One last little thing to do: the display will be attached to the case using 4 screws. Two of them are quite near to the nuts of the display. To avoid the screws to scratch on the nuts making it diffficult to get them in turn the nuts with a pliers for their flat side to point into the direction of the hole:



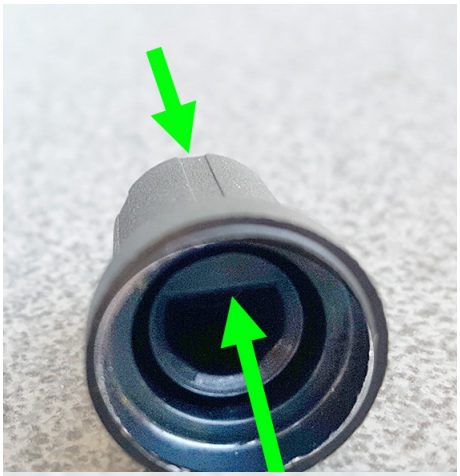


6. Assembling the knobs

As you purchased a DIY Kit, there is another step: assembling the knobs and their caps.

The 4 encoder caps do not have any markings. We suggest you start with them to get a feeling how easy/difficult it is getting knob and cap together.

Now each knob got a flat side for the D-axis. One of the 6 indentations on the outside of the knob exactly matches this D-axis side.



This is the indentation where to align the cap's marking.

If you press the cap into the knob using both thumbs ...

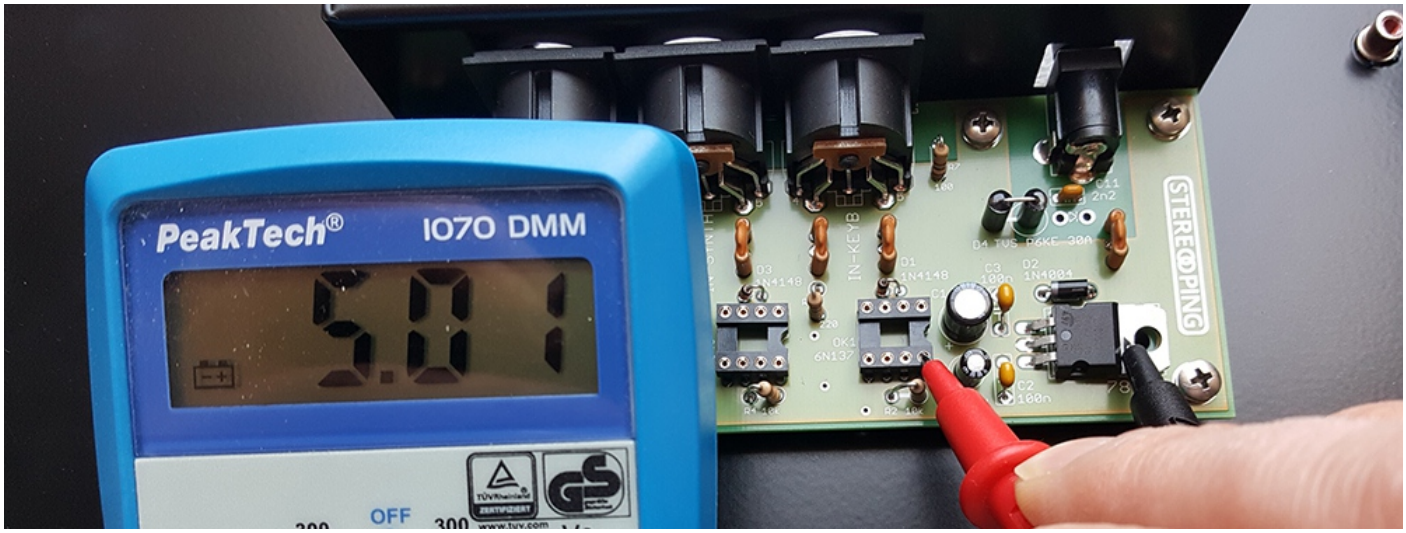
... or if you prefer to put the knob on the table and press the cap into position with one thumb (or whatever) is perfectly up to you.

If you happened to screw the alignment: press a cutter's blade between knob and cap to hinge the cap from the knob for the next try.

7. Checking voltage and getting everything together

Checking voltages

Before putting the ICs into position we should check the 5V to be ok. Screw the Power Midi board into the lower half of the case and plug in your power supply. The Multimeter's black power cable can be held onto the metal part of the 7870 regulator. The red cable will be held to pin 8 of one of the optocouplers. You should get 5V here. (Picture shows still old Power Midi PCB)

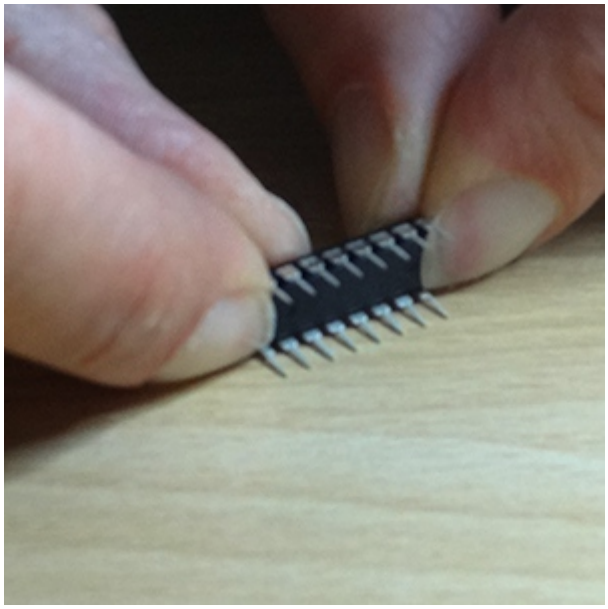


Now connect the main PCB with the flat cable to the Power Midi board. Be sure to use the right cable plug onto the right connector. Although you can not do any harm technically, the cable fits better in one direction than the other. It should point away from the Power midi PCB as shown in the picture to the end of this chapter.

Pin 16 of the Multiplexer sockets should also provide 5V. It's enough just to test one point.

ICs plugging

There are 7 ICs witch 16 legs. Six of them are identical. They carry the printon 4051 and will be pushed into the sockets between the pots. One of the chips is labeled 74HCT165 (or similar). This IC should be placed into the socket near the encoders.



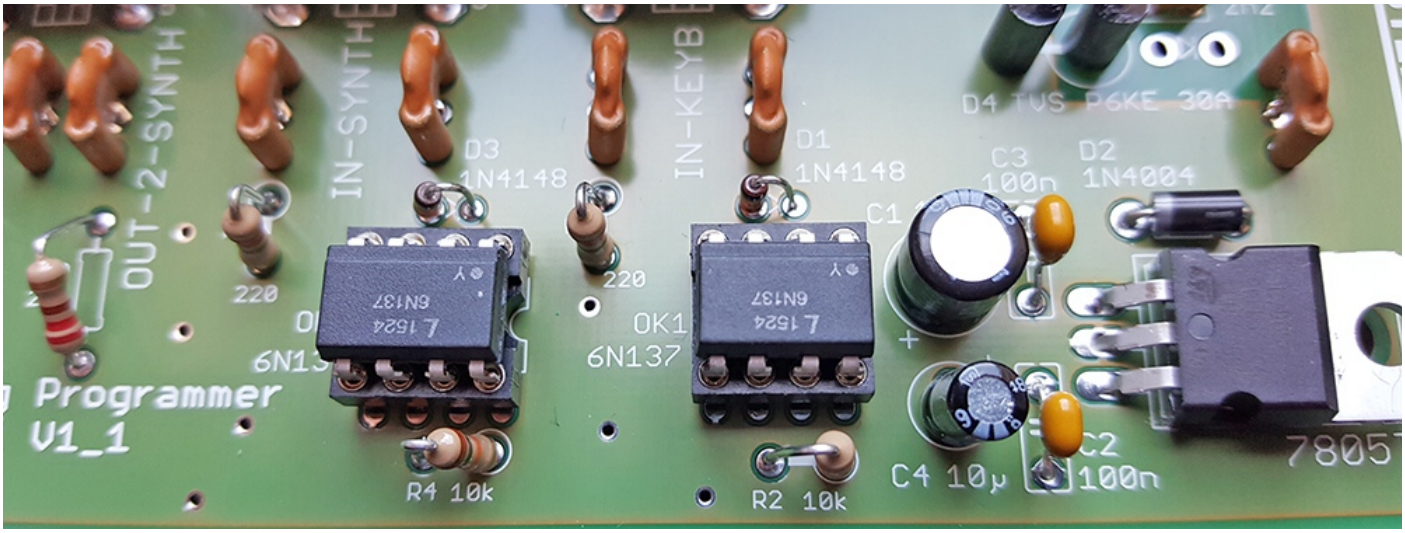
Please remove the power supply as we place the ICs now into position. Coming from factory, the IC's legs often are bent much to wide to fit the sockets You need to bend them together a tiny bit.

If the IC's legs do not fit, one or more legs could be bent completely crazy or break apart while pressing the IC into it's socket!

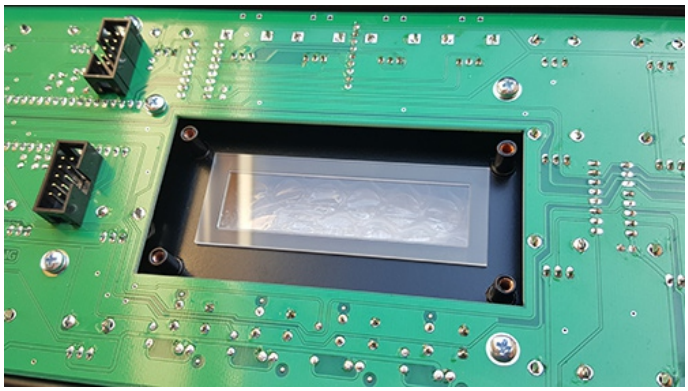
To match the IC's legs with the socket's holes you can do the following: grab the IC with both forefingers and thumbs from the top on it's short sides. Then press it on the table with the complete leg row and bend the whole leg row a bit inside by twisting the IC carefully away from you. Try to not only bend the thin part of the leg, bend the WHOLE leg which starts directly at the devices body.

When pushing them into the sockets mind the orientation and push them in the whole way down. On the main PCB the orientation is quite clear: all 16-pin ICs got their indentation towards the top edge of the PCB. The big MPU got it's marking to the left with the sticker's text running normally from left to right.

On the Power Midi board the ICs are sitting upside down:



Screwing main and display PCBs



First attach the main PCB. Before you add the display you have to put the plexiglass window into position. It is quite likely there is some dust on the window's inside and the display which you would see later. If you got some airspray you can blow it off. What we can not recommend is trying to blow the dust off with your own breath, especially after having eaten dry cookies.

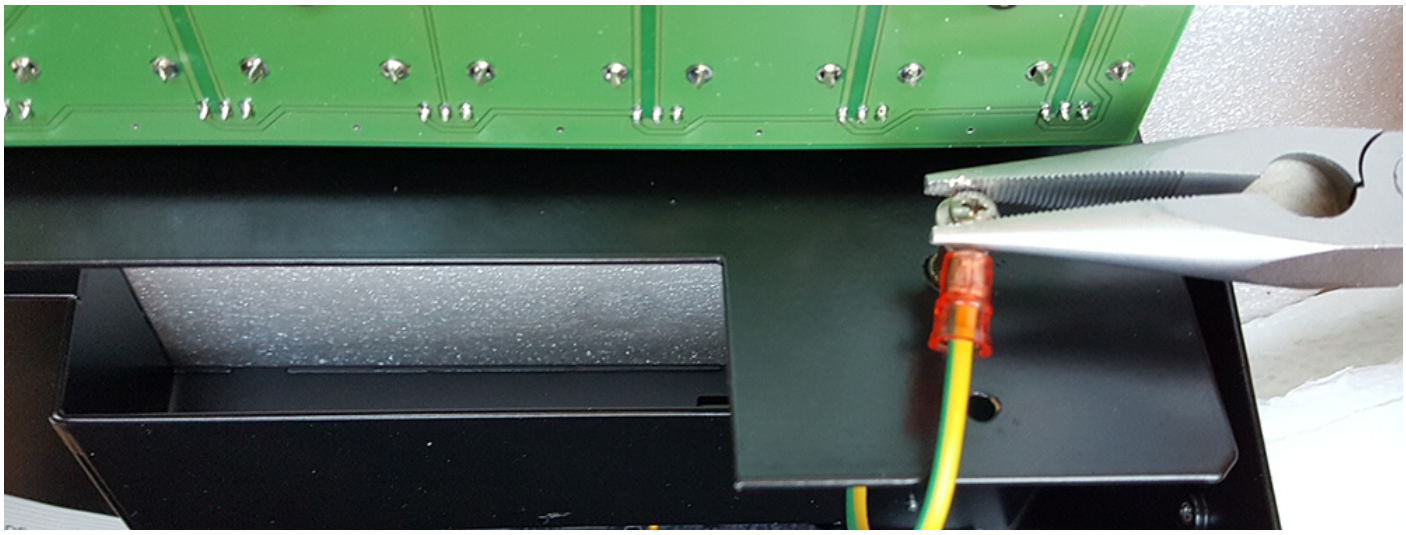
We'd advice just to add 2 screws first. Turn the case and check there is as less dust as possible between window and display. Remove the display agani to get the dust off (or new dust added) or attach the rest of the screws. Do NOT turn them to tight, it's enough if you a feeling aclear resistance.

Normally the screws are secured with a chemical substance to avoid they get loose over time. If you are worried, you can glue the tops of the screws to the PCB with a drop of suitable glue. Unless you take the Programmer with you on daily travels to your gig in an ancient stagecoach we don't see any reasons to panic about loosening the display.

Screwing the ESD-cable

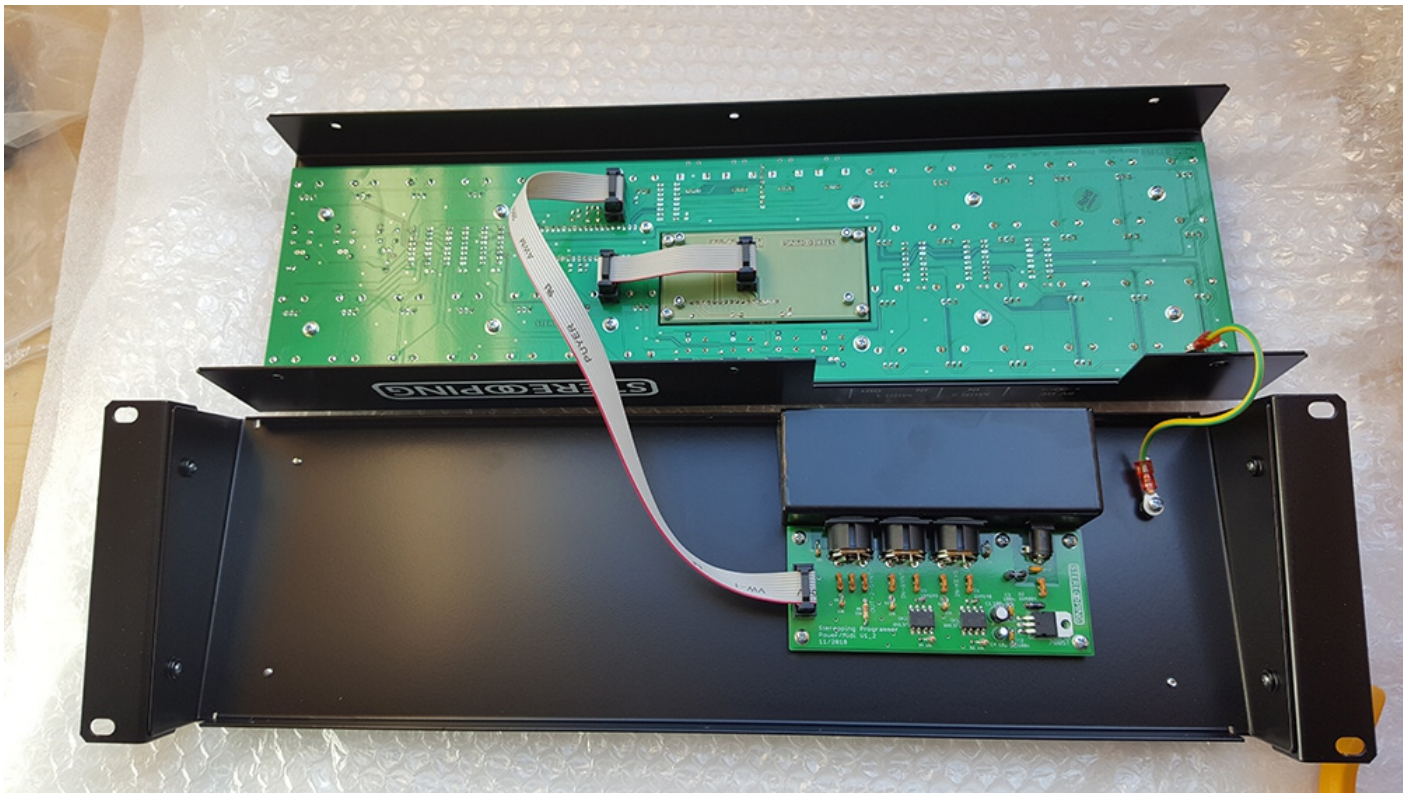
The ESD-cable is kind of a lighning rod – it connects upper and lower case part to deflect static charge to the power supply jack for protecting the electronics. Put one of the chopper discs between o-connector and screw.

The ESD-cable screw of the upper case is a bit more difficult as the opposing case wall is in the way when trying to use a screwdriver. It's easier to fix it with a pliers as shown in the picture.



Finish

This is how it should look now:



Now you can put both halfs of the case together. Mind the ESD and the flatcable to fold nicely together when attaching the upper half. It would not be good if one or both got clamped between the case parts.

Concrats, that was it. You are the greatest :-)

Firmware upload

Now we blow some life into the device by uploading the firmware. The firmware files are free to download on the Stereoping website. Use one of the free SysEx-dump tools like e.g. 'MidiOX' on PC or 'SysEx Librarian' for Mac.

- Switch off the Stereoping Programmer
- Connect **MIDI OUT** of your Midi-Interface directly to the jack **MIDI 2 IN** of the Programmer using a short cable of good quality
- There are some preferences in most SysEx-Dump Programs like 'Delay between Buffers' or 'Delay after F7'. Please choose something around 100mS (Milliseconds) here.
- Hold the first button of the row of 4 above the display. Power up your Programmer. The Display should read *'BOOTLOADER V1.0 - OS to MIDI IN 2 ...'*
- Load the new Firmware into your SysEx-Dump Program and send it out
- The display shows OS-Version and progresst. After about 2 minutes the Programmer restarts
- If you will get an error on the screen please try another cable or increase the delay mentioned above.

8. Imprint

Adress Gregor Zoll Musikelektronik
Hermann-Voss-Str. 21
59872 Meschede
Germany

E-Mail hello@stereoping.com

Homepage http://www.stereoping.com