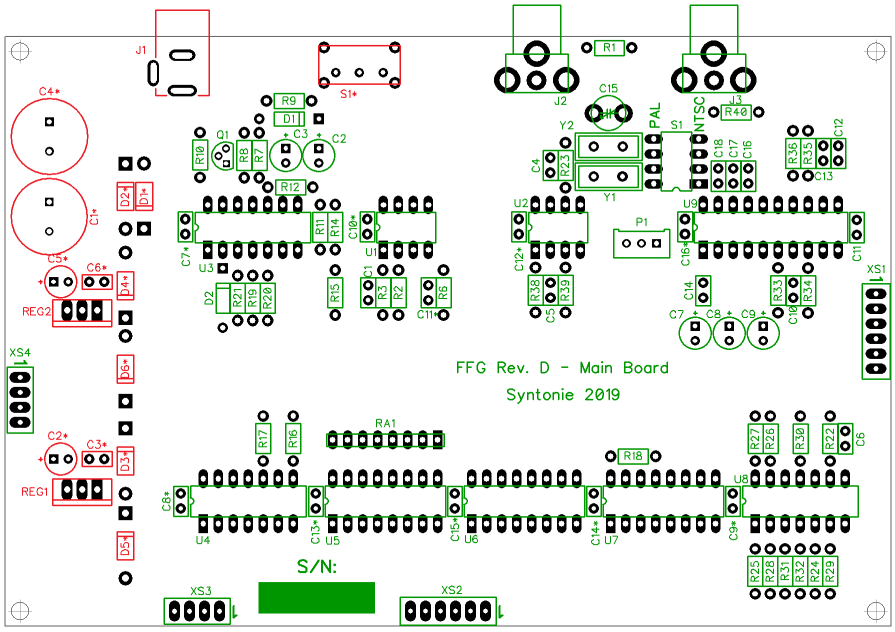
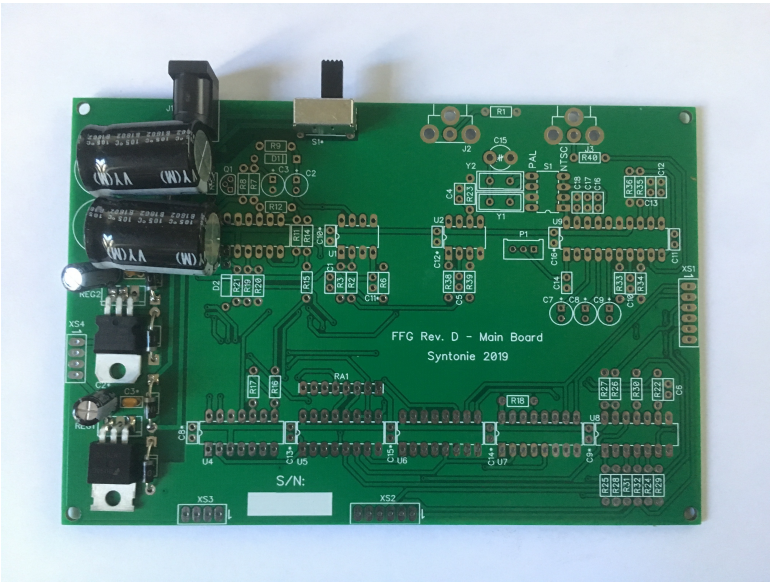
FFG REV D
BUILD GUIDE

A. Main Board

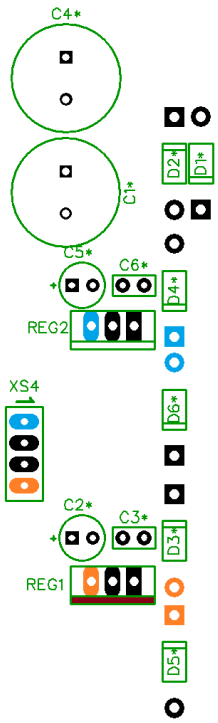
I. Power Supply

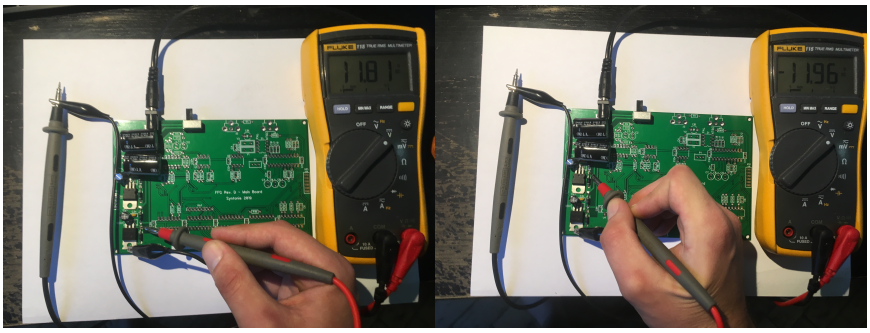


First part we will build is the power supply, it's important that it works right cause all the circuitry after will depend on it. To save space, mind to bend the big electrolytic caps and the regulator parallel to the board, as shown on the picture. Also note that electrolytic caps are polarized, make the longer leg of the cap match the "+" mark on the board/square pad, diodes are also polarized, the white band on them must match the white line on the diode silkscreen.



Part list:
D1*, D2*,D3*, D4*, D5*, D6* : 1N4001
C1*,C4*: 2200uF (Electrolytic, 35V)
C2*, C5*: 100uF (Electrolytic)
C3*,C6* : 100nF (Ceramic, marked 104)
REG1: 7812
REG2: 7912
J1: DC connector
S1*: SPDT slide switch EG1224

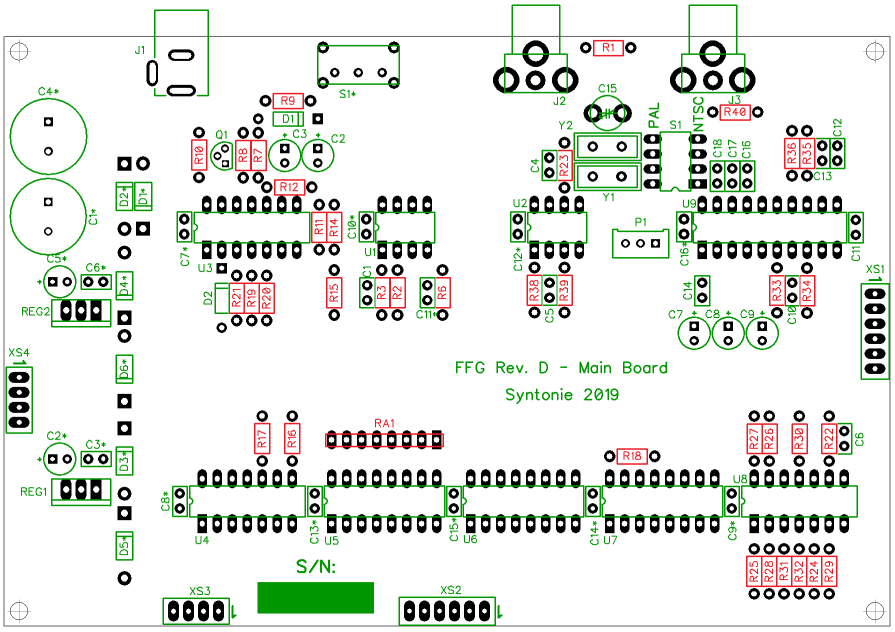




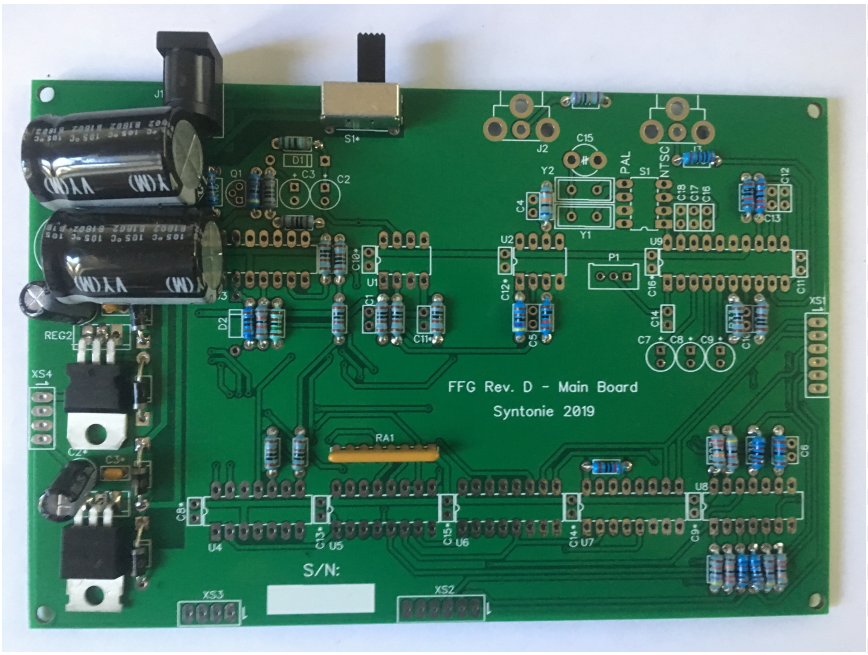
Once everything is soldered and the components leads cut, we will test the power supply. Plug a 15V**AC** power supply into J1 connector, turn S1* to the left (DC connector side) , the circuit should now be powered on. Using a voltmeter, black probe to ground (best is to use the regulator tab, shown in brown) and red probe to test, check that you have the right voltages.

Orange is +12V, blue is -12V. Now be careful as the 2 big capacitors can hold a charge when circuit is turned off, avoid shorting the two pins of either capacitors.

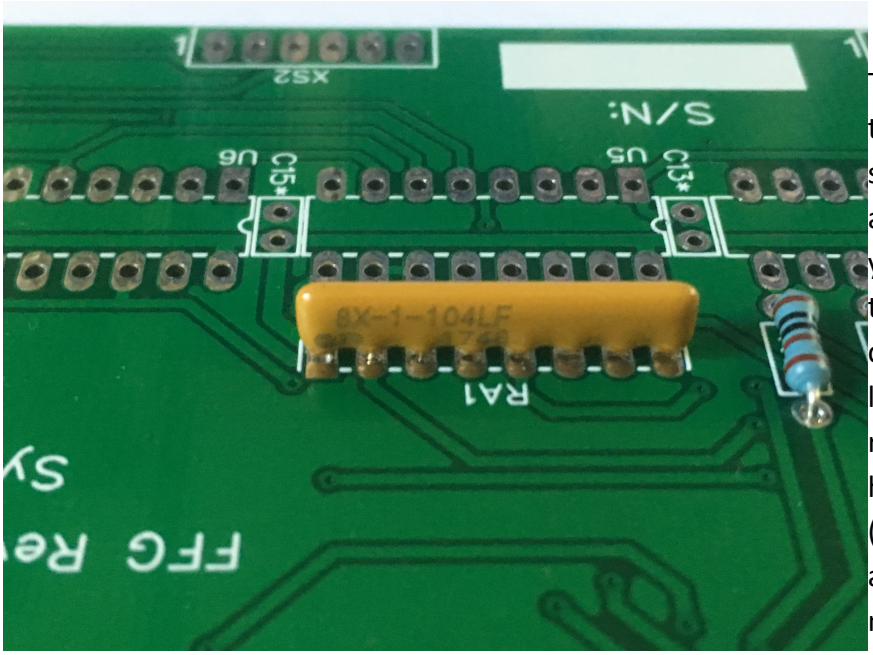
II. Resistors



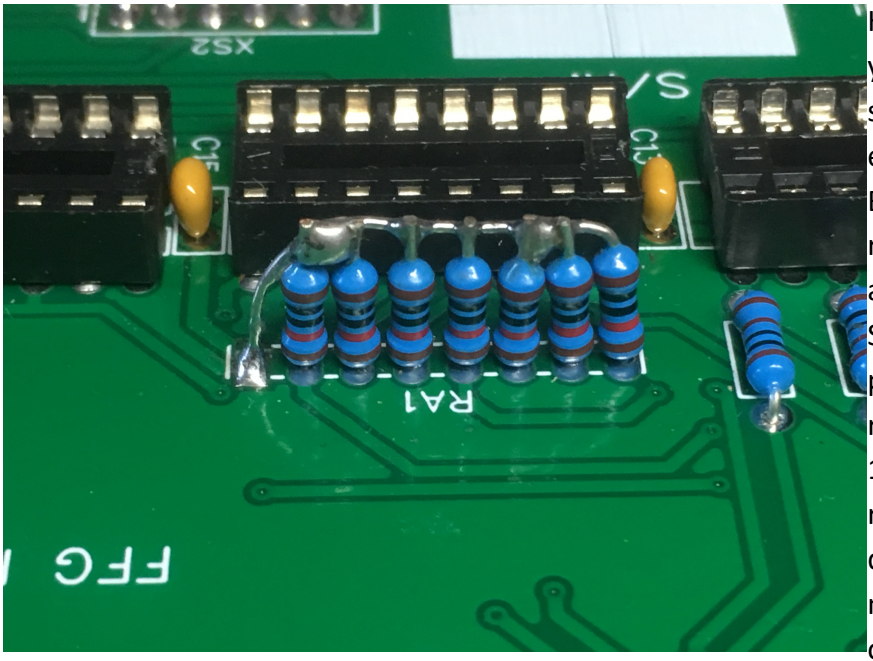
Place all the resistors, those are not polarized. RA1 is a bussed resistor array, make the dot on the body of the array match the square pad on the board. It is included in the kit, however, if you're building it by sourcing your own components, you can also use 7x 680R resistors as shown on this picture.



R1: 75R
R40: 100R
R10, R21: 680R
R18: 820R
R2, R3, R6, R7, R11, R12, R14, R15, R16, R17, R33, R34: 1K
R20: 1K5
R35: 2K2
R9: 3K9
R8, R38: 4K7
R30, R31, R32: 6K2
R36, R39: 10K
R19: 12K
R22: 15K
R24, R25, R26: 47K
R27; R28, R29: 220K
R23: 680K
RA1: 680R 7x resistors bussed array

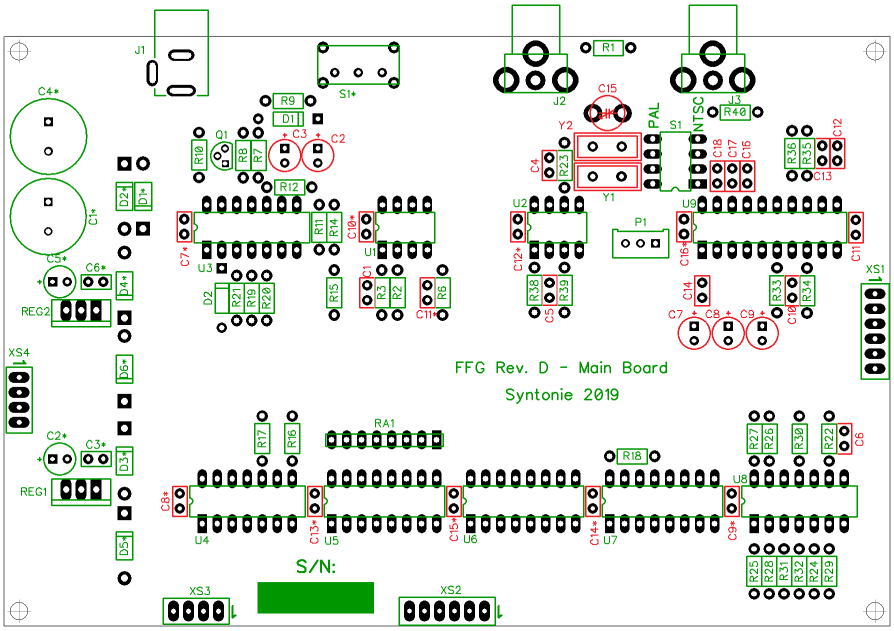


This is the resistor array, make sure the dot on the left matches the square pad on the PCB. Also, the array displayed is marked 104, yours should be marked 681. (Was testing with higher value resistors cause the 7x 680R resistors draws a lot of current, making the positive regulator and comparator chip heat, but using weak pullup resistor (high value), generates visual artifacts, that's why the value needs to be pretty low)

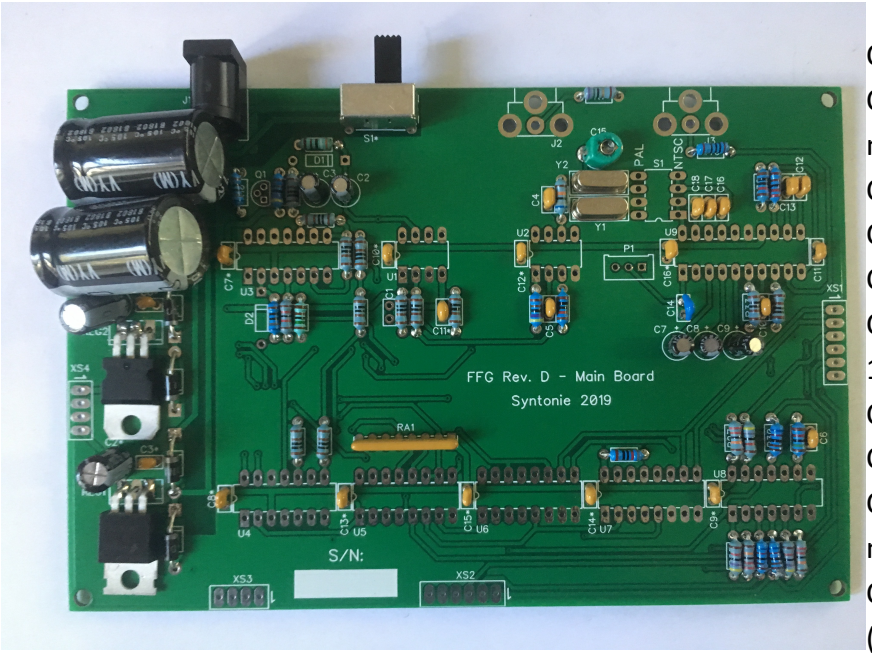


However, if you want to go full DIY, you can use 7x 680R resistors, solder one side of each resistor in each pad except the square one. Bend the remaining lead of the resistor on the right so it goes against all the other resistors legs. Solder this long leg to the square pad, and then solder it to the 6 remaining resistors. Yeah, those are 1k resistor (weak vs strong pullup resistors again), it works fine, but doesn't really helps the heating matter, so I went for 680R as on the original schematic.

III. Capacitors

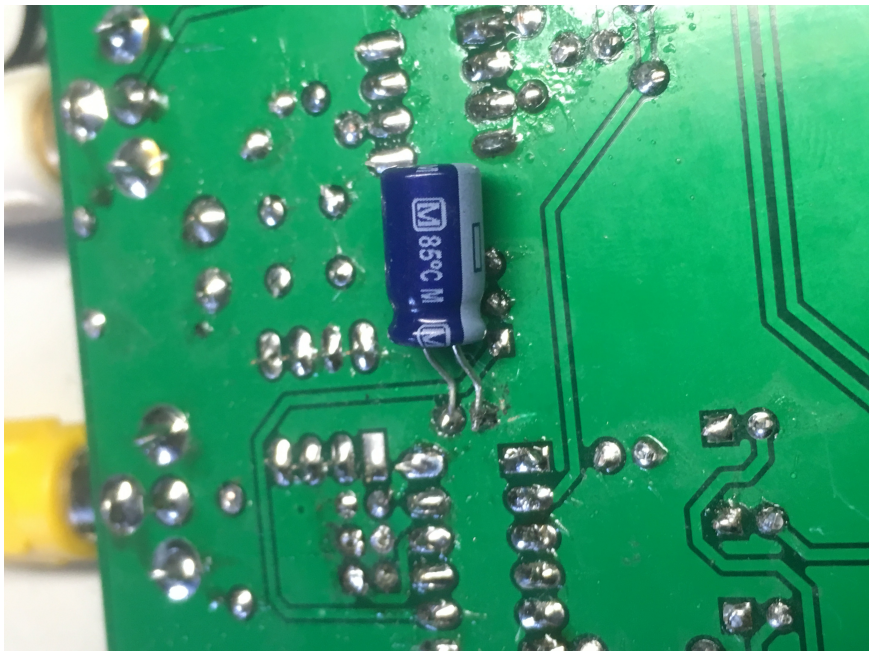


Place all the capacitors, again mind the orientation of electrolytics caps, and also the little asteriks, those are power supply caps, don't mix them with parts that don't have the asteriks. Also place the two crystal oscillators.



C15: 30pF variable capacitor
C16, C17, (C1): 220pF (Ceramic, marked 221)
C6: 470pF (Ceramic, marked 471)
C11: 1nF (Ceramic, marked 102)
C14: 1.2nF (Ceramic, marked 122)
C10, C18: 10nF (Ceramic, marked 103)
C4, C5, C12, C13, C7*, C8*, C9*, C10*, C11*, C12*, C13*, C14*, C15*, C16*: 100nF (Ceramic, marked 104)
C2, C3, C7, C8, C9: 10uF (Electrolytic, 35V)
C17*: 100uF (Electrolytic, 35V)
Y1: 4.43MHz Xtal (PAL)
Y2: 3.58MHz Xtal (NTSC)

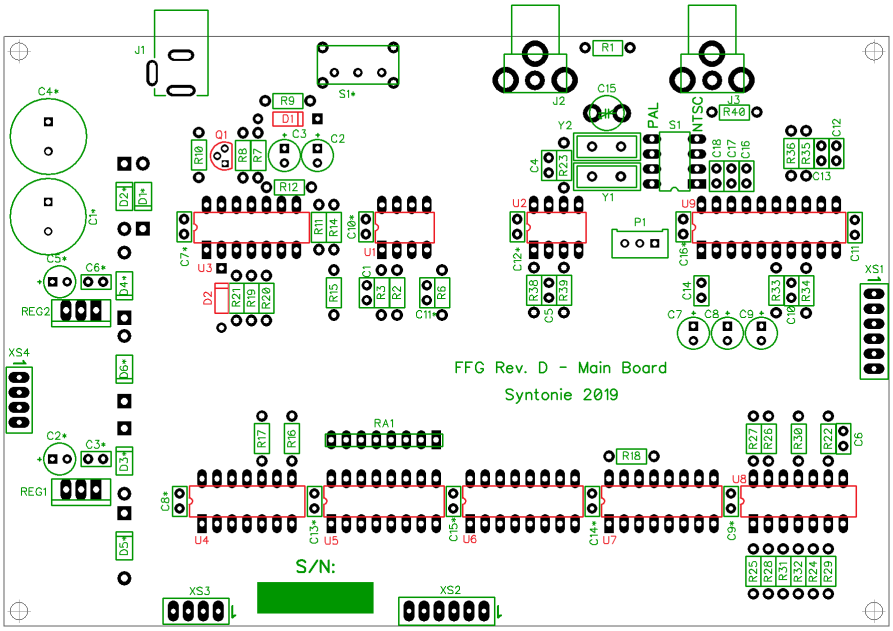
C1*, C2*, C3*, C4*, C5* and C6* have already been placed during the first part. Y1 and Y2 are not caps but crystal oscillators that will generates the subcarrier frequencies needed for the MC1377 to encode color to the output signal.



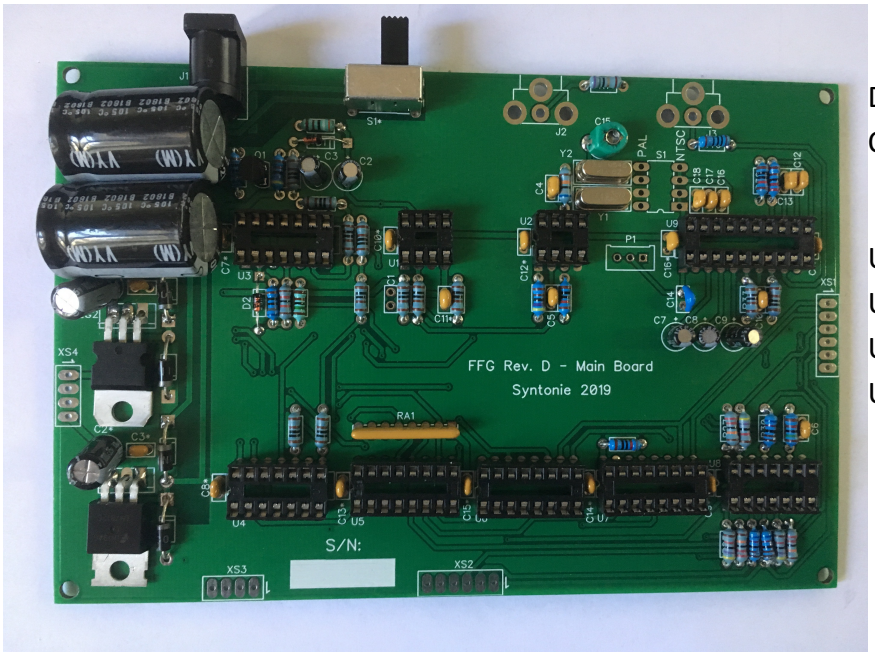
1. C17* isn't on the board, you need to solder it accros C16* on the other side of the board. You can also solder it when everything else has been placed on the mainboard, should be simpler.

C1 makes a low pass filter to remove the chrominance. 220pF helps a bit to filter the chrominance, however 470pF seems to be too much and makes the RGB encoder act weirdly. You can also leave C1 empty, leaving the signal unfiltered. Depending of the chrominance amplitude (= color saturation), it will false the comparators a little and might "bleed" into the RGB encoder (interfere with the chrominance generated by the oscillators) causing some alternating colors on saturated. It's particularly visible with a color bar at the input, less with normally satured pictures, and none with a black and white input.

IV. Diodes/Transistors



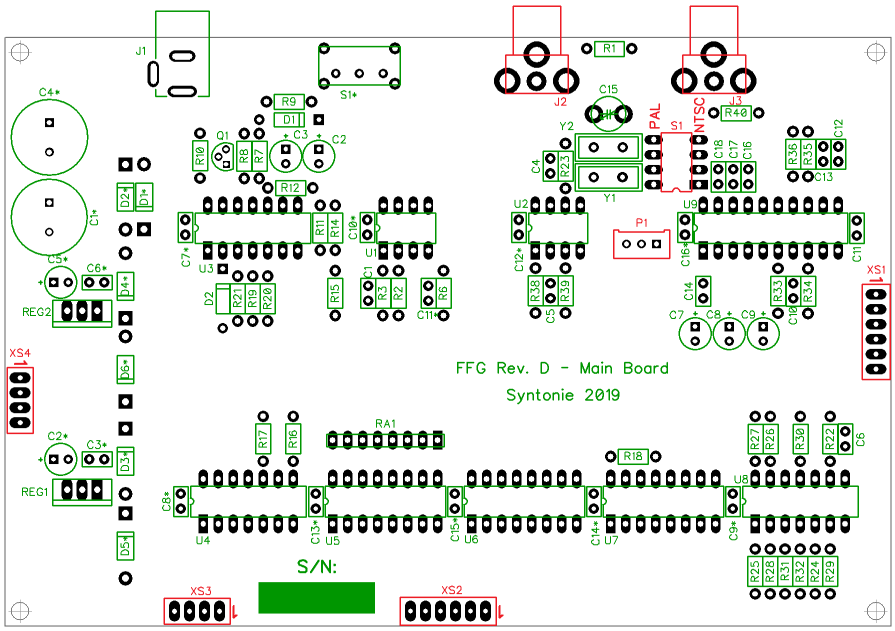
Place all the diodes (mind the orientation), the transistor and also IC sockets for each IC. Don't place the IC in their sockets yet. If you're not using sockets (which will surely be more reliable, but it will be more complicated to replace an IC in case of fault), do not solder them yet, we will test the circuit before that:



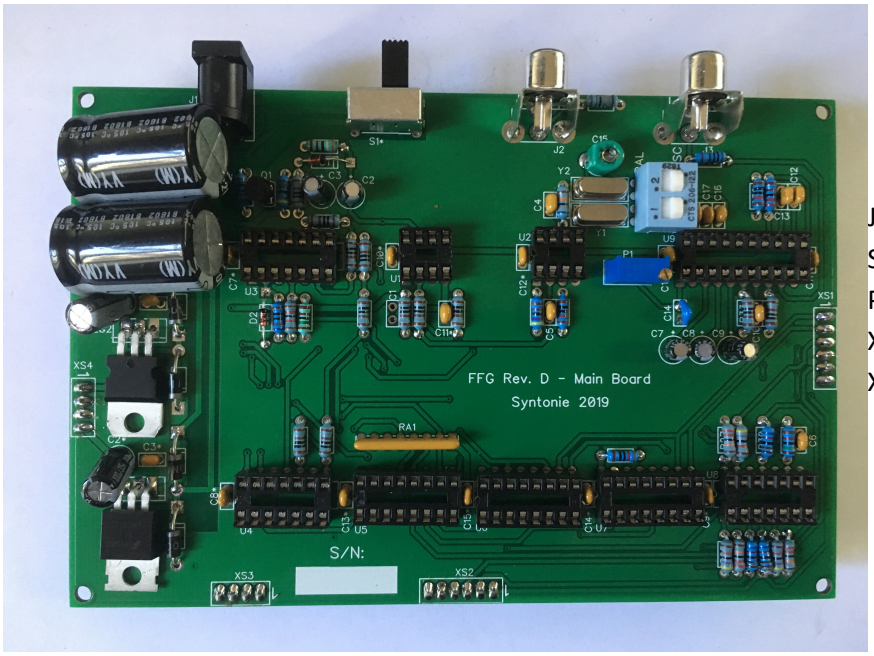
D1, D2: 1N4148
Q1: 2N3904

U1, U2: 8 pin DIP socket
U3, U4, U8: 14 pin DIP socket
U5, U6, U7: 16pin DIP socket
U9: 20 pin DIP socket

V. Hardware

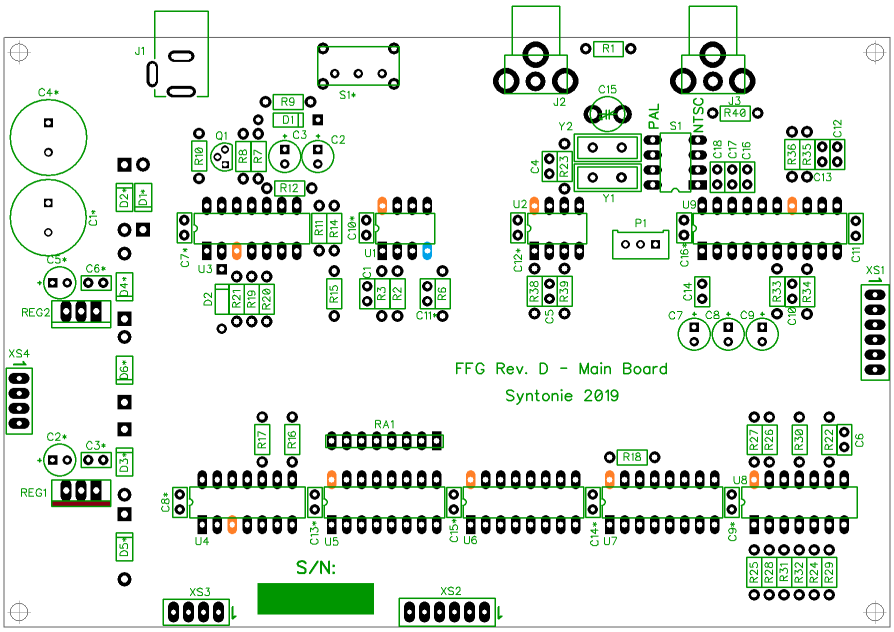


Now, we can place all the hardware parts. Note that XS1, XS2, XS3, and XS4 are soldered on the other side/solder side of the board, all other parts are soldered on the silkscreen side/component side.



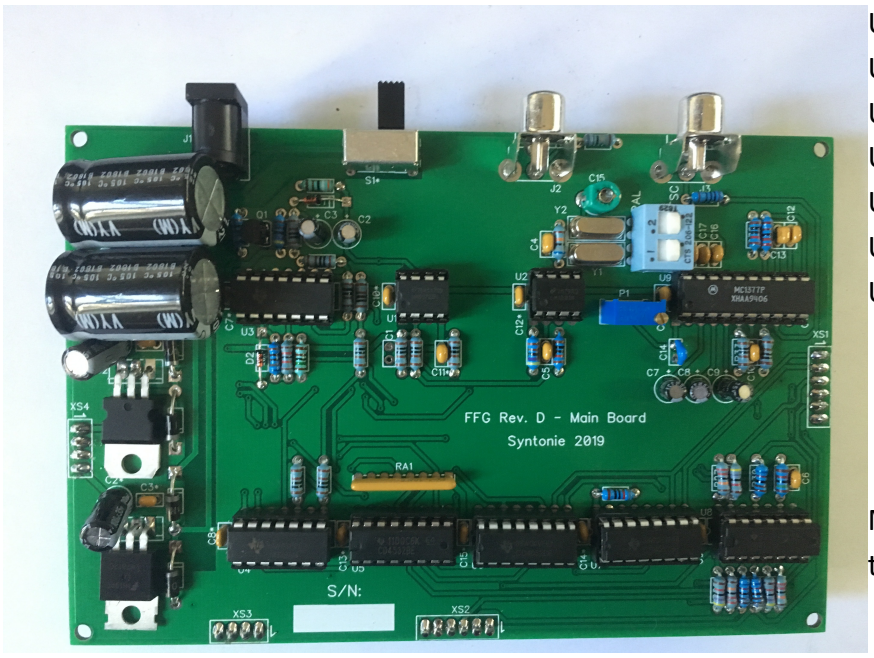
J2, J3: RCA connector horizontal
S1: DPDT dip switch
P1: 100k trimmer 3296W
XS1, XS2: 6 pin female connector
XS3, XS4: 4 pin female connector

VI. Integrated Circuits



Now that every components have been placed on the board except the ICs, we will test every IC supply pin to check that we have the expected voltages. Once again, with your voltmeter, black probe to the ground, use the red probe on the orange points shown on the board, you should read +12V. Now do the same for the blue pad, you should read -12V. If you can't read those value, check that every component is soldered at the right spot, with the right orientation, and that there is no short between pads or cold solder joints.

You can place the ICs :

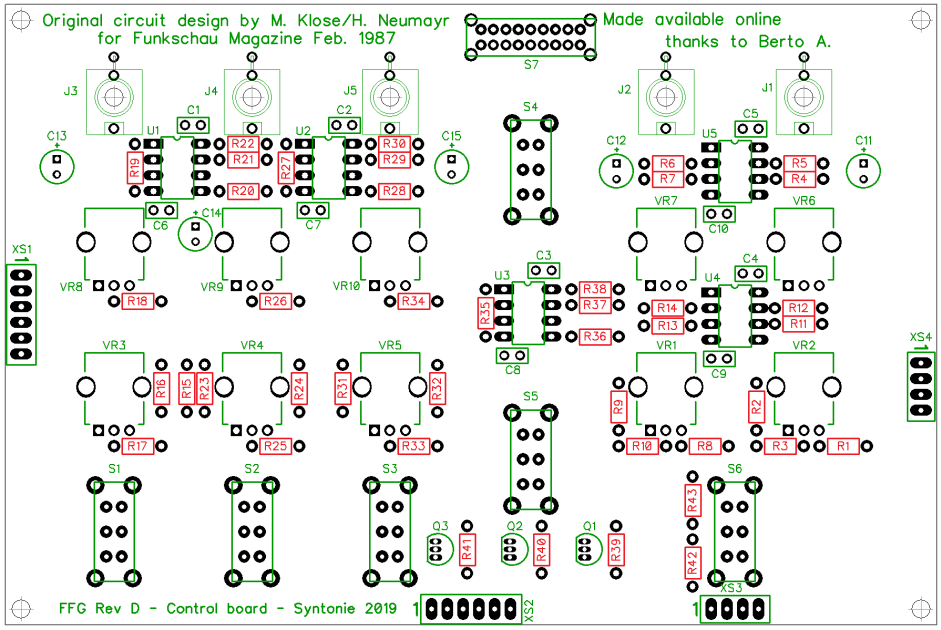


U1 : LM6172
U2 : LM1881
U3, U4 : LM339
U5 : CD4532
U6, U7 : CD4052
U8 : CD4070
U9 : MC1377P

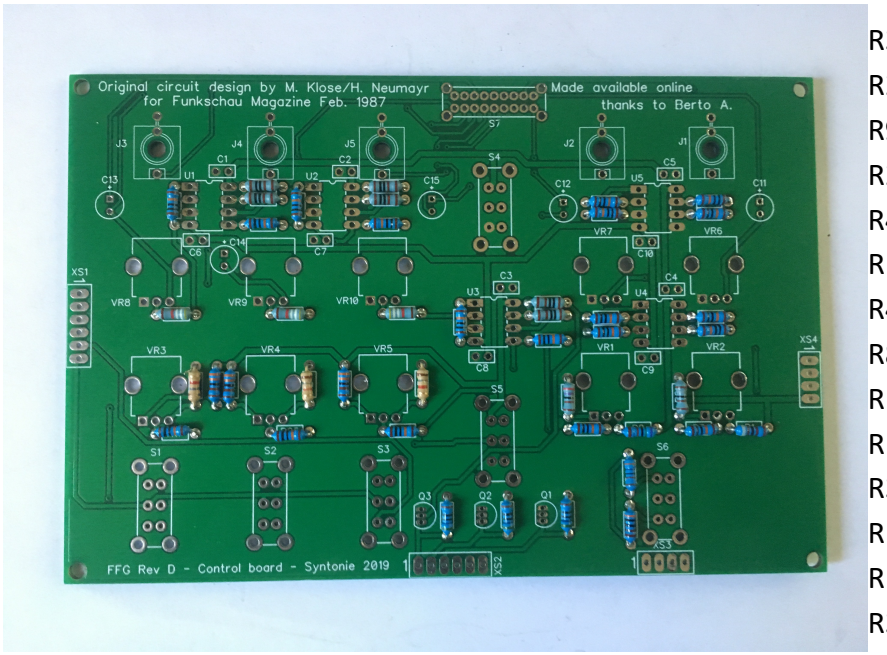
Mainboard is now done, let's get to the control board.

B. Control Board

I. Resistors



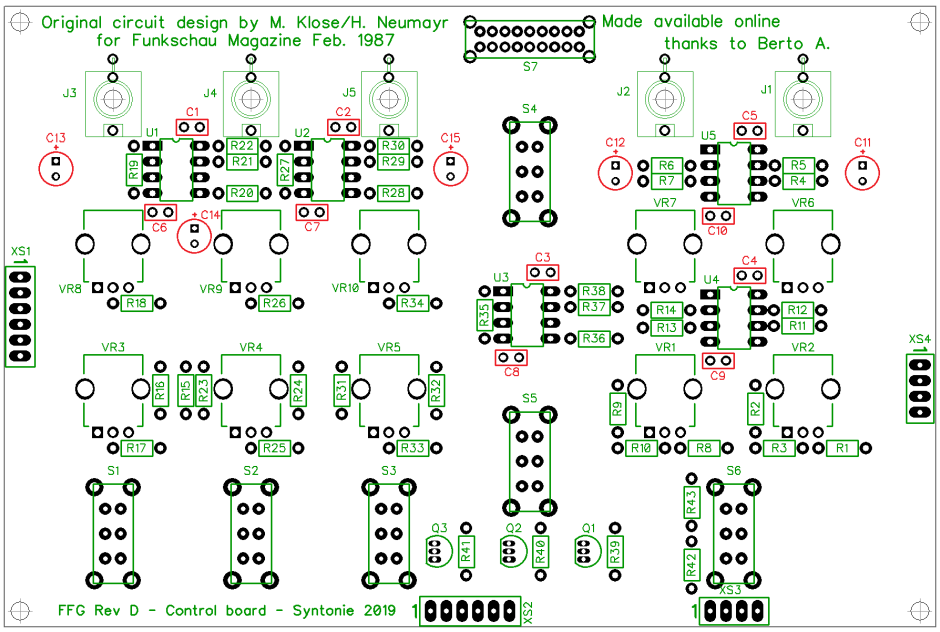
Place all the resistors:



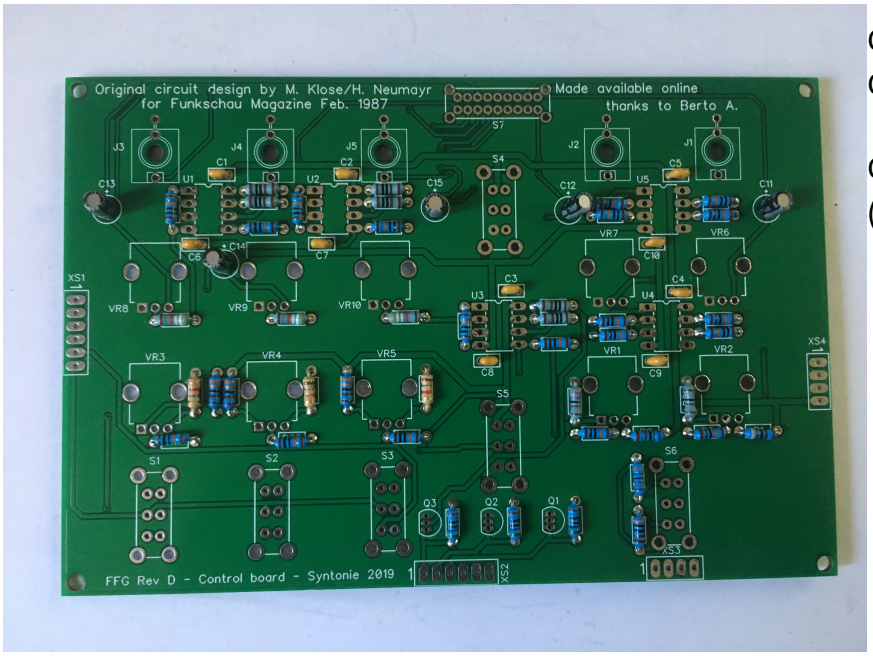
R39, R40, R41: 560R
R21, R22, R29, R30, R37, R38: 1K
R9, R16, R24, R32: 1K8
R2: 2K7
R42 : 3K
R15, R23, R31 : 3K3
R43 : 3K6
R8 : 9K1
R1 : 7K5
R18, R26, R34 :49K9
R3, R4, R5, R6, R7, R10, R11, R12,
R13, R14,
R17, R19, R20, R25, R27, R28, R33,
R35, R36 : 100K

(note : I only had 5%/beige resistors for R16, R24, R32 at the time I took the picture, kit includes 1%/blue resistors)
(note 2 : R1 and R6, are swapped on the picture, follow the component liste above or BOM. Same for R2 and R9)

II. Capacitors



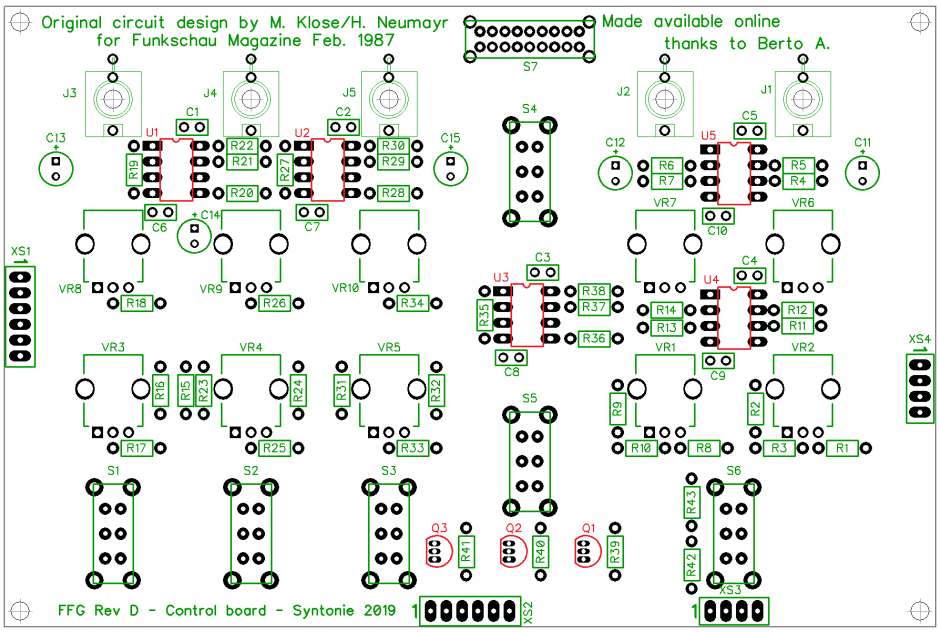
Place all the capacitors. The kit include small enough capacitors so they don't go higher than the top part of the jacks, however if you're sourcing your components yourself, you can use normal sized capacitors and bend them as we did for the power supply part of the mainboard, else they might touch the front panel.



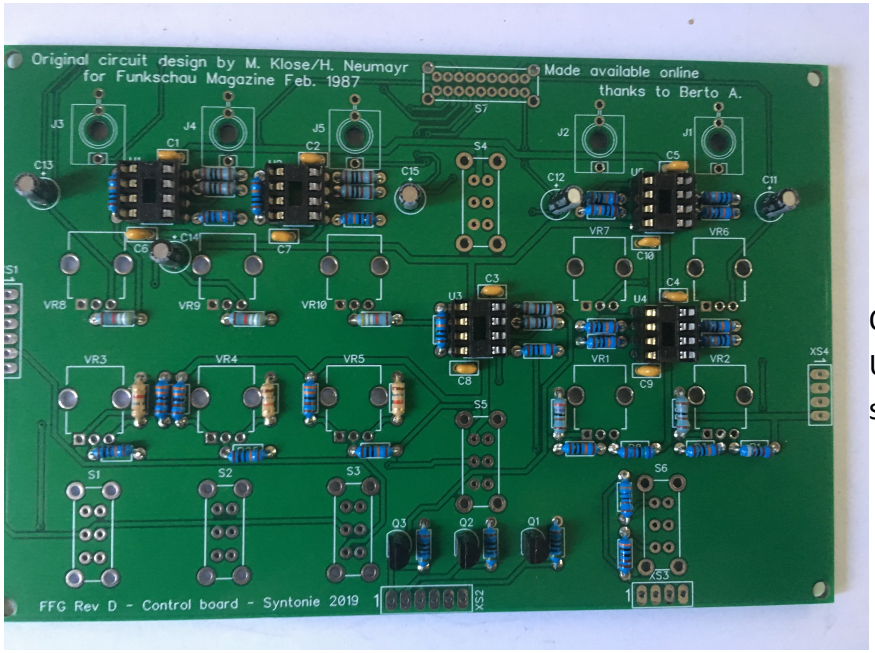
C1, C2, C3, C4, C5, C6, C7, C8, C9, C10: 100nF (Ceramic, marked 104)

C11, C12, C13, C14, C15: 10uF (Electrolytic, 35V)

III. Transistors



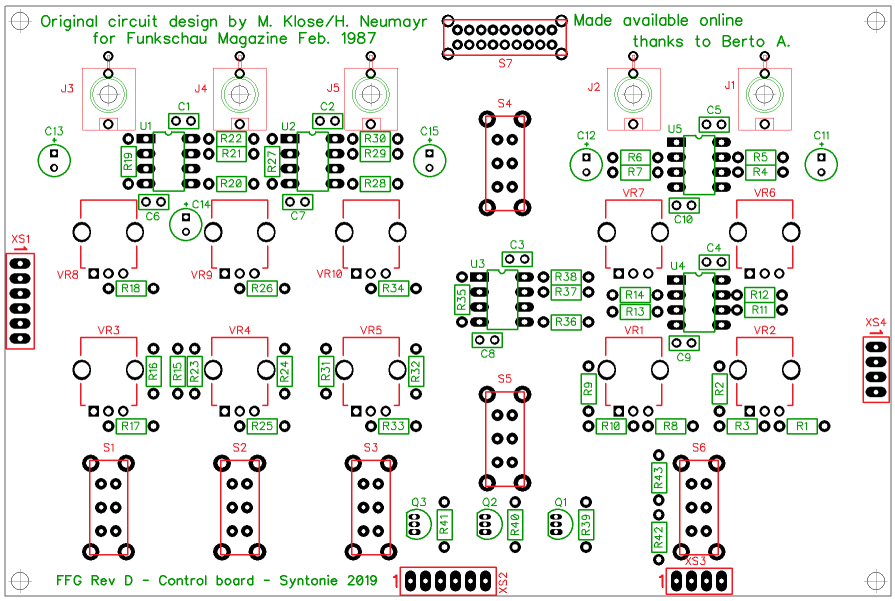
Place the 3 transistors, you can also place IC sockets, don't place the ICs yet :



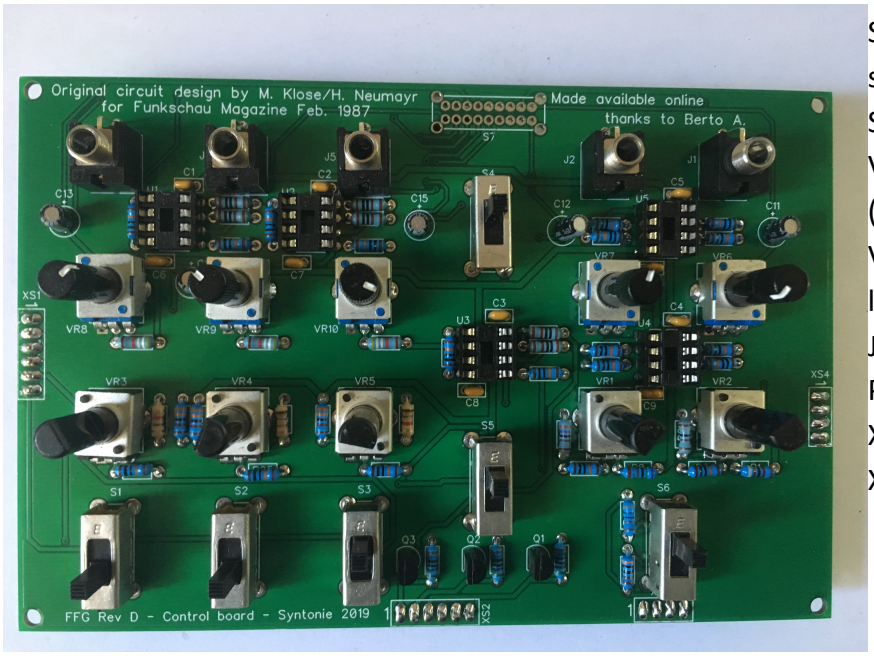
Q1, Q2, Q3 : 2N7000

U1, U2, U3, U4, U5 : 8 pin DIP socket

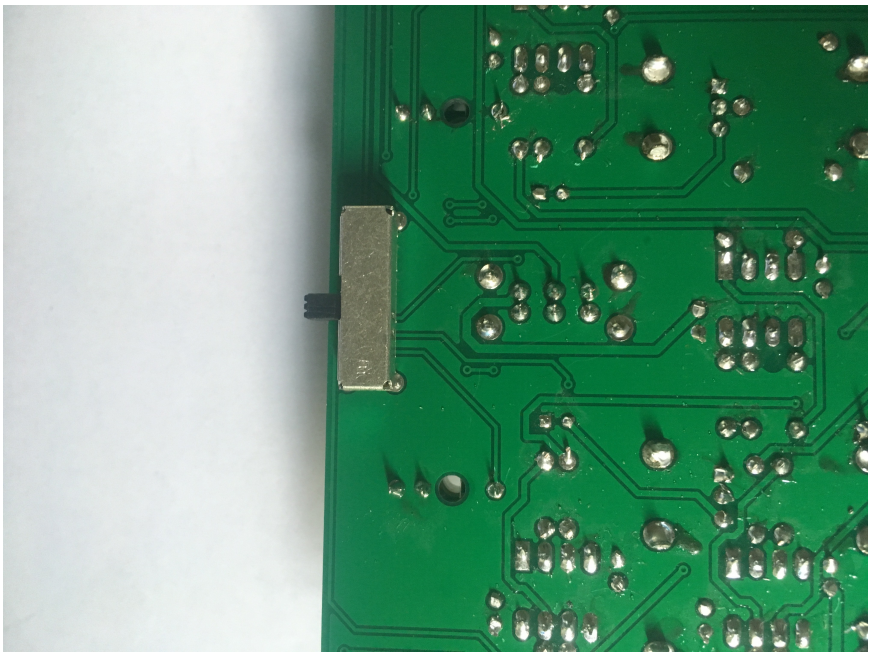
IV. Hardware



Place and solder all the switches, then all the potentiometers (check for value under the body of the pot), then the 5 jacks connectors and finally the board connectors and S7 (on the solder side of the board):

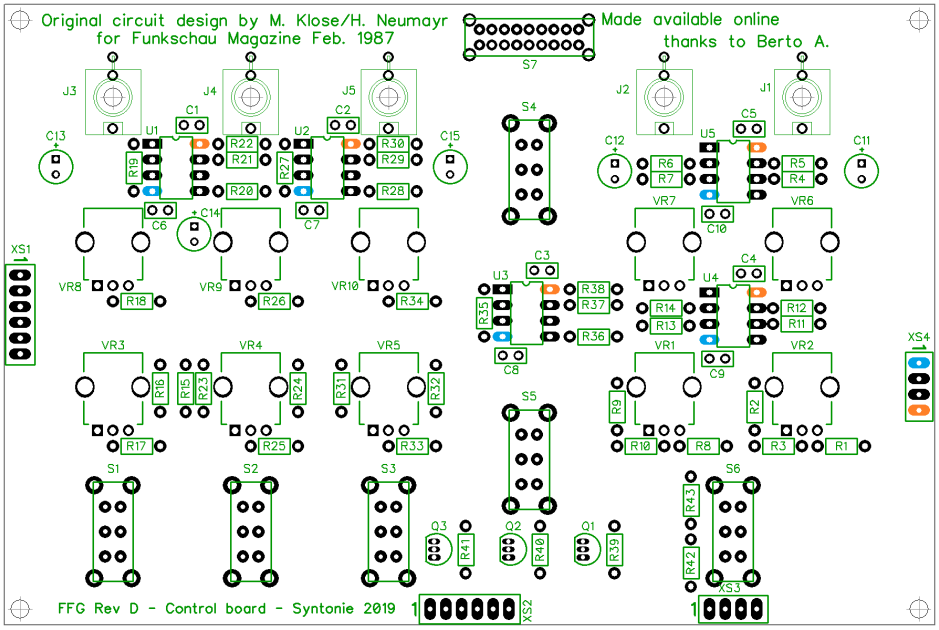


S1, S2, S3, S4, S5, S6: DPDT Slide switch EG2201
S7 : 6PDT Slide switch EG6201
VR1, VR2, VR3, VR4, VR5: 1K linear (marked B102)
VR6, VR7, VR8, VR9, VR10: 100K linear (marked B104, blue body)
J1, J2, J3, J4, J5 : 3.5mm mono jack PJ398SM
XS1, XS2 : 6 pin male connector
XS3, XS4 : 4 pin male connector

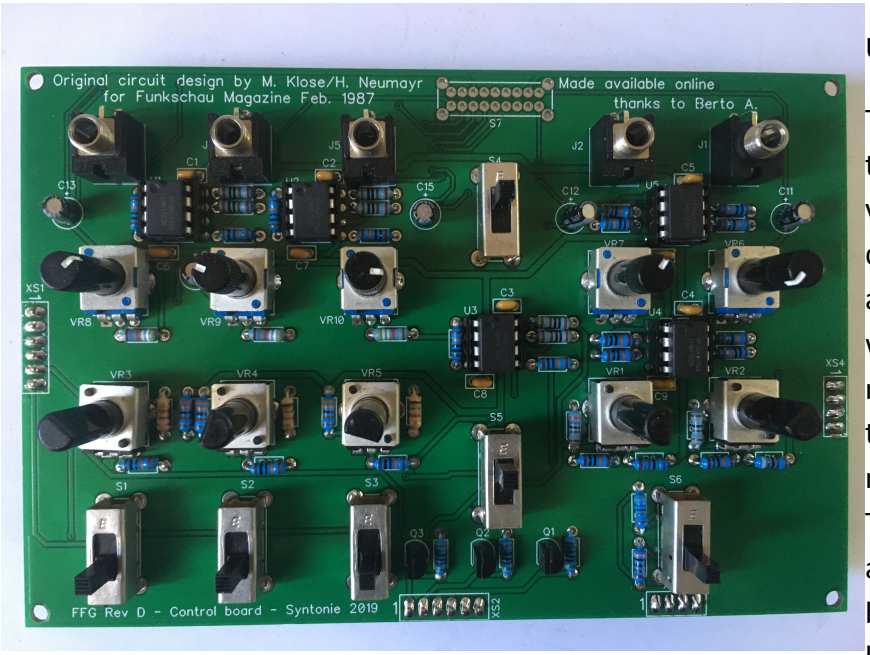


Don't solder S7 on the same side of the board as other controls, solder it on the other side of the board like this.

V. Integrated circuits



Before placing the ICs, we will test the power supply voltage again. Connect the boards together, the regulated DC from the mainboard comes from XS4 connector, and it is then distributed to all the ICs. The 2 middle pins of XS4 are ground, you can put the black probe of your multimeter on one of those pads, and the other probe on each of the point shown. Once again, orange is +12V, and blue is -12V. If what you read is correct, you can place all the ICs. Else check for component placement/orientation, and solder shorts/cold solder joints.



U1, U2, U3, U4, U5 : LM6172

The op-amp used are LM6172, so the circuit can be modulated using video rates control voltages. If you don't have any video rates modules and plan to use your FFG strictly with audio rates modules, you can replace all of them by TL072 as those are also cheaper. (Don't replace U1 on the mainboard by a TL072 though, as it is used to amplify/filter the video signal, bandwidth of the TL072 is too limited for video to pass through). That's why IC sockets are also great, you can put TL072 first and then replace them by LM6172 later.

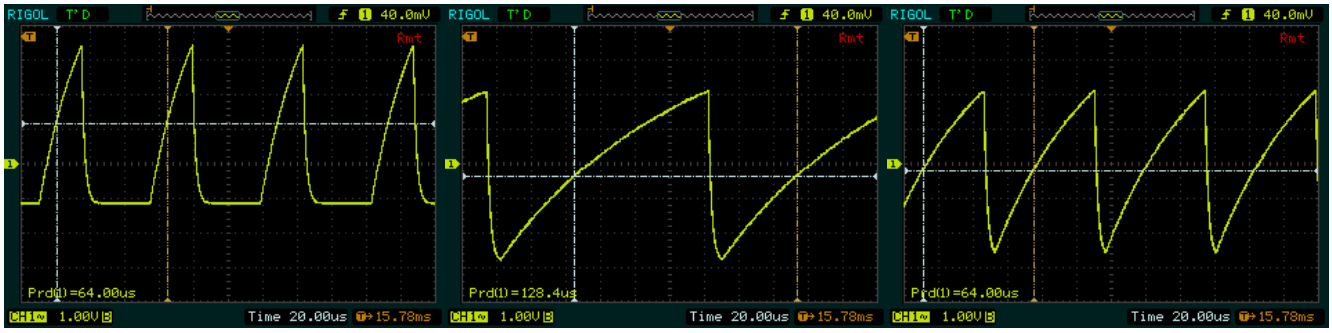
C. Calibration

Now that all the components are soldered, and IC placed in their sockets, it's time to test the whole circuit. Connect both boards together, connect power supply and switch the circuit on. Plug a video source into J2, and a monitor into J3. You should have something displayed now. And it's probably black and white.

First thing to check is S1 on the mainboard, the dip switch. Check that it matches the standard you're using (PAL or NTSC), the switches needs to be both up or down depending of the standard chosen.

Now you might have colors but maybe not, then you need to adjust P1, the trimmer potentiometer on the mainboard. It will adjust the ramp generator inside the MC1377.

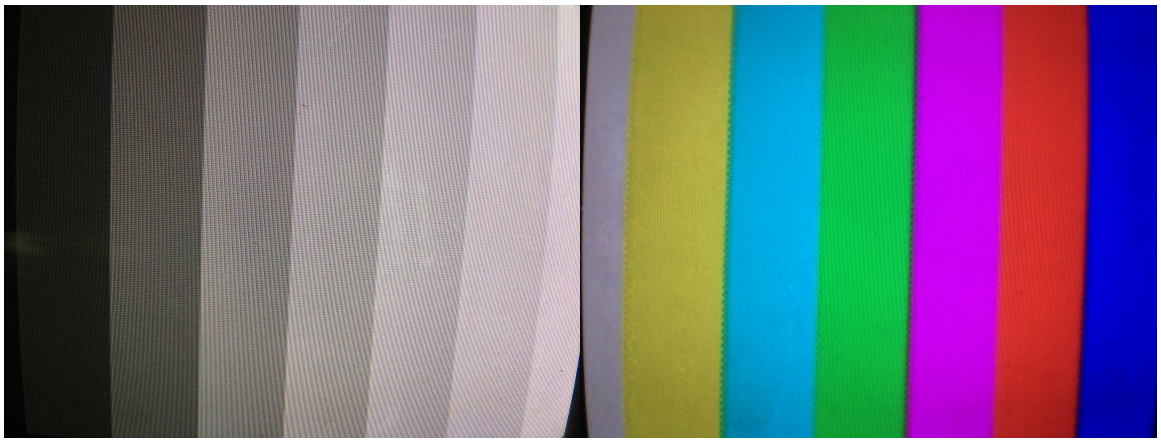
You can precisely calibrate it using a scope : probe at pin 1 of U9 (MC1377P), you should now see a rising ramp. Now turn P1 until the ramp signal is as shown on the oscilloscope capture, no flat part at the bottom, and the period of the signal is 64us (cause I'm in PAL for the tests, 63.5us in NTSC).



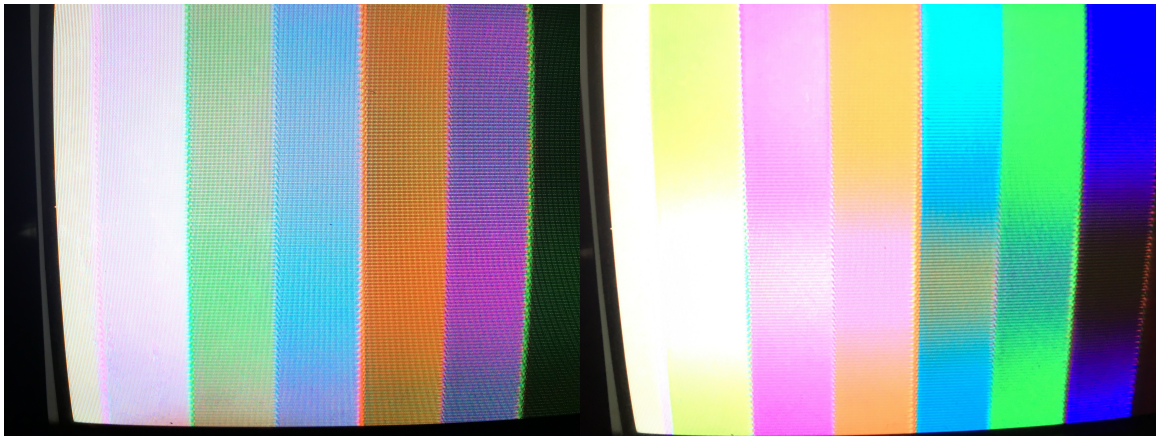
Good period, bad ramp shape Good shape, bad period (128us) This is what you're aiming for

If you don't have an oscilloscope, no worries. Best procedure is to turn P1 completly to one side or the other (it will click once you reached the end) and then turn about 12-13 turns (the potentiometer has 25 turns in total), it should set it around it's middle position. Now, while checking on a monitor, you should have some colors displayed. If not, try to turn C15 (it only has one infinite turn), if you only have a tiny spot where color are displayed when turning C15, adjust P1 again until the colors are back. Then if P1 is set correctly, C15 should only change the color saturation a bit and have a small deadzone where it is black and white.

If the colors are pale or off (it should be Red, Green, Blue, Magenta, Yellow, Cyan, Black and White), you'll need to adjust P1 again.
Here is some shots using a greyscale bar as a source, when all the invert/bitswap switches are down, it gives a colour bar/test pattern at the output.



Video input : Greyscale bars Video output : 8 colors colorbar/test pattern



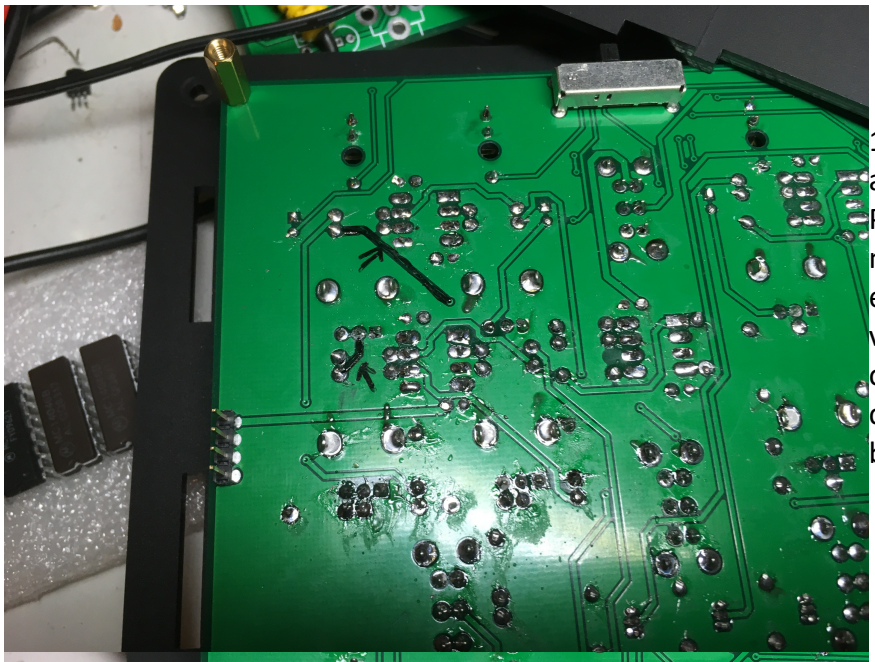
This is what I mean when I say pale/off colors, in this case P1 needs to be adjusted.

If you're using a video mixer/profesionnal CRT monitor/capture card, best is to do your calibration with the mixer/pro monitor/capture card after the FFG. Those are generally more picky with the signal you send them, so you might need to readjust P1 if you did it on an old consumer CRT for exemple (these complains less with out of spec signal).

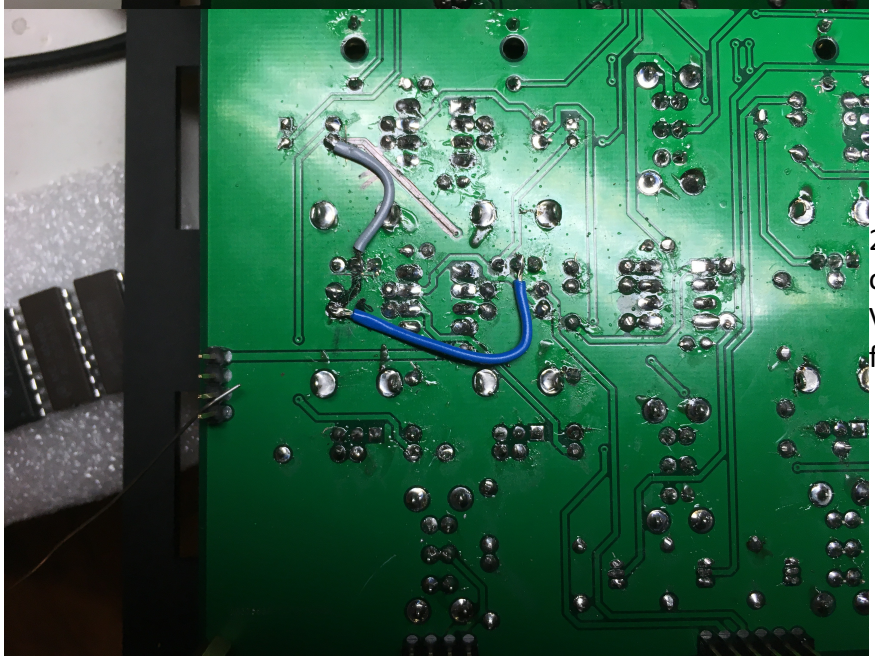
Now you can test the knobs/switches, and also CV inputs, to see if everything work as it should. Check the user guide to see what does what.

Issue #1 :

Low and high threshold level CV inputs are swapped, one controlling the other and vice versa.
If you want to fix it, you need to :



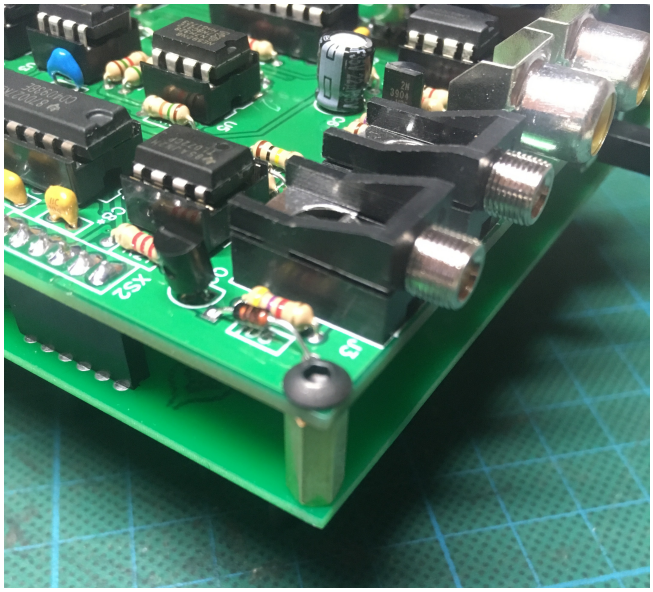
1) Cut the two traces in black using an exacto knife or something sharp. Put your multimeter in continuity mode to check if the traces are cut effectively. The bigger trace got a via (little hole that goes on the other side of the board), so it continue on the other side of the board, going to VR7 middle pin.



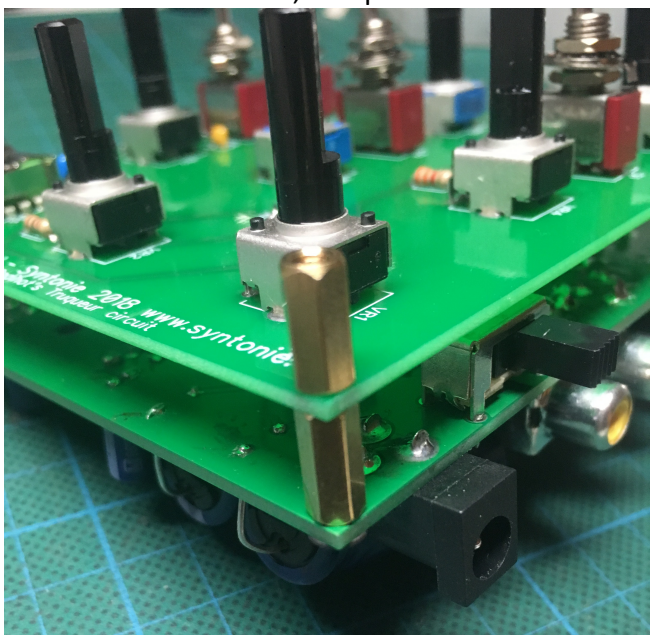
2) Replace the 2 traces with pieces of wire. Grey wire goes from R4 to VR6 middle pin. Blue wire goes from R11 to VR7 middle pin.

D. Enclosure assembly (copied from TRQ for now, same procedure)

Place the 11mm FF spacer between the two board, secure them with 6mm screws on the mainboard, components side.

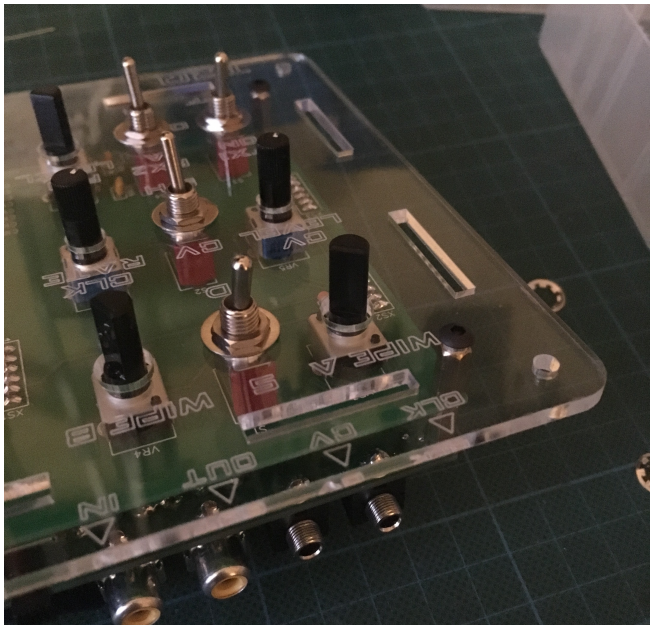


Place the 10mm MF spacers on the controlboard, components side.

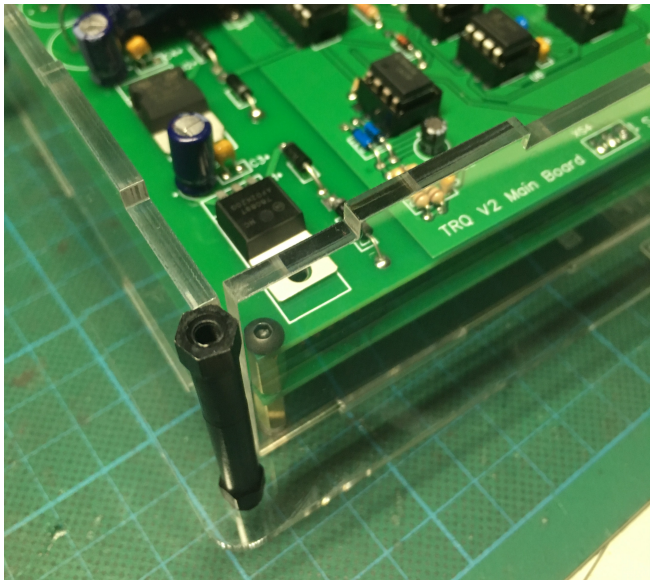


Now, remove the nuts from the 5 3.5mm jack, and place the faceplate on the circuit boards, secure them using 6mm screws. You can put the nuts on the jack back.

Now, place the 45mm spacers in each corner of the frontplate using 6mm screws.



Add the lateral plates.



Add the bottom plate, and use the 8mm screws + rubber feet to secure it to the rest of the assembly.


FX1
ON
FX2
A
FX1
LEVEL
TRQ