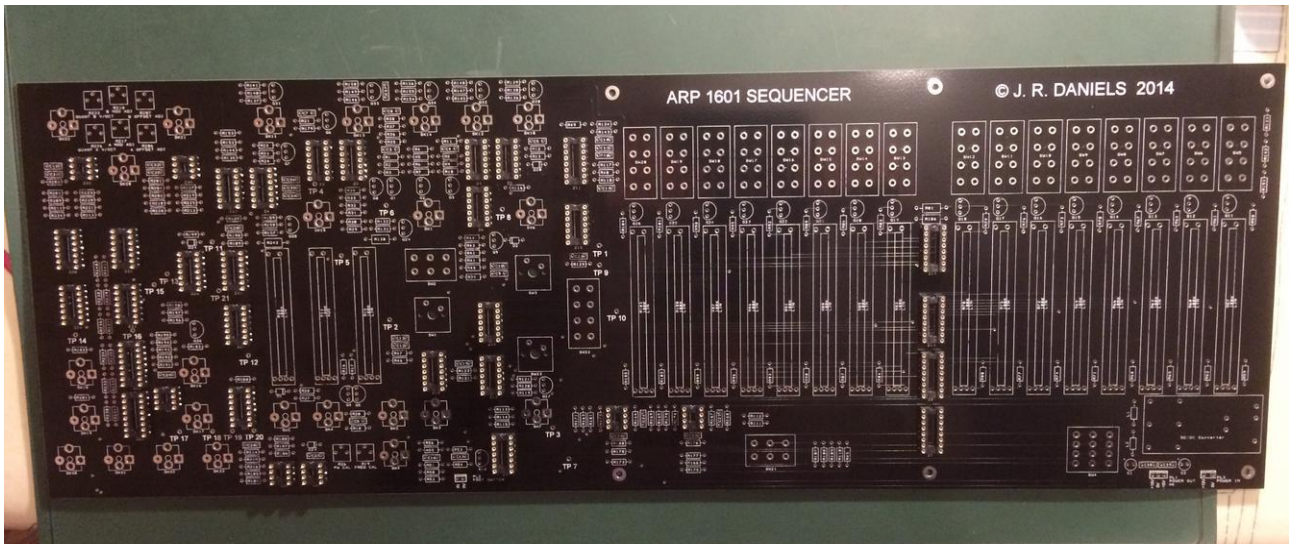
ARP 1601 Sequencer Clone

Build Guide (Rev 1 or 2 PCBs)

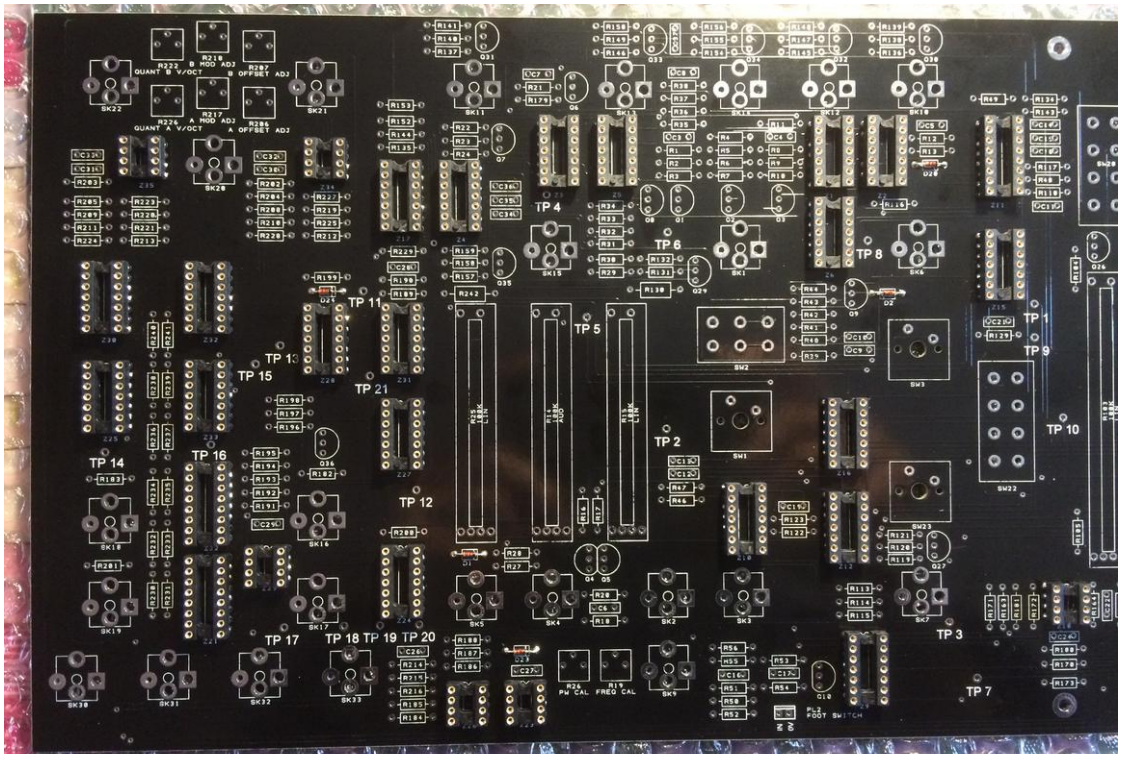
The only difference between Rev 1 and Rev 2 PCBs is the addition of a 12uH inductor and 47uF electrolytic capacitor before the DC/DC converter in the power supply section.

See the end of the document for recommended mods. Also mentioned at the appropriate stages.

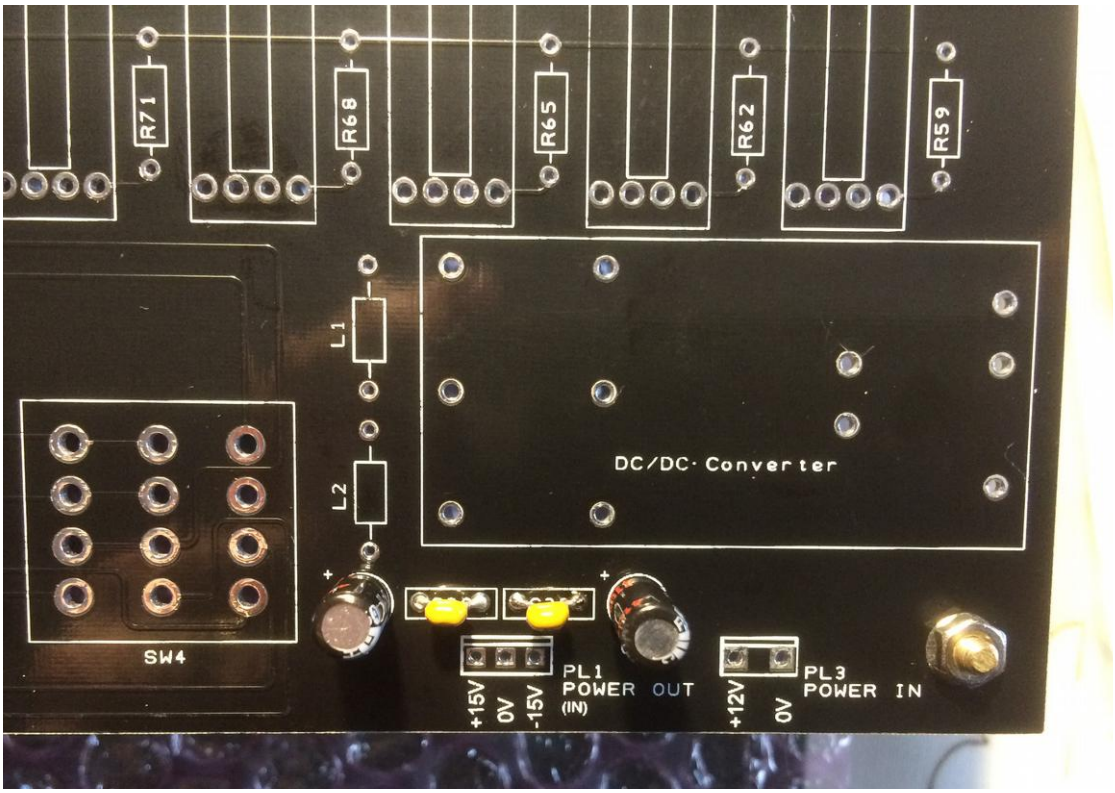
1. Solder in all the IC sockets if you're using them. Note that the notch is always facing upwards, so Pin 1 is always towards the top of the PCB. That's 400 pins soldered already in approx one hour, a quarter of the total pin count.



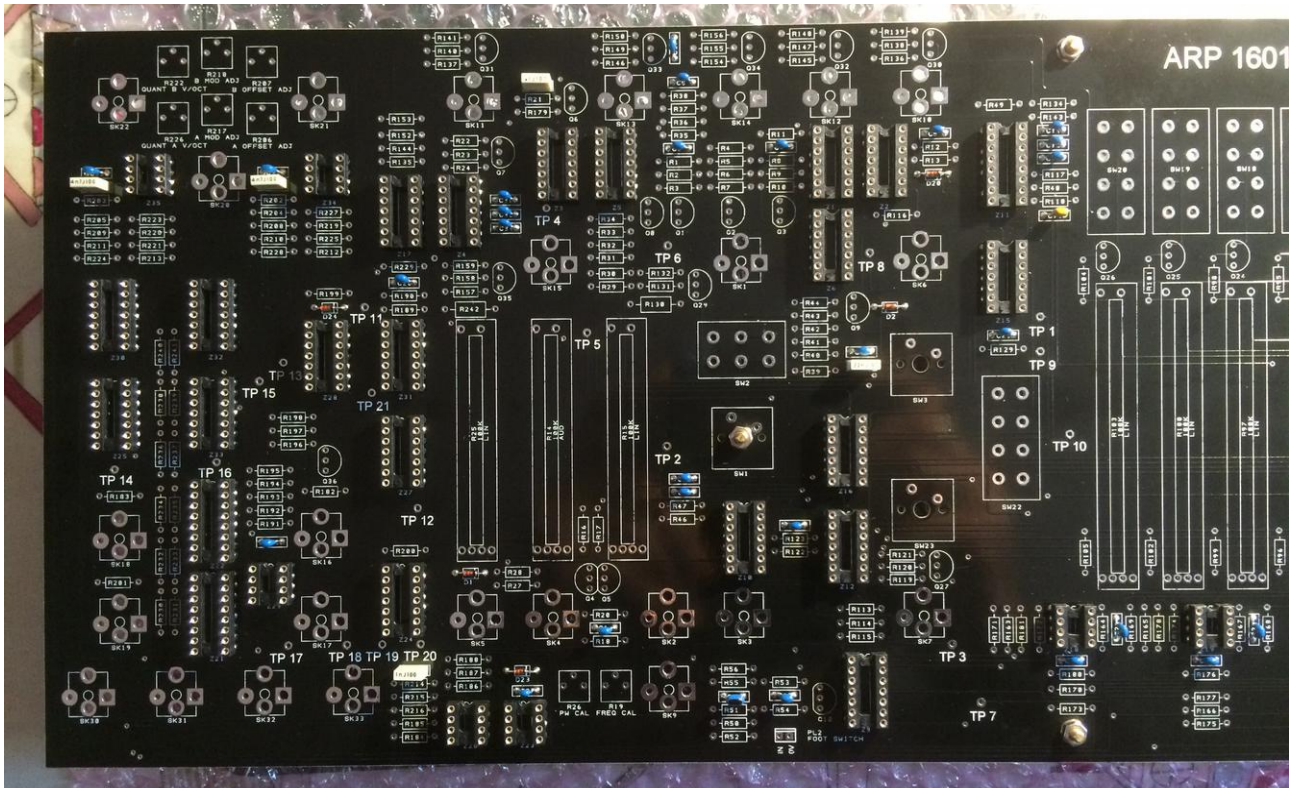
2. Add the five diodes. Check orientation – black line on diode to white line on PCB (pointing to the right on all but D2 which is to the left). Note that the diode numbering is not D1 to D5 inclusive, as most of the diodes are LEDs and form part of all 19 sliders.



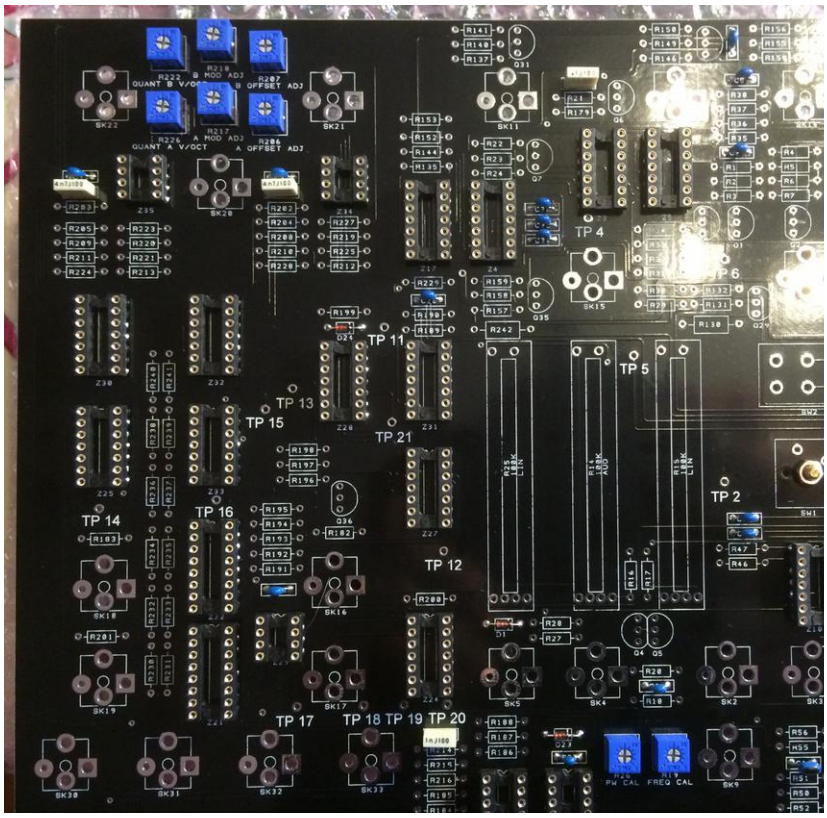
3. Add the power supply smoothing capacitors. Note orientation of the two electrolytics, and the two 100nF X7R types. For Rev 2 PCBs, L3 and C40 are fitted by PL3.



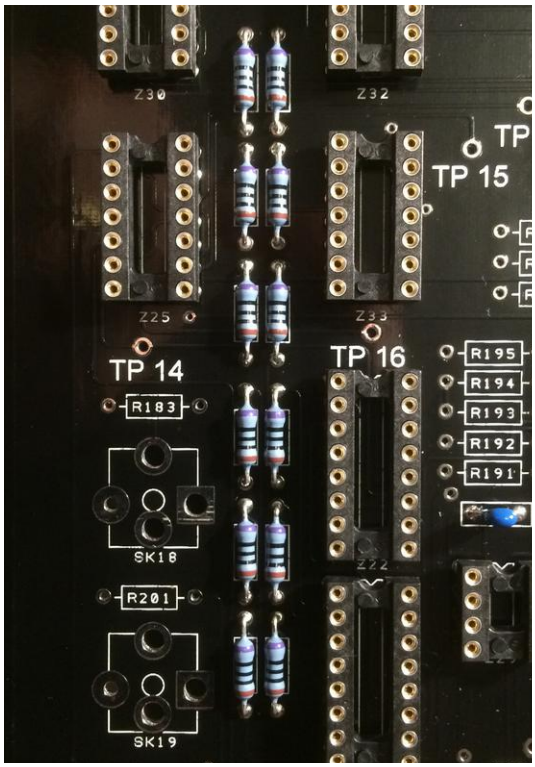
4. Add the remaining capacitors. Note the location of the film-type (grey) caps and the X7R (yellow) type in the photo. **Mod: Instead of a 10nF C0G/NP0 capacitor for C5, use a 33nF or 39nF, whichever is available. There will be another capacitor to add later.**



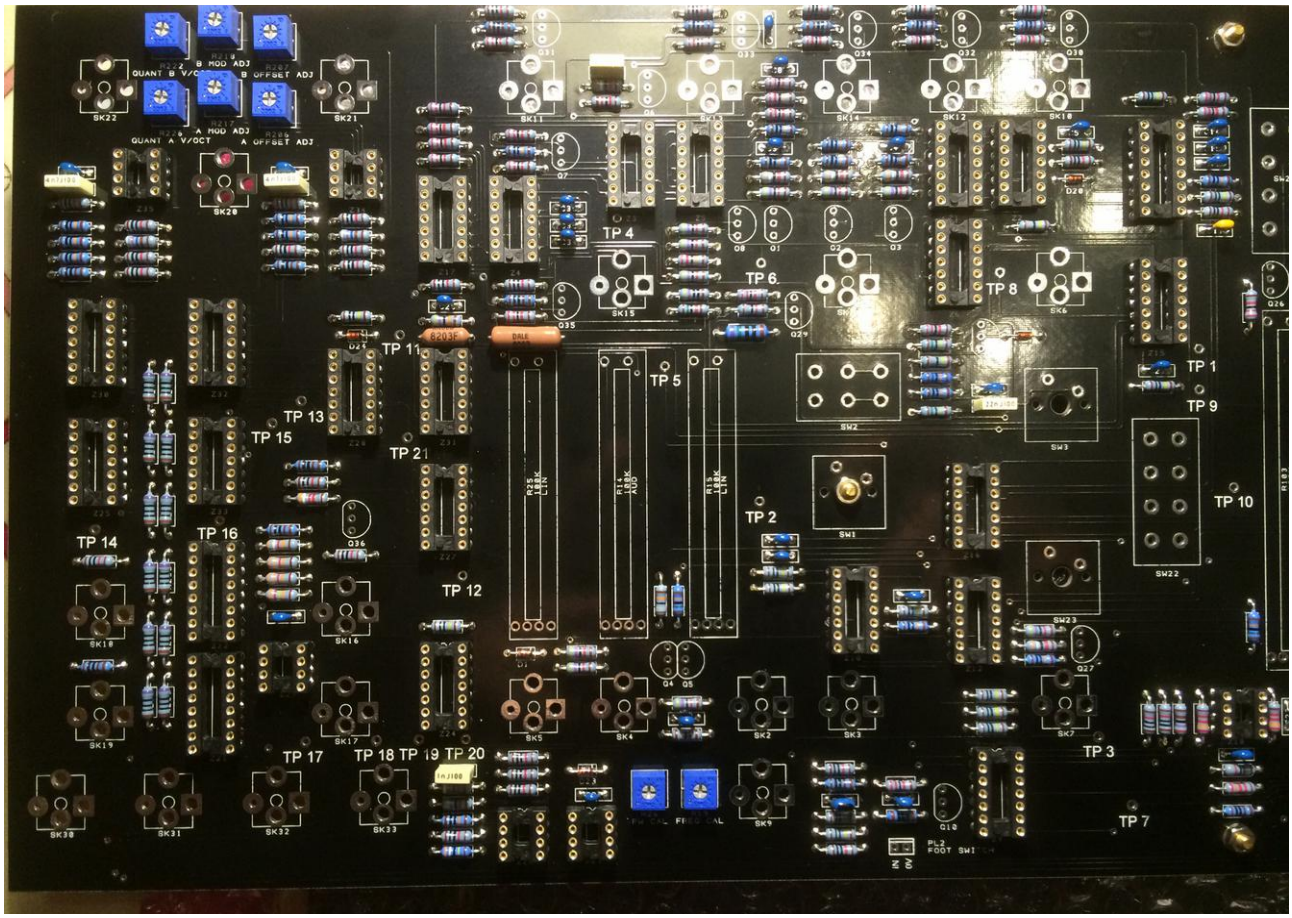
5. Add the eight trimmers. Leave approx at half-way setting on each. Note there is some doubt whether two of them (R217 and R218) should be 1k or 2.5k. The original schematic shows 1k and the parts list 2.5k. From the final position after tuning, 1k seems to be the more appropriate value. Both are in the BOM should you wish to experiment. **You need 4 x 652-3362P-1-102LF (1k) which go in R217, R218, R222 and R226; and 4 x 652-3362P-1-104LF (100k) and these go in R19, R26, R206, and R207. Note that the BOM contained an error (now corrected) – you will need 4 x 1k and 4 x 100k trimmers (not 2 and 6 respectively).**

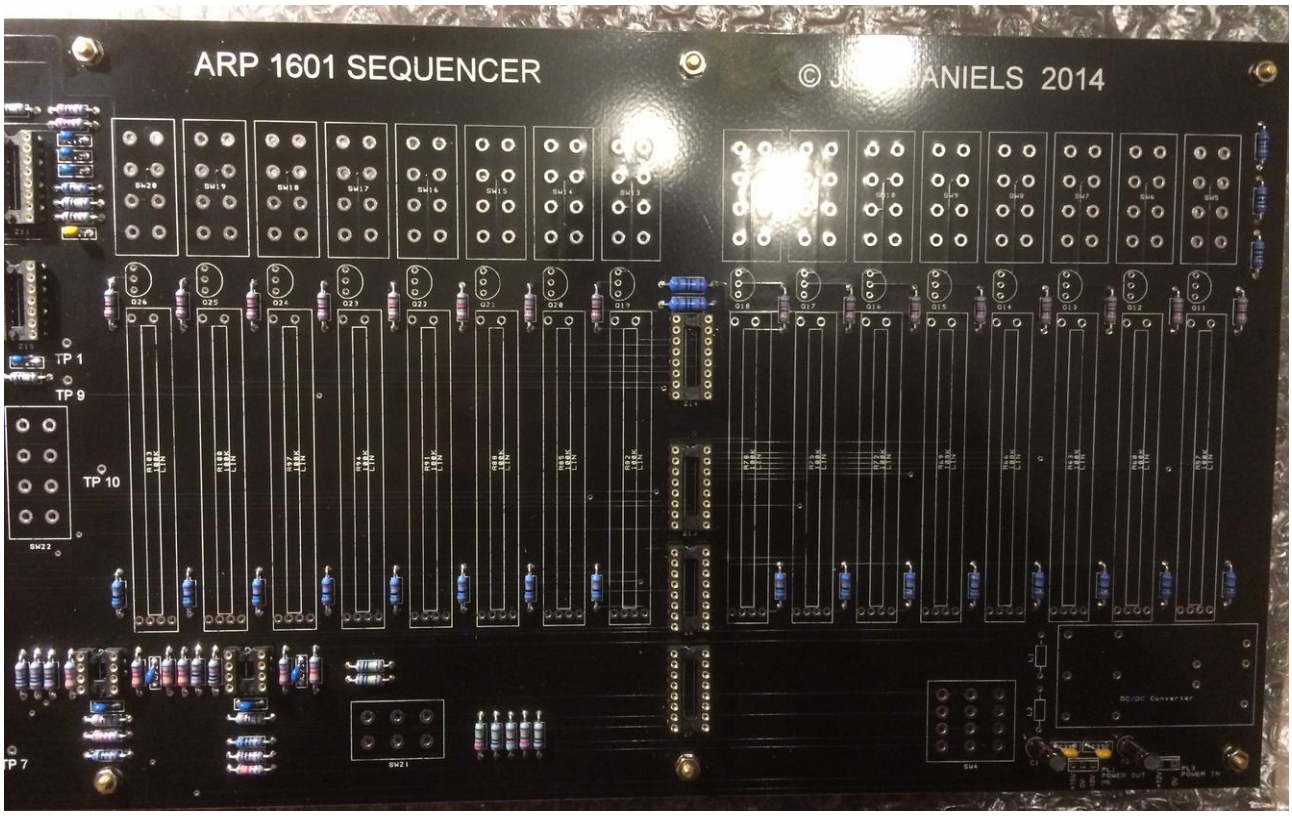


6. Add the twelve 100R precision resistors which make up the equivalent of the quantizer resistor pack Z20 in the original design.

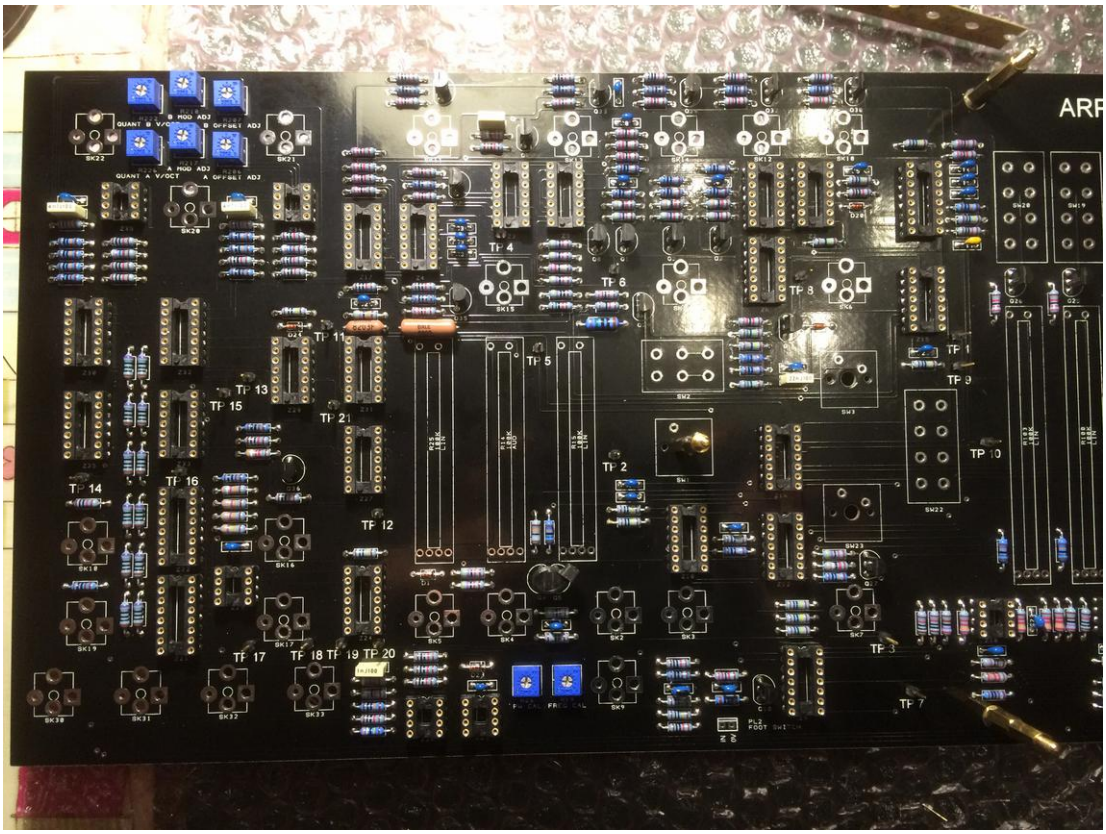


7. Add the remaining resistors. Note the three (larger) ½-Watt resistors for the LED chains (R81, R106 and R242). I have substituted the oversized ceramic body resistors in recent versions of the BOM – they should fit perfectly well now.

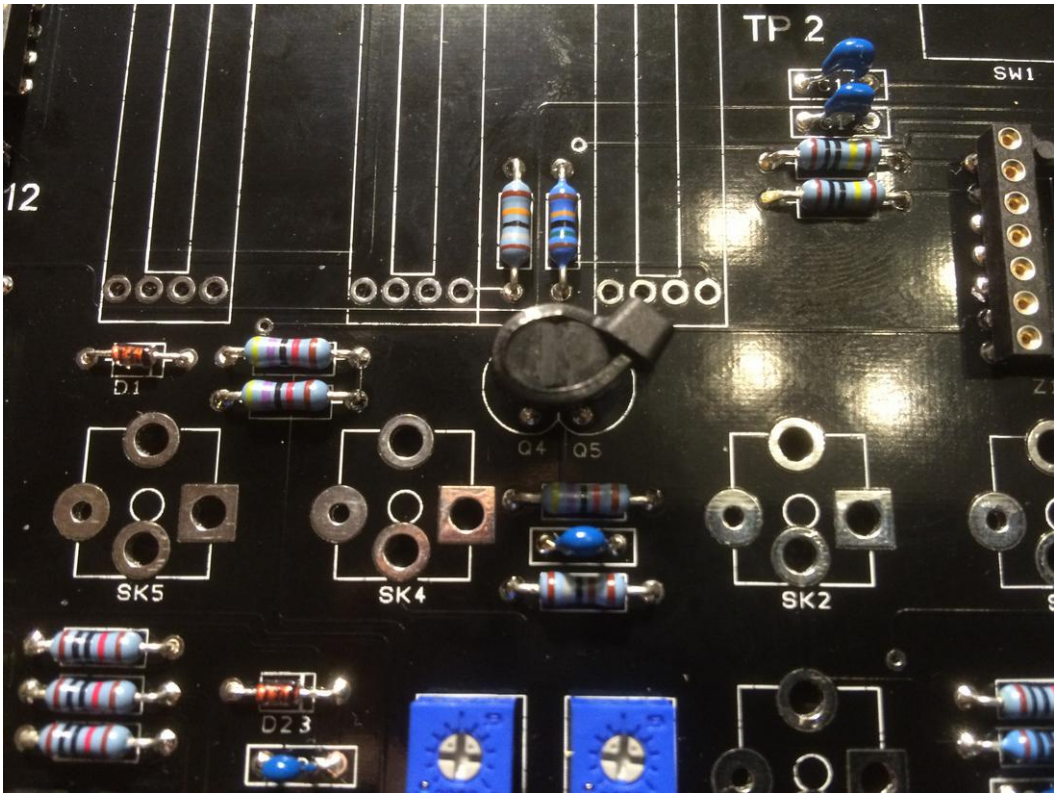




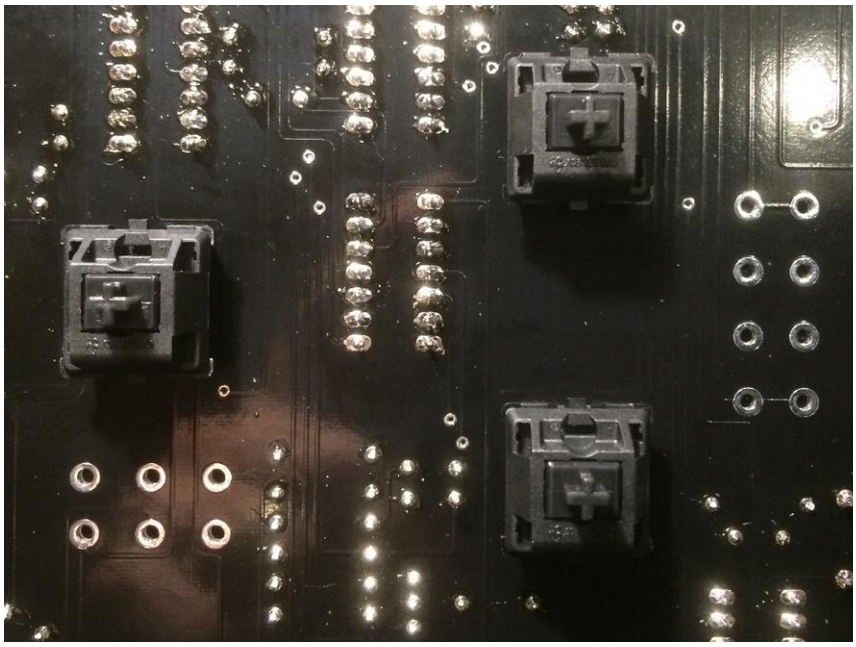
8. Next up the transistors. There are only two types – 2N3904 (NPN) and 2N3906 (PNP). No need to match any of them, but if you can try to get Q4 and Q5 roughly similar in spec, noting that Q4 is a 2N3906 and Q5 a 2N3904.



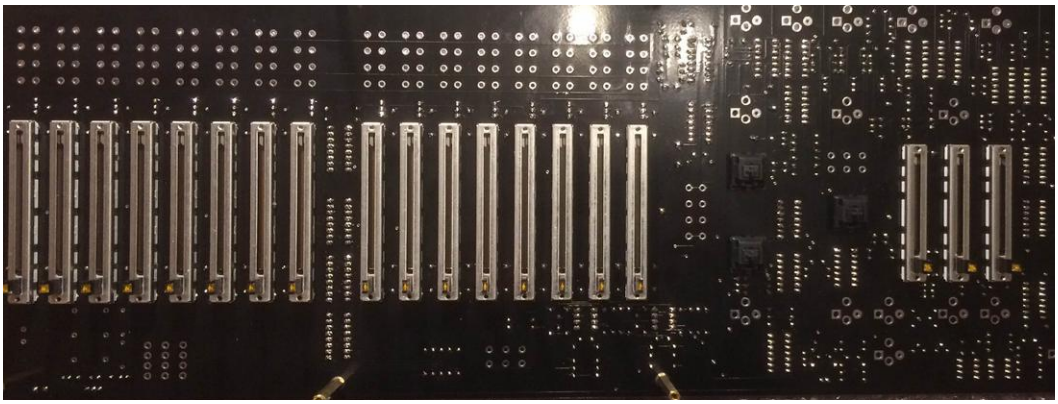
Nevertheless, Q4 and Q5 need to be pushed together for thermal coupling, and held with a cable tie or similar



9. Next flip the board and push in the three Cherry push-button switches flush to the board and solder in place. You may wish to melt the ends of the thin plastic mounting lugs over with the soldering iron to permanently fix them in place but I didn't find it necessary. **Mod: Solder a 1nF MLCC capacitor directly across the SW1 terminals – eliminates contact bounce/multiple trigger issues with the start/stop push-button.**

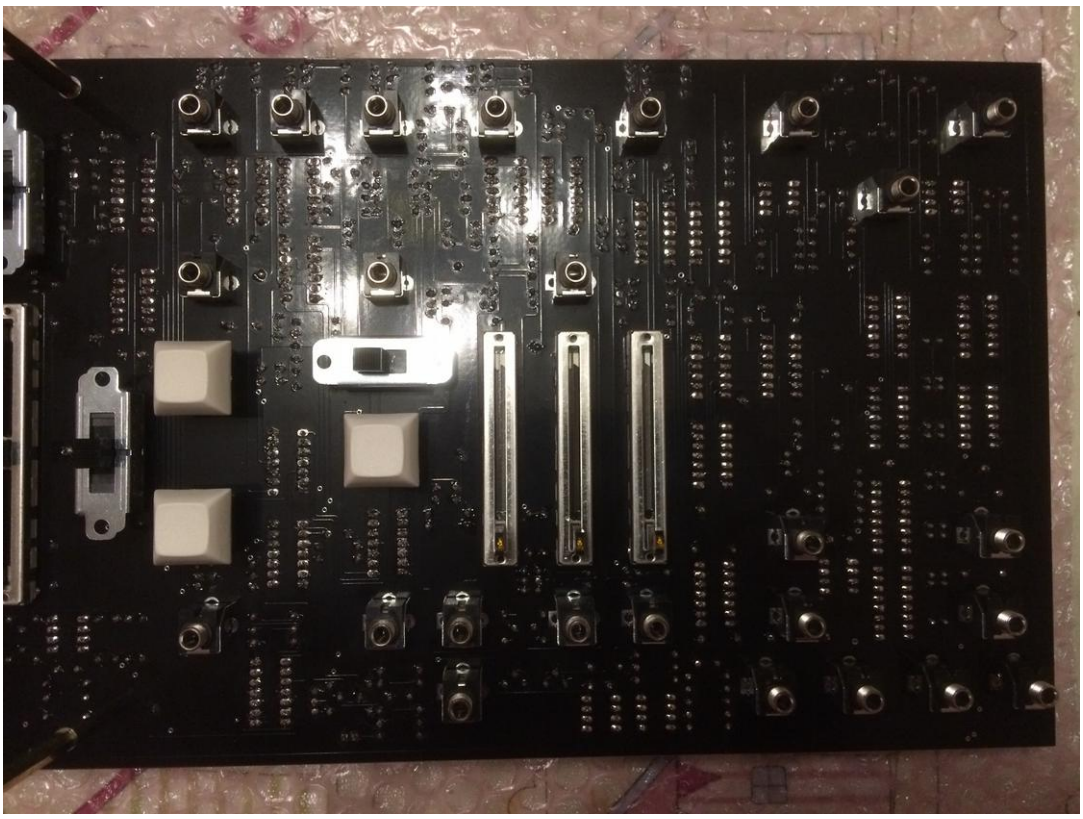


10. Next up is the 19 sliders. Note the 45mm 100K AUD one (R14). All the rest are LIN. Make sure you push each one firmly down to the board when soldering the first couple of pins to keep it absolutely vertical to avoid fouling on the panel slot.

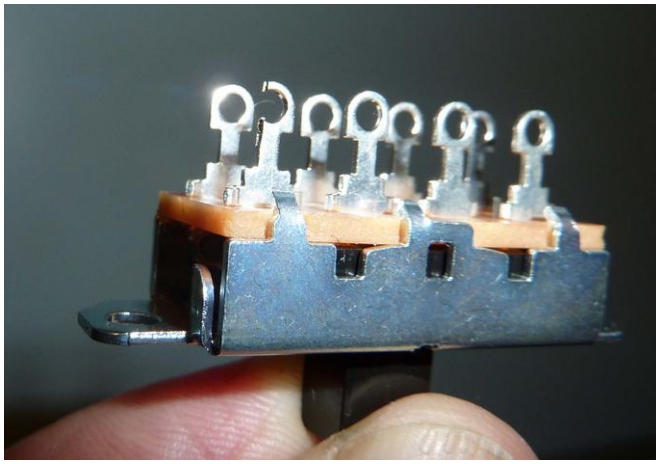


11. Next check the clearance on all drilled holes on the front panel for M3 screws and the jacks, relieve slightly if necessary using a handheld taper reamer as the powder coating may make each one a little tight. Fit 6 x M3 x 12mm* stand-offs to the 6 PCB mounting holes and place all 25 jacks into the PCB loosely. Offer up the front panel over the jacks and loosely hand-tighten the jack nuts and also fix the PCB to the six stand-offs. Keeping the panel flat and pressing down gently on the PCB solder all jacks into place making sure they are all flush to the PCB. Remove the jack nuts and take the PCB off the front panel again. Note the orientation of all the jacks is the same. Picture shows push-button key caps fitted and slide switches pushed into place as well – we'll do these next.

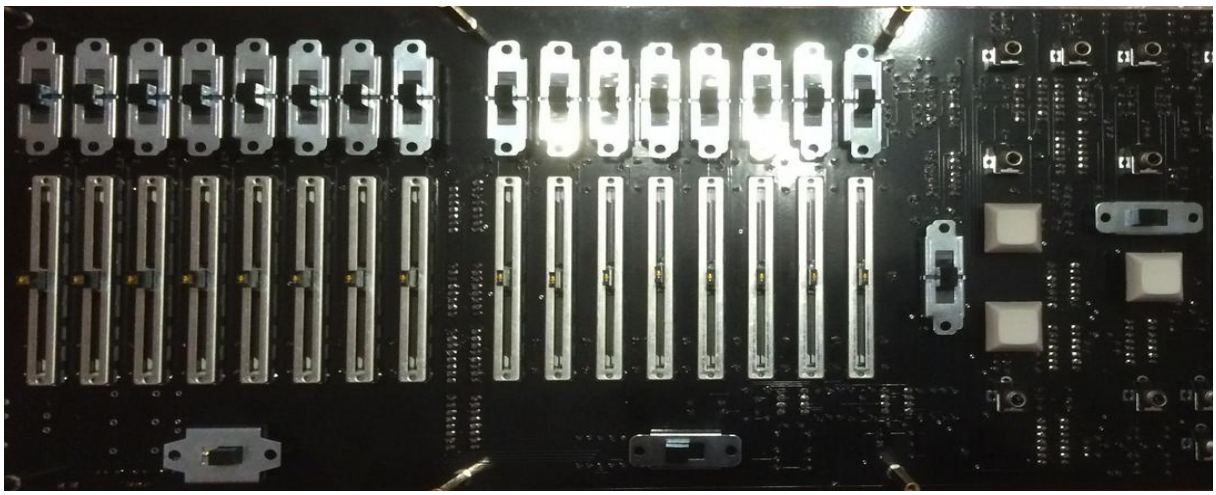
* If you can find them, 11mm stand-offs work a little better (closer to the height of the jacks), or carefully turn or file off 1mm from each.



12. Now the fiddly bit begins. You will need to trim off the sides of every solder lug on every switch with sharp side-cutters, that's 160 pins in total. Do not snip off the lug horizontally as this will leave the pin a little too short so snip off each side as shown in the picture. File each one as necessary to remove any burrs and to make a slight taper to aid fitting it to the PCB as the pin spacing on these switches can be very slightly variable. Half of first pin shown – now do the other side of the pin and repeat this another 159 times.



13. Push each slide switch into its PCB pads gently – file the pins a little more if still a tight – then fix front panel back onto the stand-offs and jacks again. With the panel inverted and supported, perhaps using more stand-offs as I did, carefully press down each slider flush against the front panel, again being careful not to distort the front panel. Carefully lift the panel and PCB assembly to check each switch sits centrally in its panel aperture, then solder all pins in place.



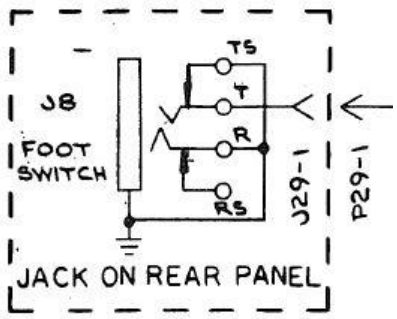
14. Fit the three headers, the 21 test point pins (from the snap-off 0.1” header strip), the DC/DC converter and two ferrite beads on wire links for L1 and L2. The three-pin power header (PL1) then becomes an Aux supply for anything else you may wish to power, up to the rating of the DC/DC converter. The 1601 draws only circa 64mA on the +15V rail and 34mA on the -15V rail so there's plenty to spare for other uses should you wish. If the ferrite beads are too long, trying halving them with a Dremel and slitting disc – a hacksaw won't touch it.

If using a linear power supply omit the 2-pin power connector (PL3), the DC/DC converter, L1, L2 and L3 and C40 and use the three-pin +15V/0V/-15V header PL1 as your power input.

15. Observing strict static precautions, insert each IC into its appropriate socket, being very careful to make sure each one is in the right socket and is the right way round. Pin 1 (the notched end) is always pointing towards the top of the board with the screen-printed text the right way up as in all the photos above.

16. Wire the main power switch appropriately – if using a DP switch for a linear power supply DC output rails you'll need to file out the aperture accordingly.

17. Wire the lead from PL2 to the ¼” stereo foot-switch jack is such that it is shorted out when no plug is inserted, with the IN pin to the T(ip) contact in the diagram below and 0V to the TS, R(ing) and barrel contacts, and that the foot-switch itself must therefore be a normally closed momentary switch, or latching on-off.



Now connect up the power and try it out. If all is well use the set-up procedure in the Service Manual to correctly adjust it.

Mods Summary: Replace C5 with a value of 33nF or 39nF, and add a 1nF MLCC directly across the pins of SW1 to eliminate the contact bounce issue.

Then have fun, lots of fun!! Enjoy.