

# TH-VCO1

## Build Instructions (detailed)

This is a detailed build instruction for Rev 1.0 of the TH-VCO1, based on the Eurorack Adaption by Tom Whitwell by Music Thing Modular and the original Circuit by Thomas Henry. Boards and Panel were designed by David Huss in the course of a synthesizer workshop at HFBK Hamburg.

The TH-VCO1 is a stable and wide range Oscillator with three different waveforms, two FM-inputs and CV-controllable Pulsewidth. The Panel is 10 HP wide (50.50 mm) and the VCO uses max 55mA on  $\pm 12V$ .

To build this module some basic tools are needed: soldering iron, solder, cutting pliers, flat pliers, multimeter, cutter

Make sure to be concentrated on details. Some parts can look very similar and should not be confused with each other! Searching for errors afterwards takes far more energy than doing it right from the start.

More details (BOM, Manual, Calibration Manual) and other projects can be found under <https://code.hfbk.net/t6/sdiy/wikis/home>

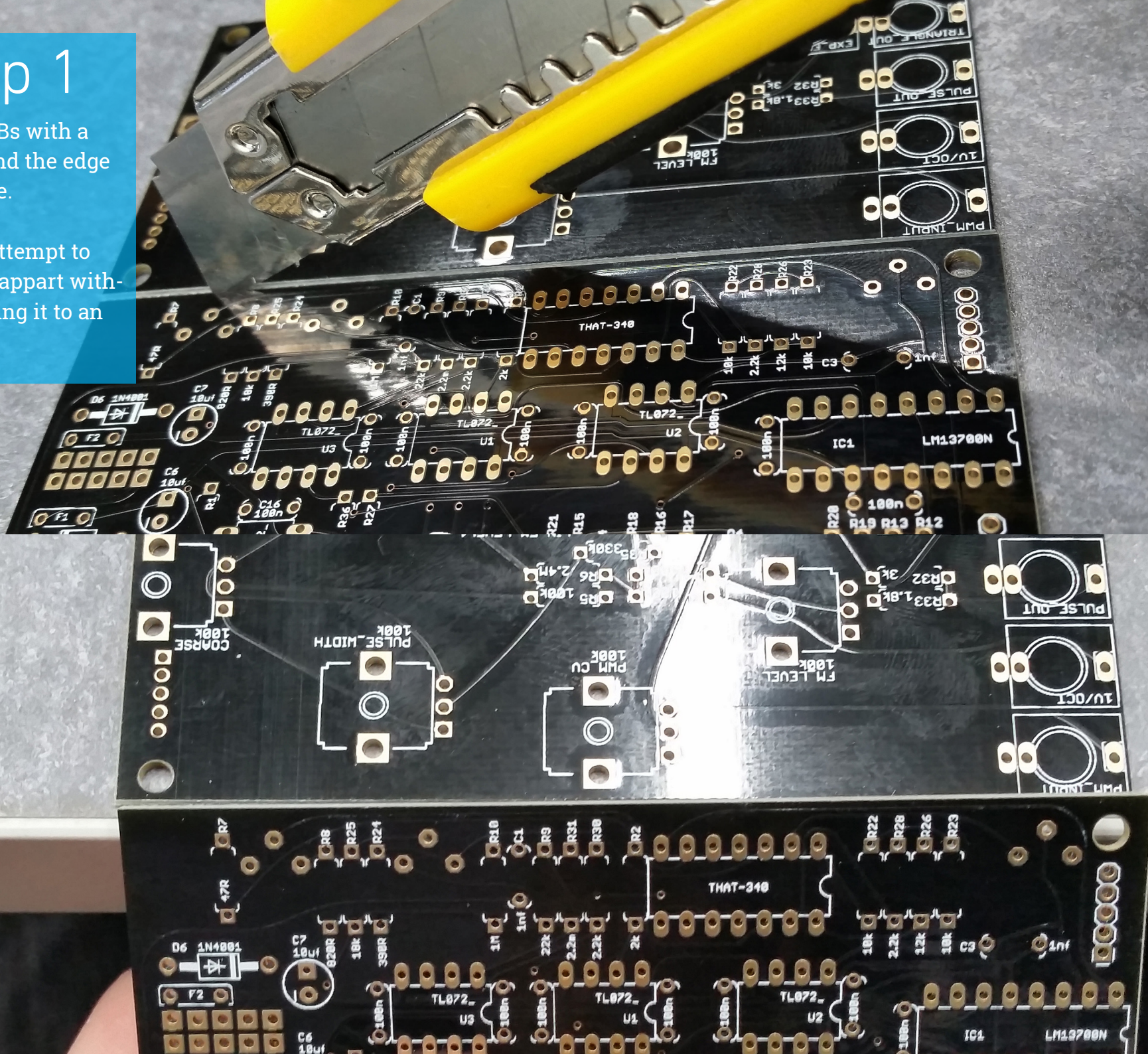




# Step 1

Split PCBs with a  
cutter and the edge  
of a table.

Do not attempt to  
break it apart with-  
out putting it to an  
edge!





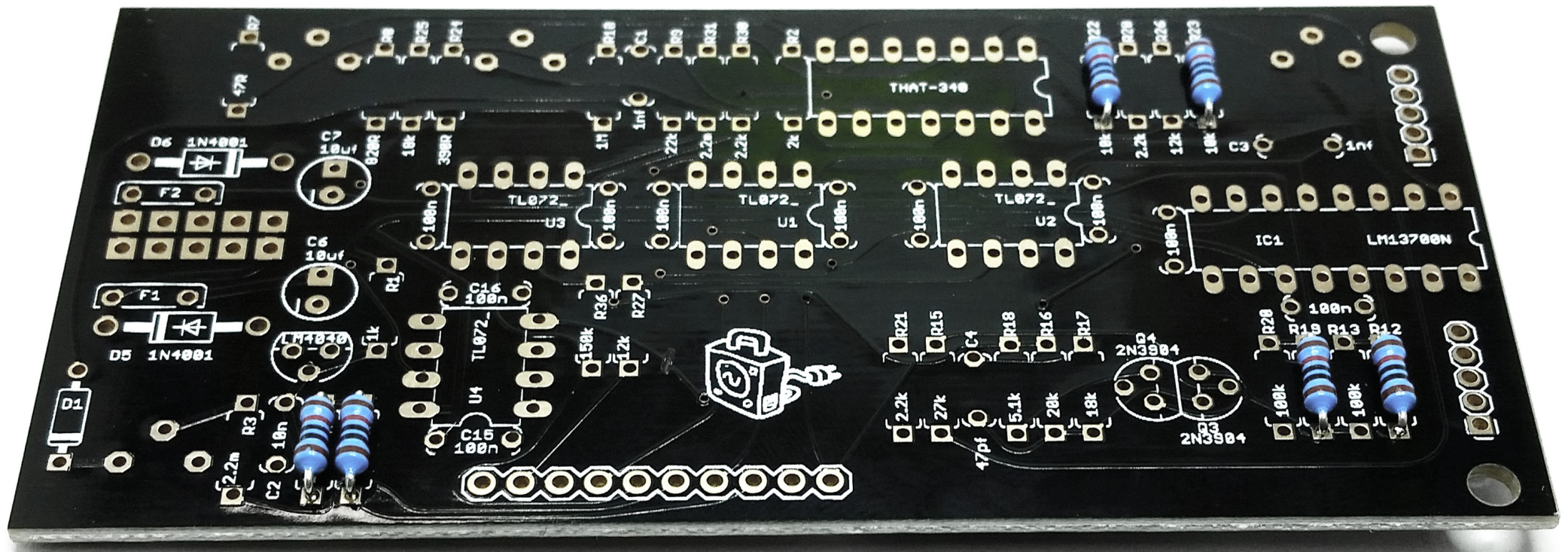
## Step 2

On components PCB:

solder all 10k resistors (6 pieces) in.

Careful not to confuse them with 18k!

6x Resistor  
10k





## Step 3

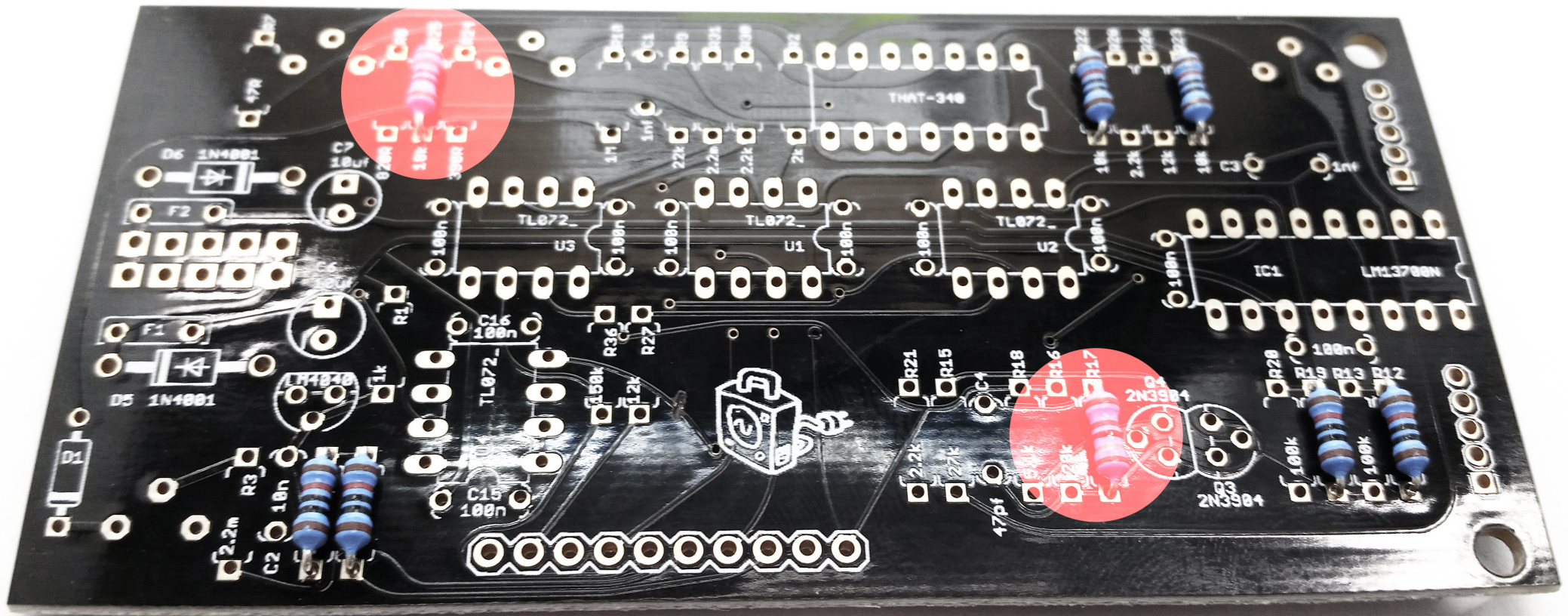
## On components PCB:

solder all 18k resis-  
tors (2 pieces) in.

Careful not to confuse them with 10k!

## 2x Resistor

18k





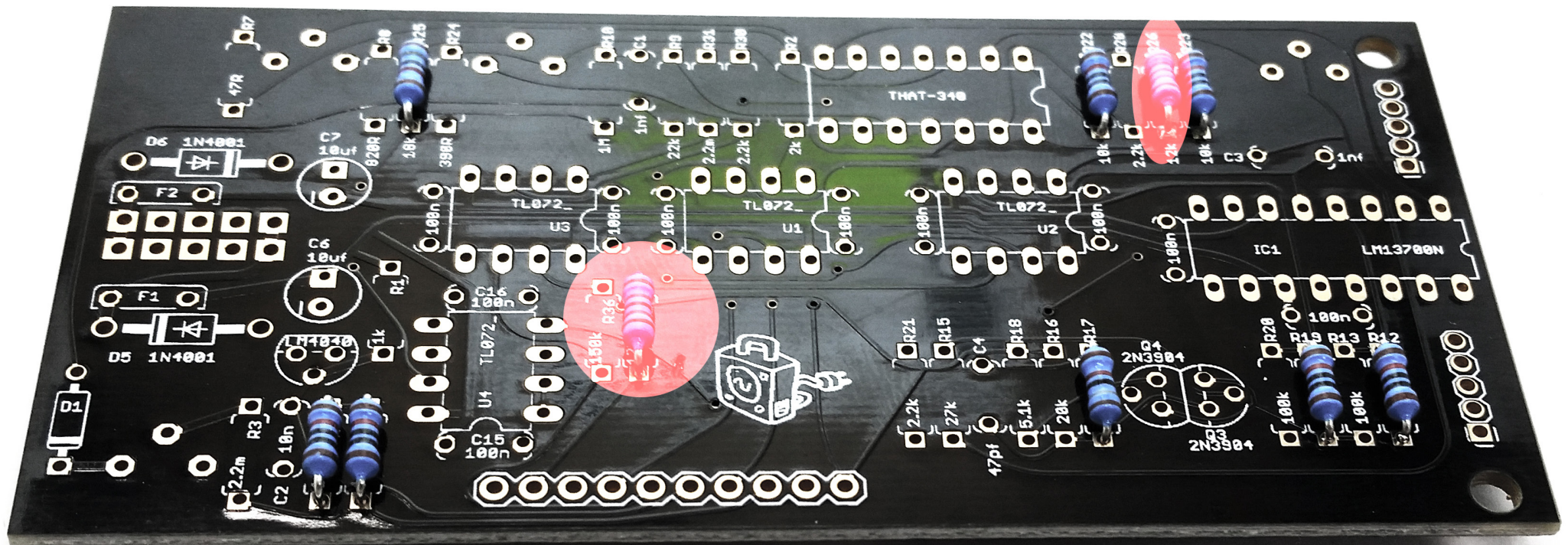
# Step 4

On components PCB:

solder all 12k resistors (2 pieces) in.

## 2x Resistor

### 12k





## Step 5

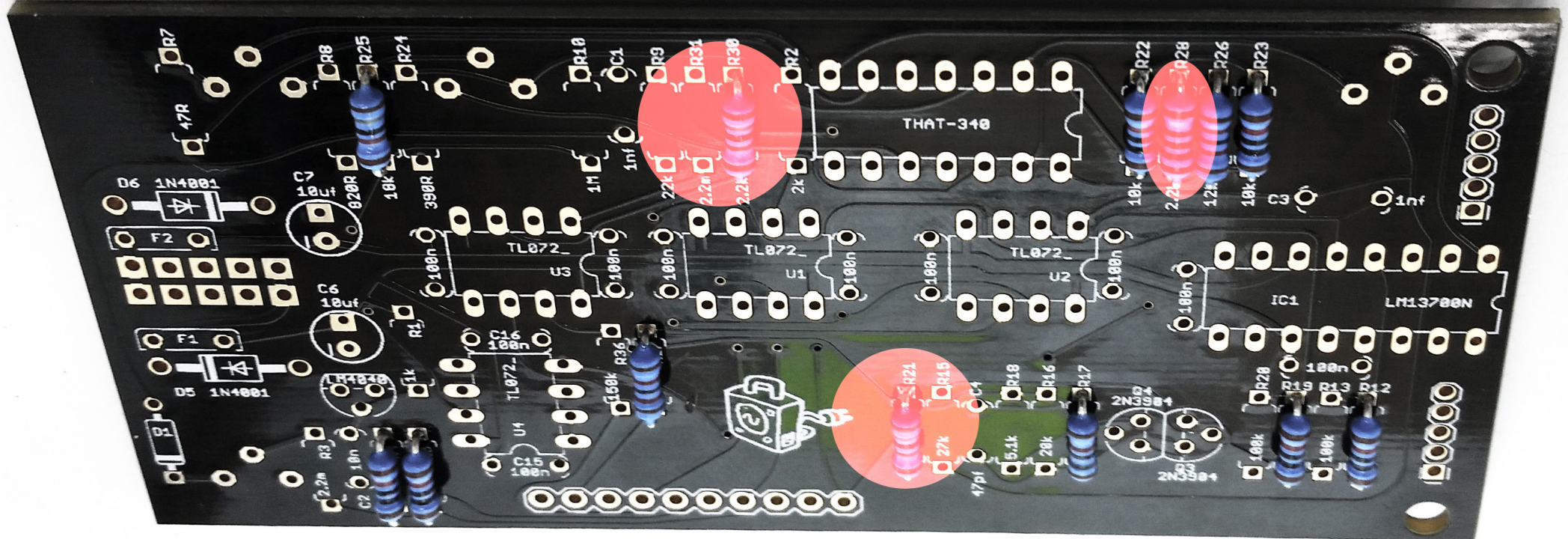
On components PCB:

solder all 2.2k resis-  
tors (3 pieces) in.

Do not confuse with  
2.2m or 22k!

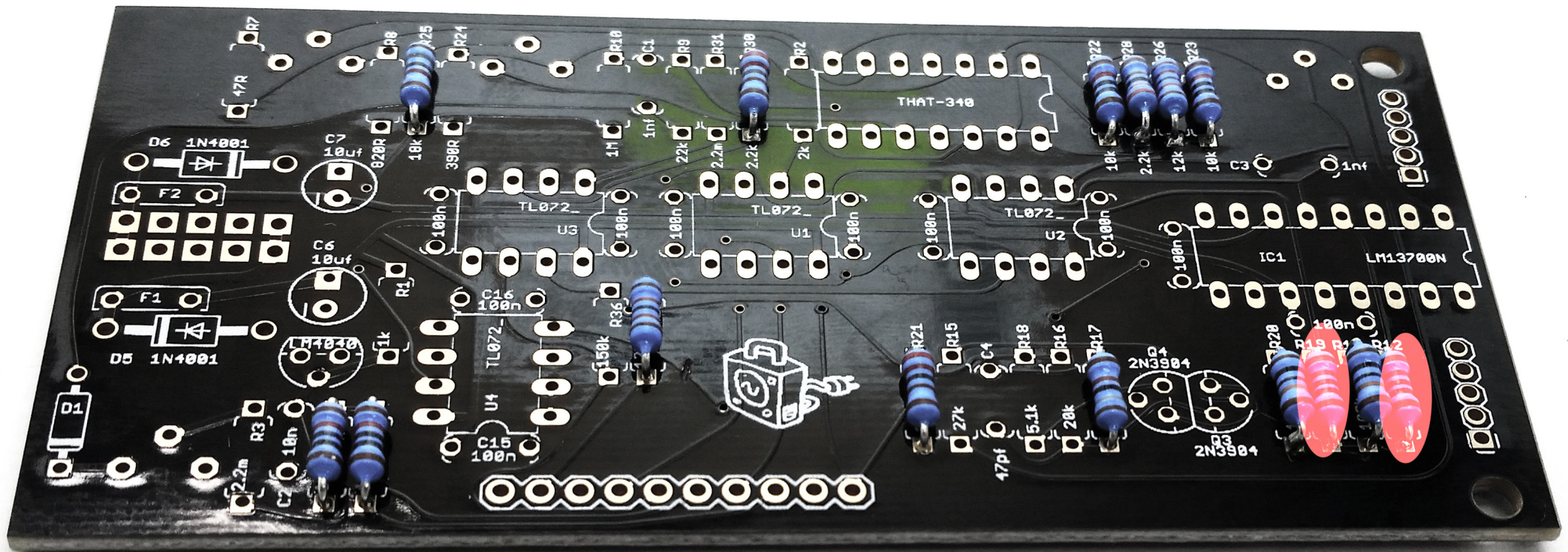
## 3x Resistor

2.2k





solder all 100k resis-  
tors (2 pieces) in.





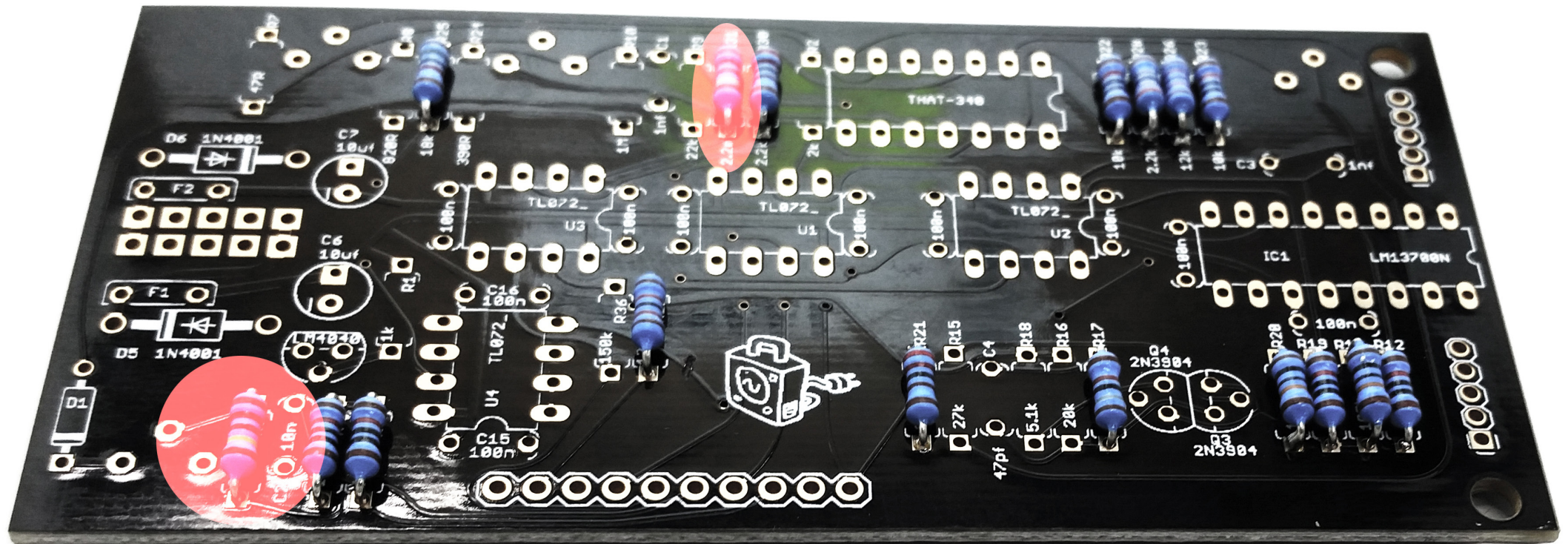
# Step 7

On components PCB:

solder all 2.2M resistors (2 pieces) in.

## 2x Resistor

### 2.2M





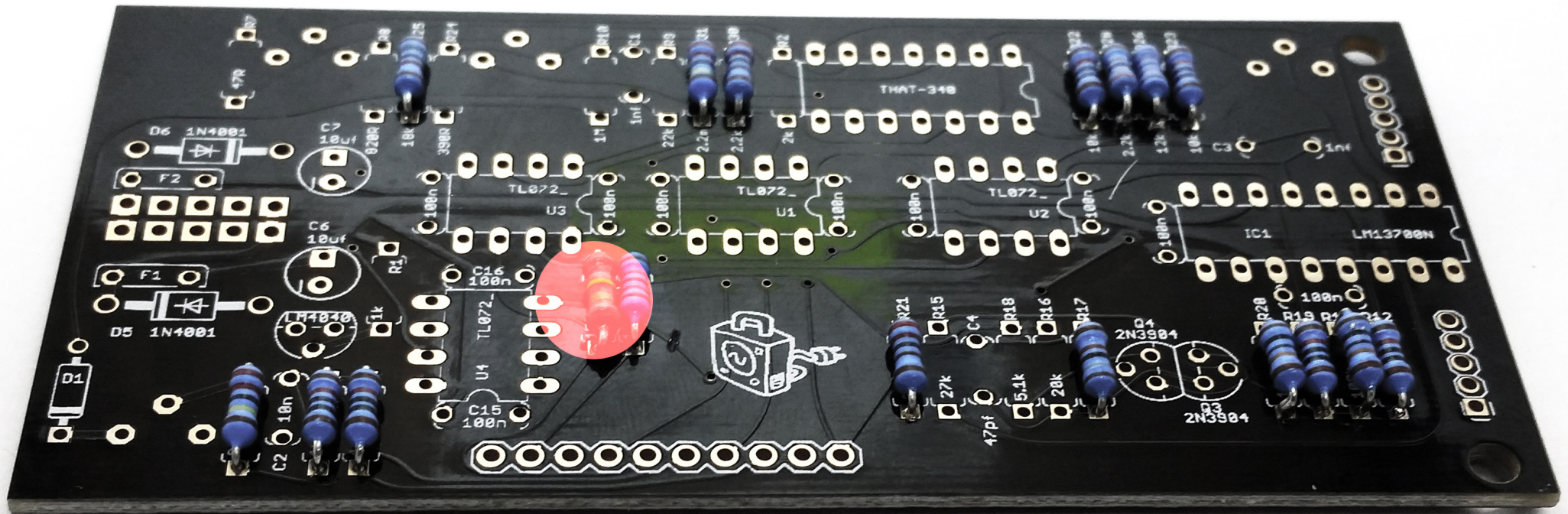
# Step 8

On components PCB:

solder R36 with 150k in (replace this with a different value to change PWM control scale, leave this out and add a 200k precision trimmer if you want to add the PWM Scale Trim Mod)

Note that in this instruction we used a different valued resistor

1x Resistor  
150k





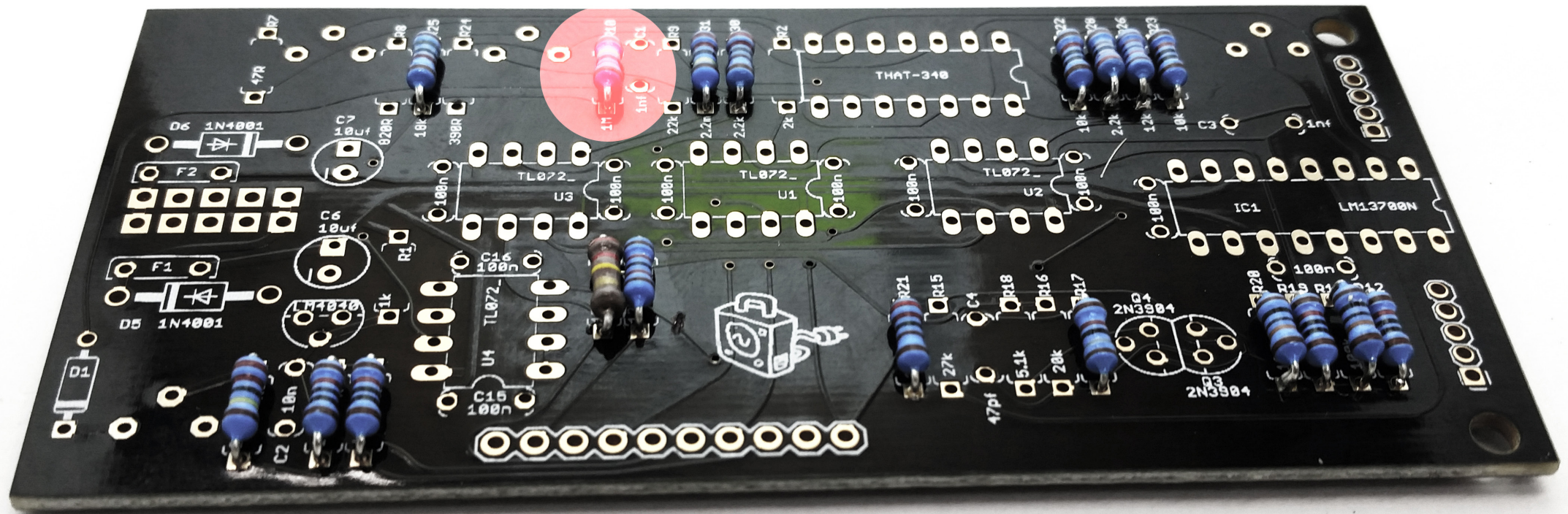
# Step 9

On components PCB:

solder R10 with 1M

1x Resistor

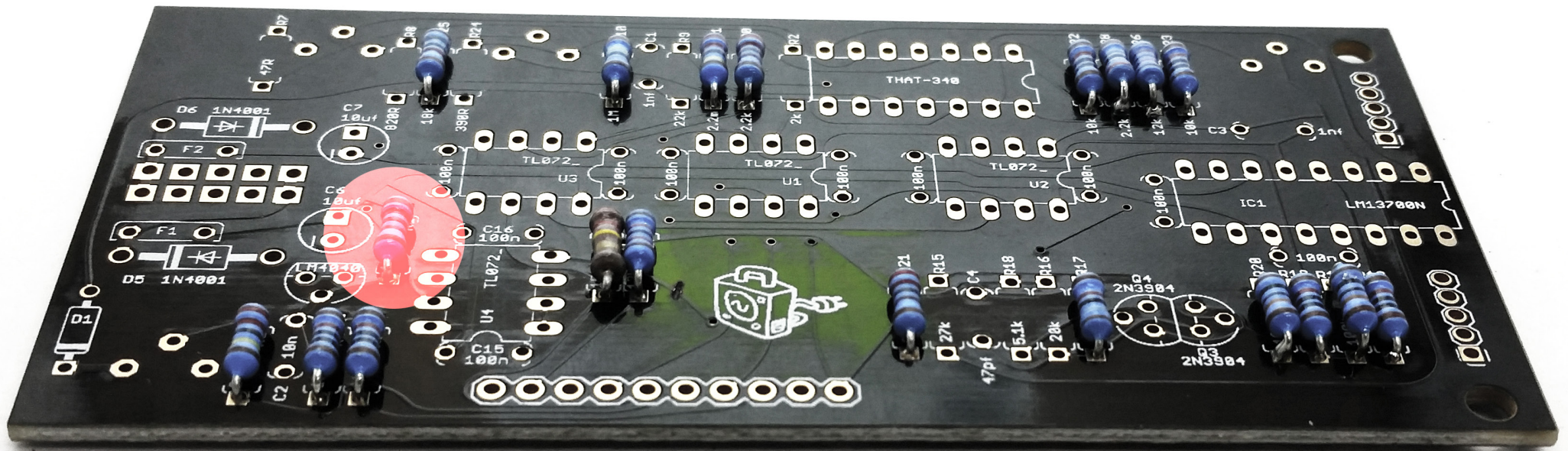
1M





solder R1 with 1k

1k





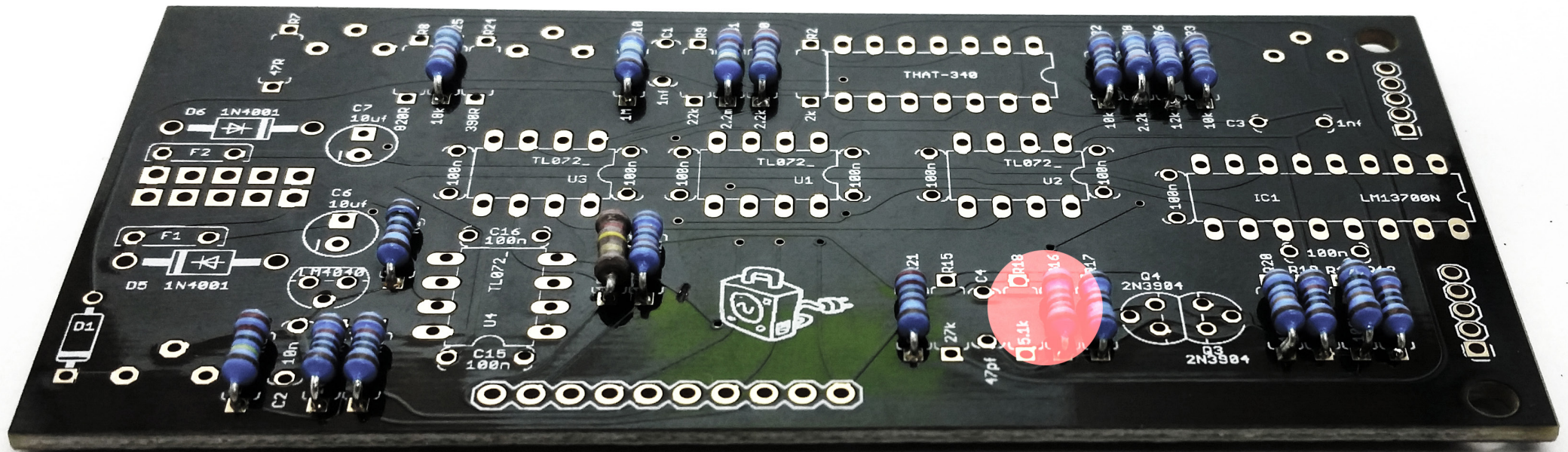
# Step 11

On components PCB:

solder R16 with 20k

1x Resistor

20k





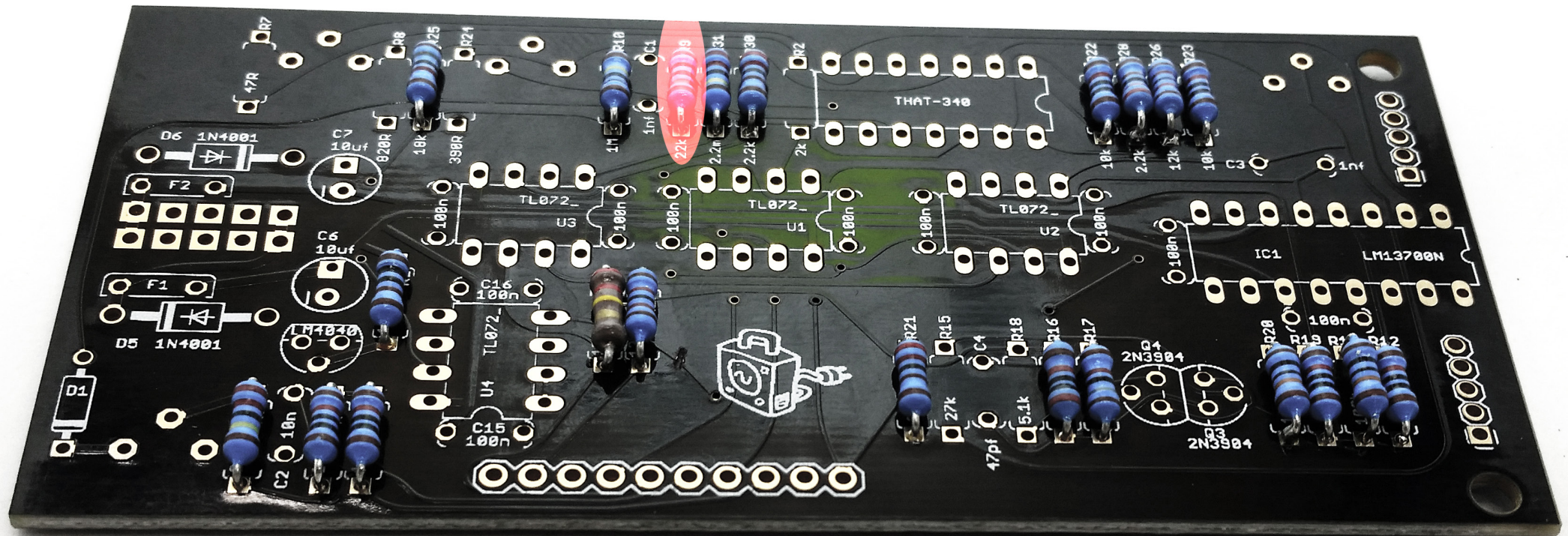
# Step 12

On components PCB:

solder R9 with 22k

1x Resistor

22k





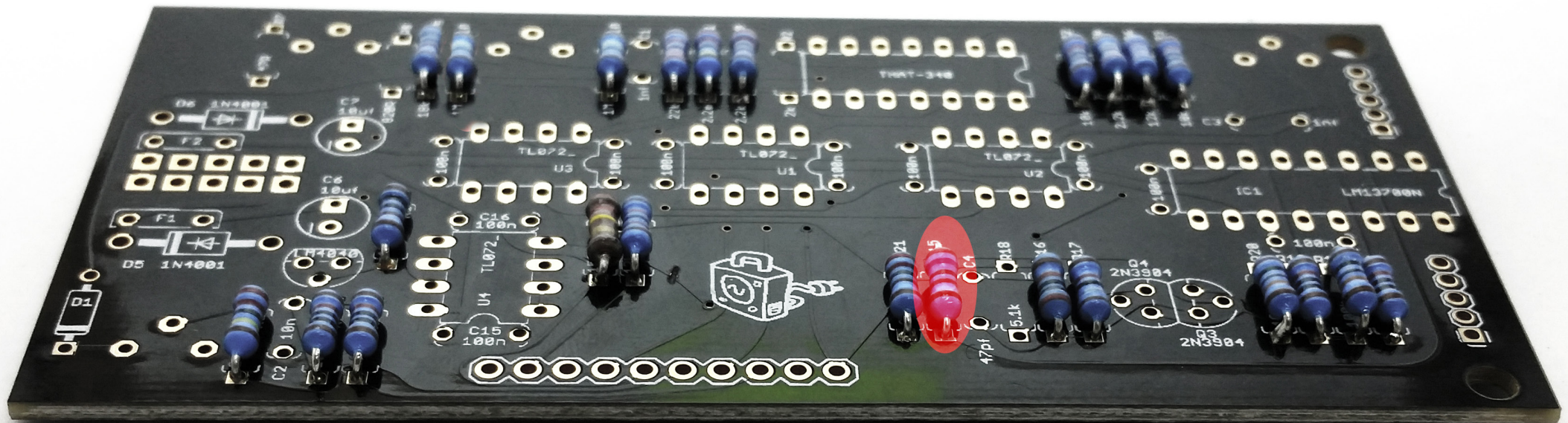
# Step 13

On components PCB:

solder R15 with 27k

1x Resistor

27k



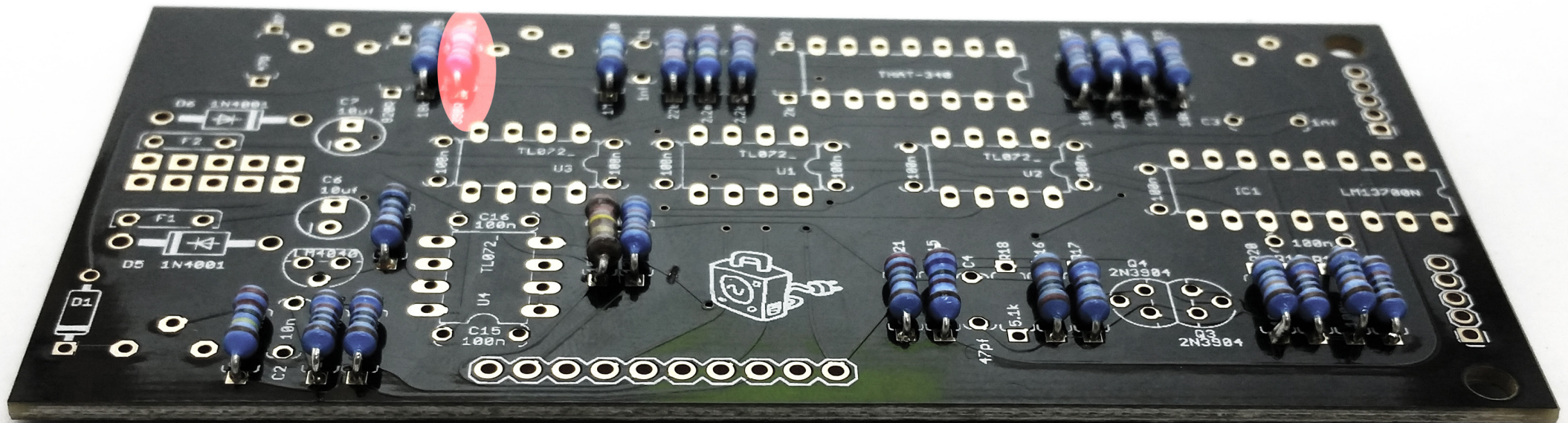


# Step 14

On components PCB:

solder R24 with 390R

1x Resistor  
390R



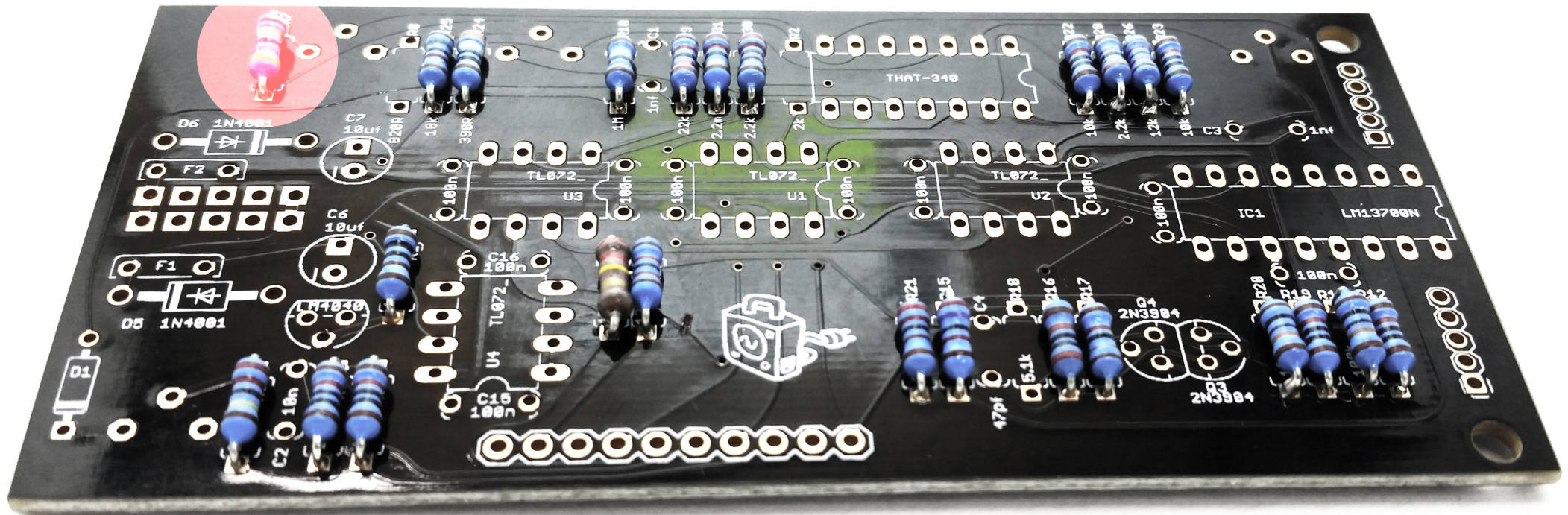


# Step 15

On components PCB:

solder R7 with 47R

1x Resistor  
47R





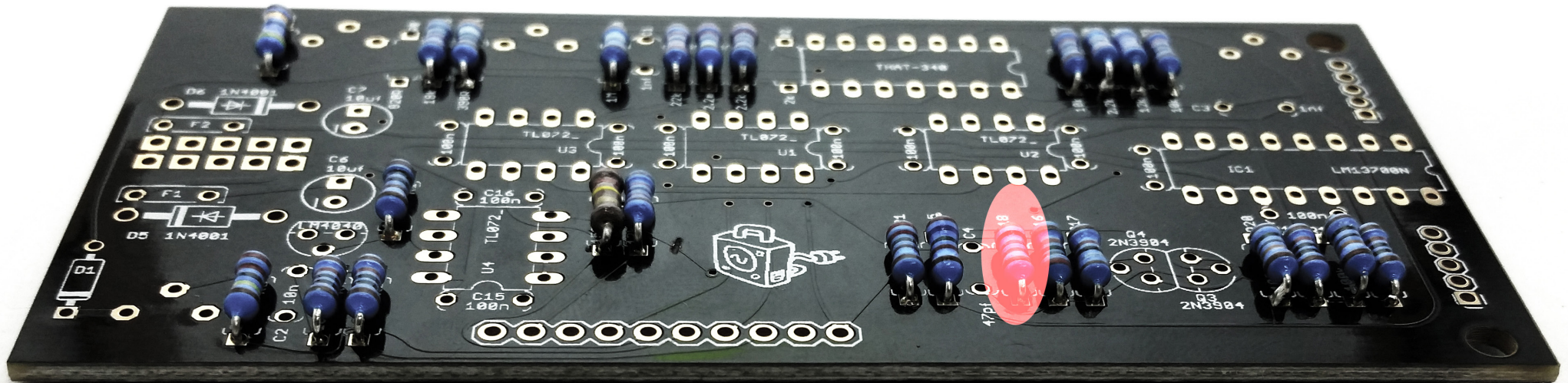
# Step 16

On components PCB:

solder R18 with 5.1k

1x Resistor

5.1k



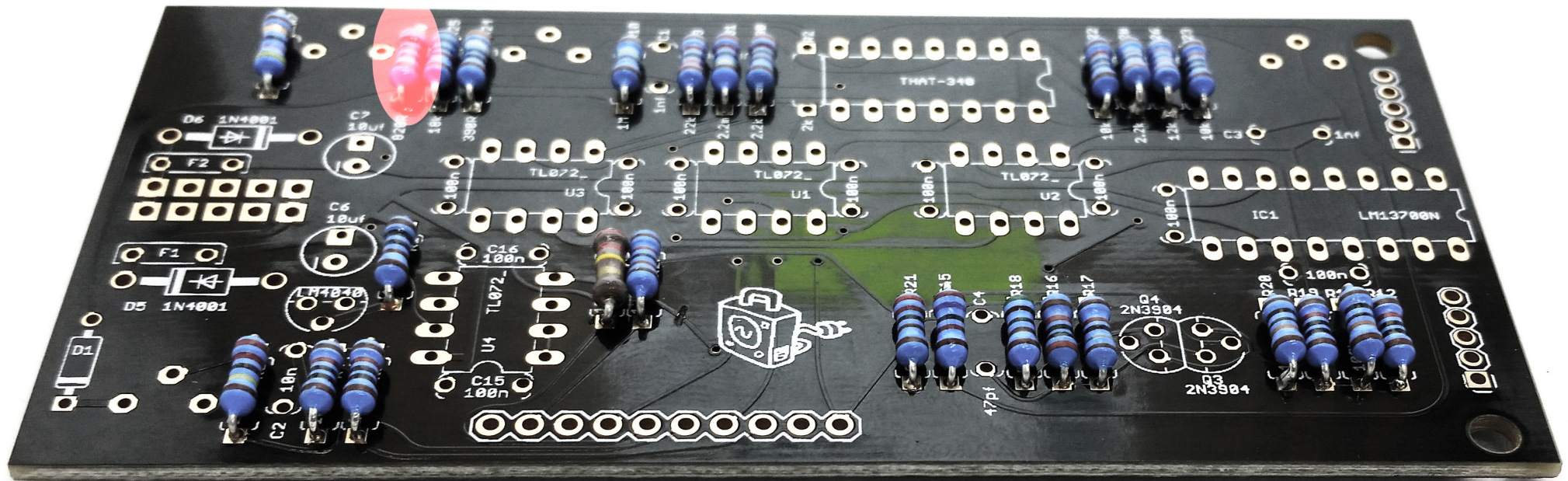


# Step 17

On components PCB:

solder R8 with 820R

1x Resistor  
820R





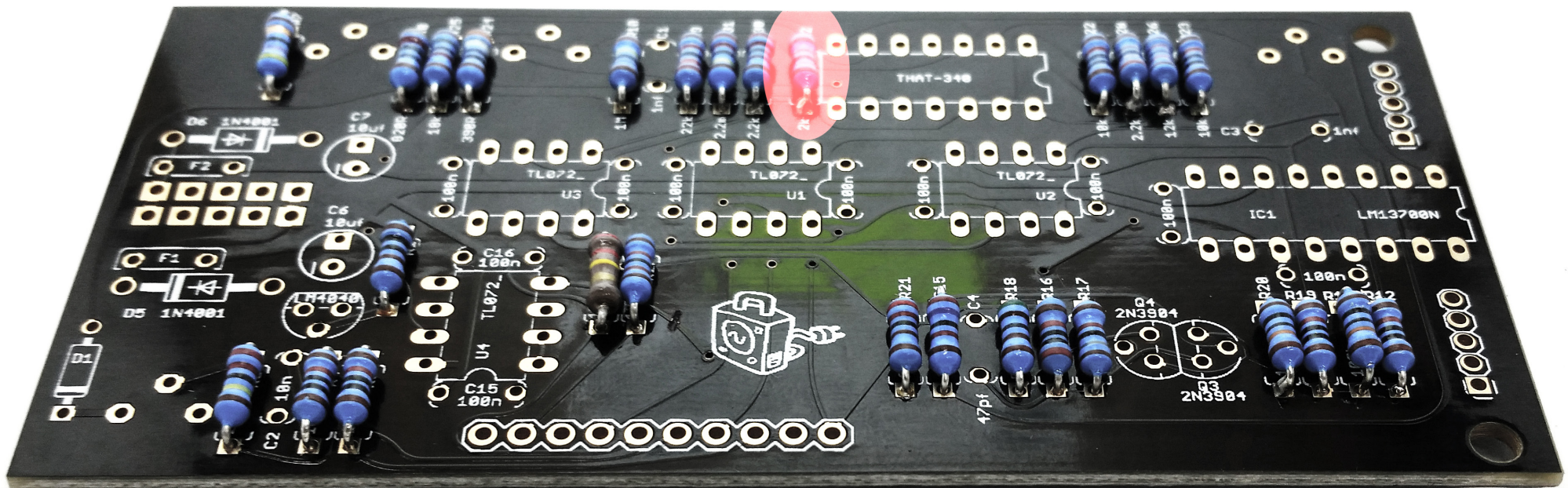
# Step 18

On components PCB:

solder R2 with 2k.

If you intend to use matched transistors instead of the THAT340 chip (see [Step 28B](#)), do not solder your 2K 3300ppm tempco in yet but after soldering the transistors. All three parts need to be in thermal contact which can be achieved by glueing them together with either epoxy or cyanacrylate.

1x Resistor  
2K





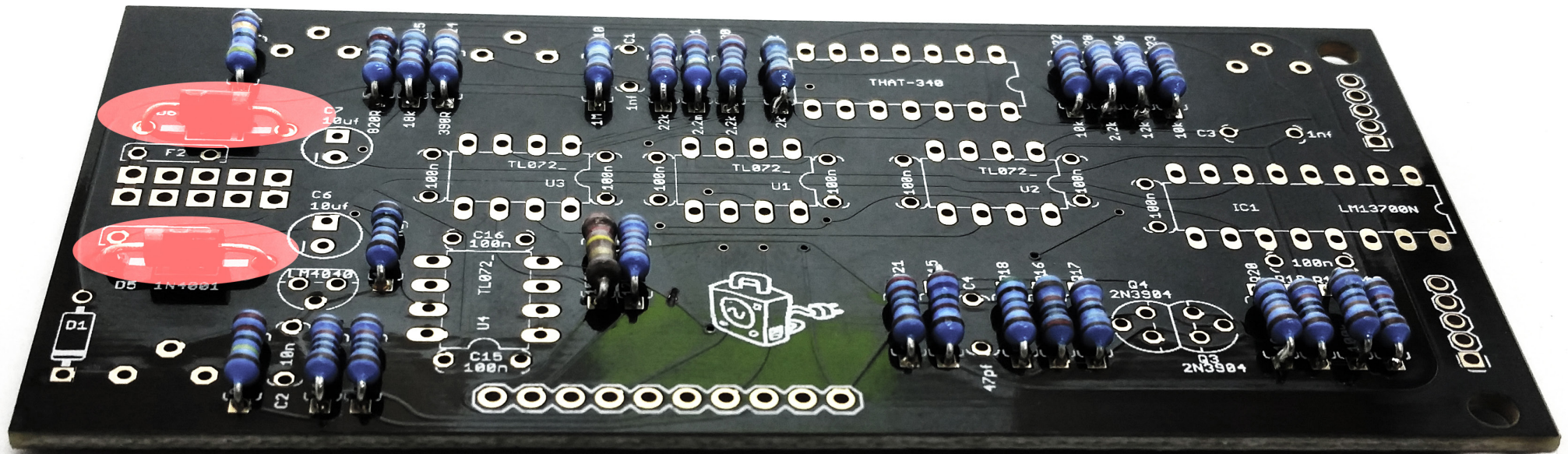
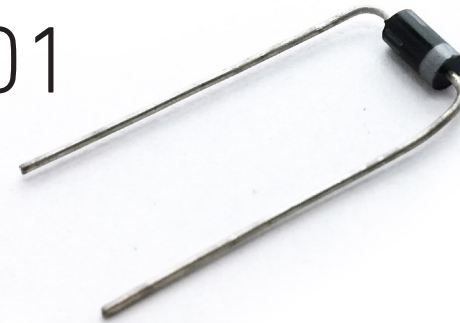
# Step 19

On components PCB:

solder the two 1N4001 diodes in.

Careful with the orientation! The white stripe needs to be oriented into the right direction as indicated on the silkscreen

2x Diodes  
1N4001





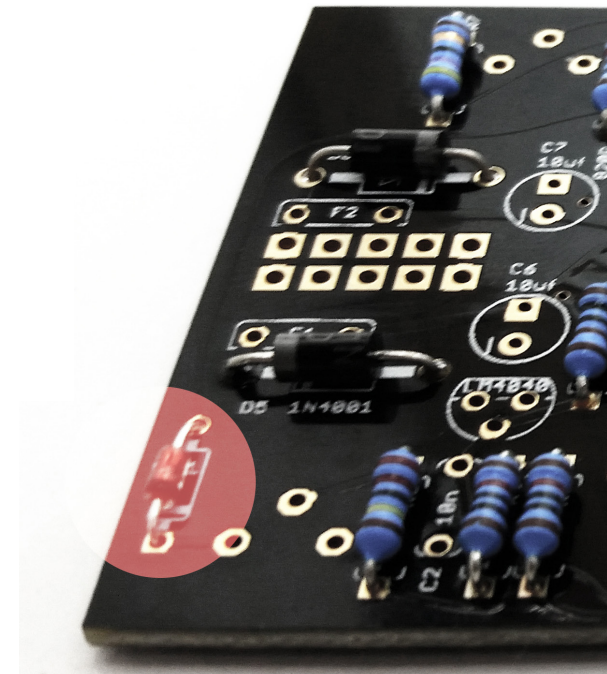
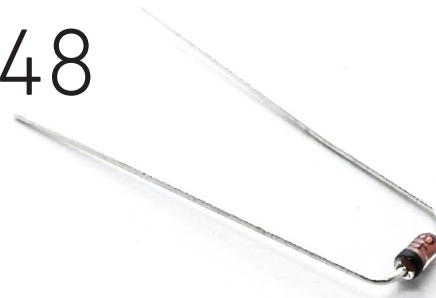
# Step 20

On components PCB:

solder D1, 1N4148

Careful with the orientation! The black stripe needs to be oriented into the right direction as indicated on the silkscreen

1x Diode  
1N4148





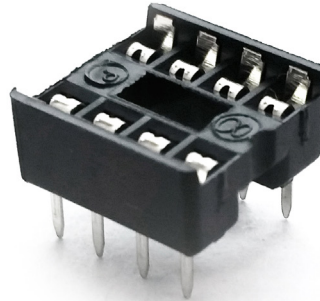
# Step 21-22

On components PCB:

solder the IC sockets. Solder one leg first, solder the rest once you made sure everything is straight. Orient them right to avoid mistakes when putting the ICs in later.

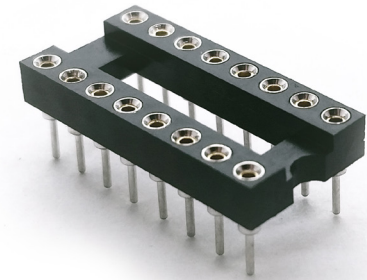
4x IC socket

8

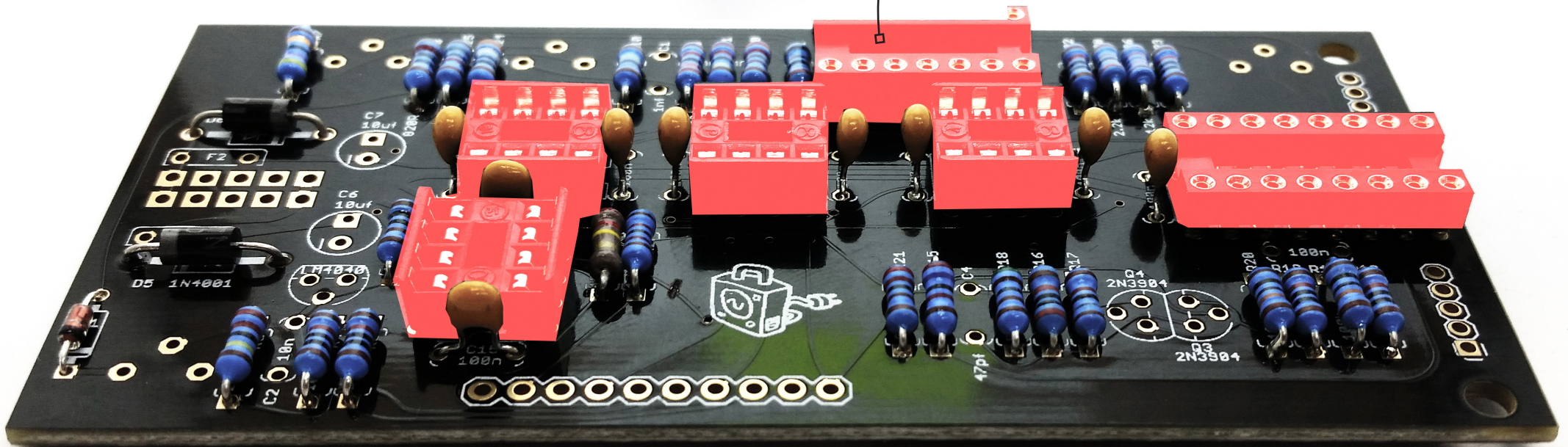


2x IC socket

16+14



Do NOT solder the 14 Pin IC socket if you don't have a THAT340 but plan to use separate matched Transistors! (see [Step 28B](#))



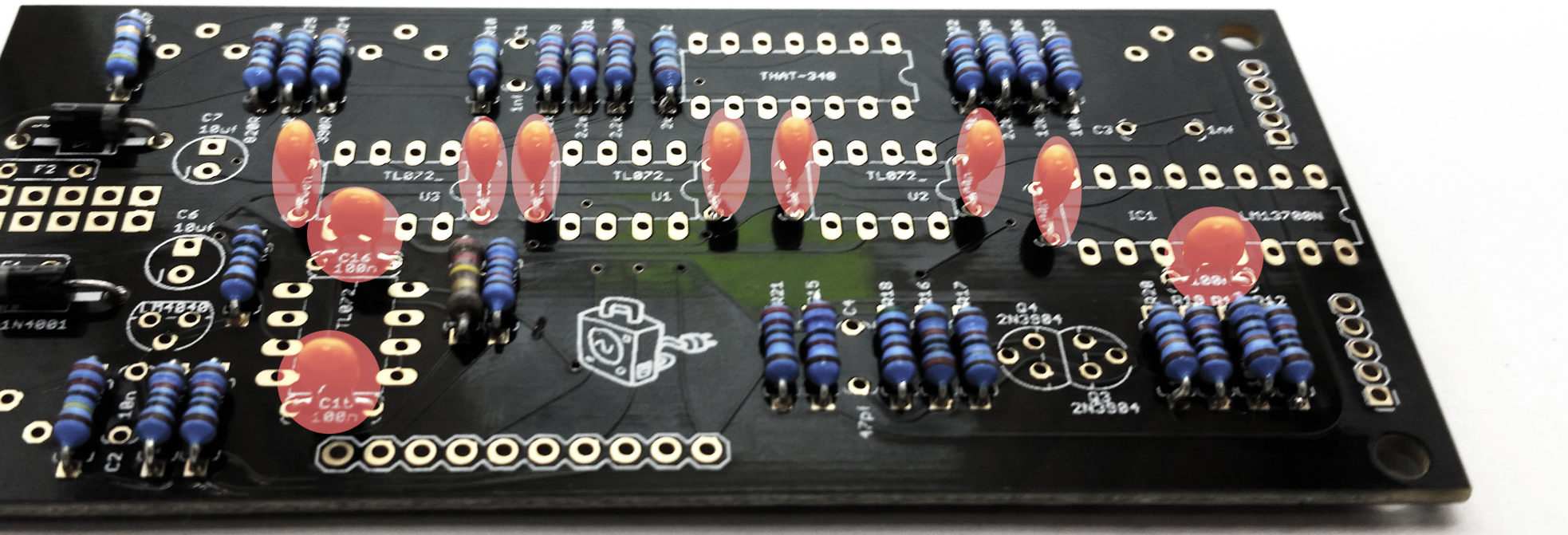
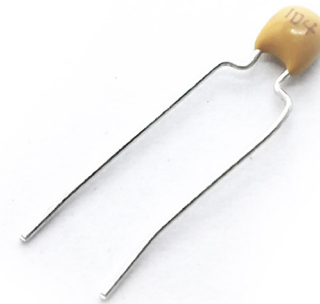


# Step 23

On components PCB:

solder all bypass capacitors  
(100nF ceramic)

10x Capacitor  
100nF





# Step 24

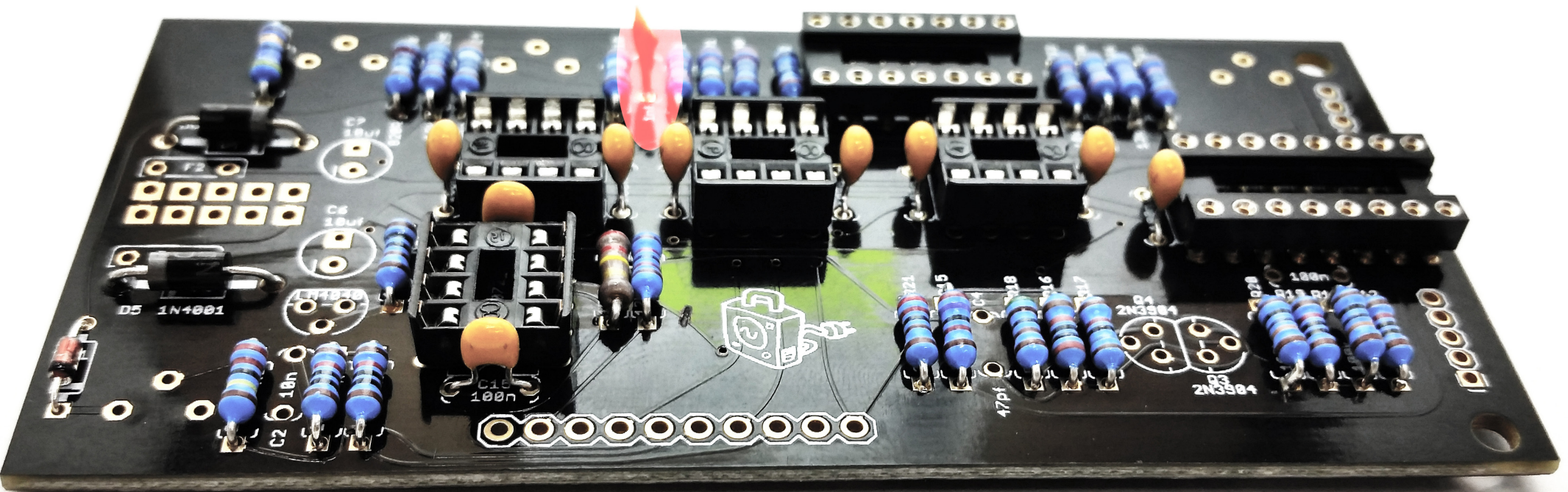
On components PCB:

solder C1 with 1nF.

Careful – do not confuse  
with C3!

1x Capacitor

1nF





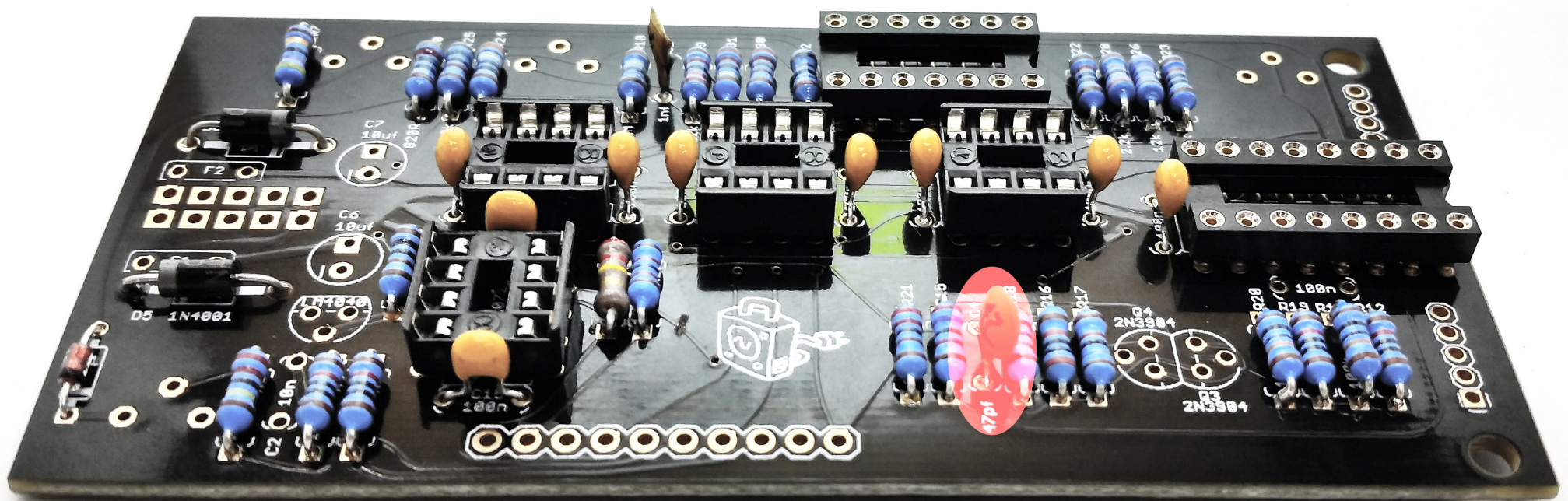
# Step 25

On components PCB:

solder C4 with 47pF

(print says 47)

1x Capacitor  
47pF





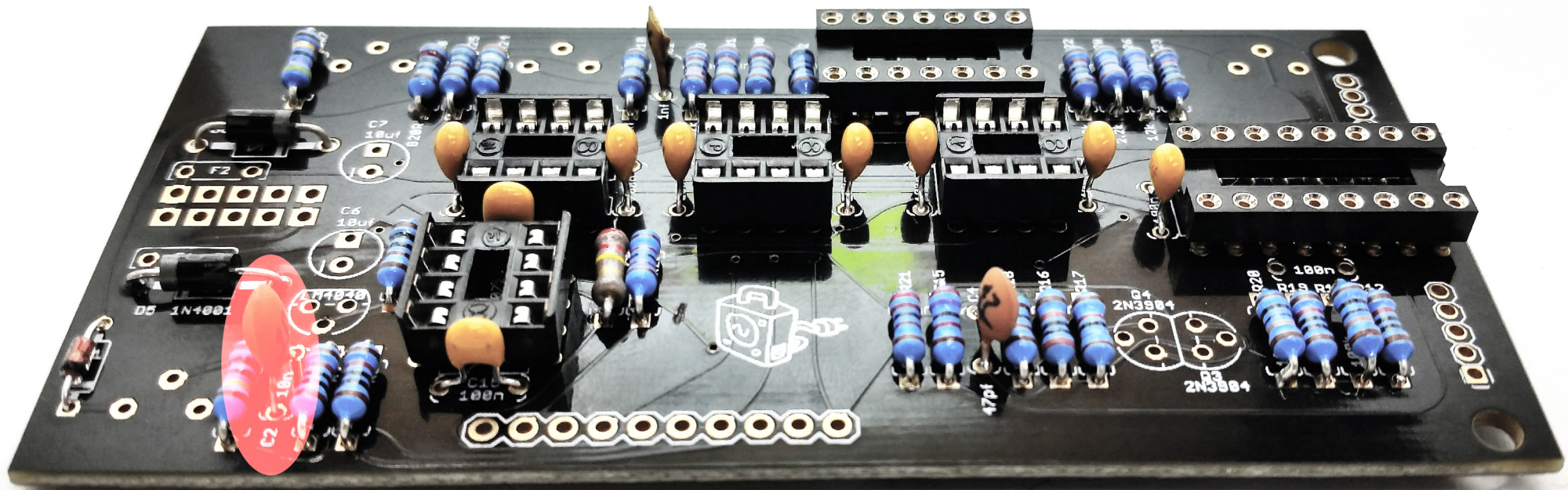
# Step 26

On components PCB:

solder C2 with 10nF

(print says 103)

1x Capacitor  
10nF





# Step 27

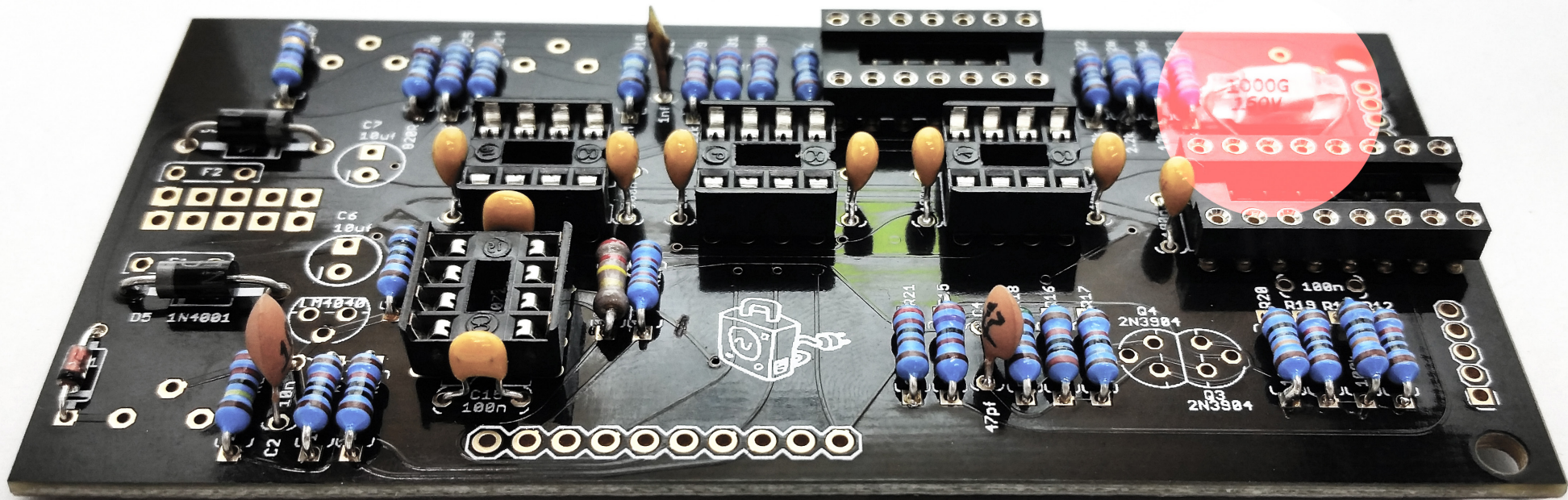
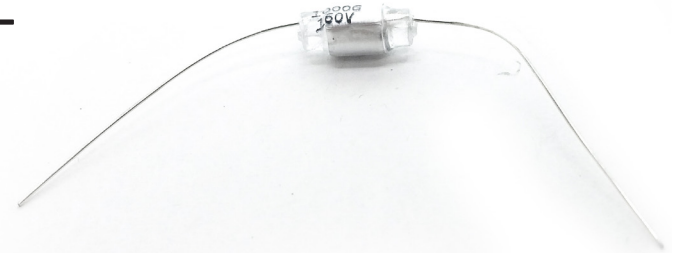
On components PCB:

solder C3, 1n Styrofoam/Styroflex/Glimmer.

(This Capacitor is for tuning and should be somewhat stable with changing temperature).

The value of this capacitor can be adjusted for a different frequency range. If you don't need frequencies higher than 13 kHz, but want improved LFO capabilities use 2.4nF.

## 1x Capacitor 1nF





# Step 28

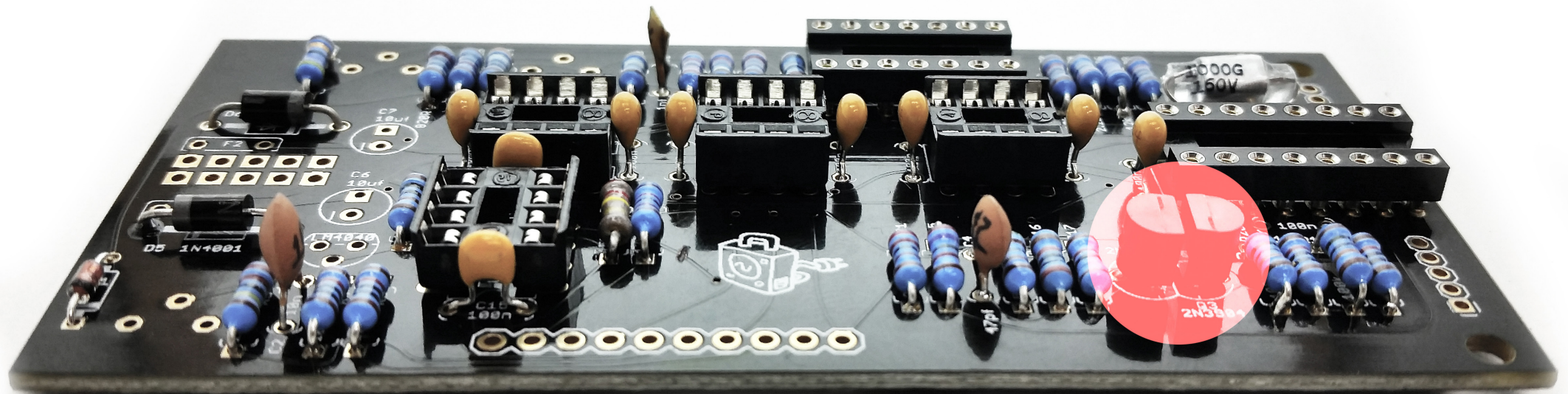
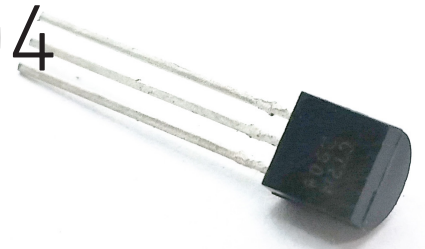
On components PCB:

solder the Transistors Q3 and Q4, 2N3904.

Careful to orient them according to the silkscreen.

Q3 and Q4 are the heart of the discrete schmitt trigger of the oscillator core.

2x Transistors  
2N3904



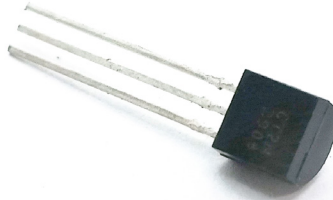


(THIS STEP IS ONLY NEEDED IF YOU DON'T USE THE THAT340 CHIP)

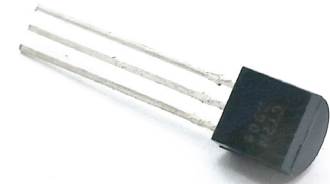
## Step 28B

If you intend to use matched transistors instead of the THAT340 chip, solder the transistors according to the graphic below.

2x matched PNP-Transistors  
2N3906



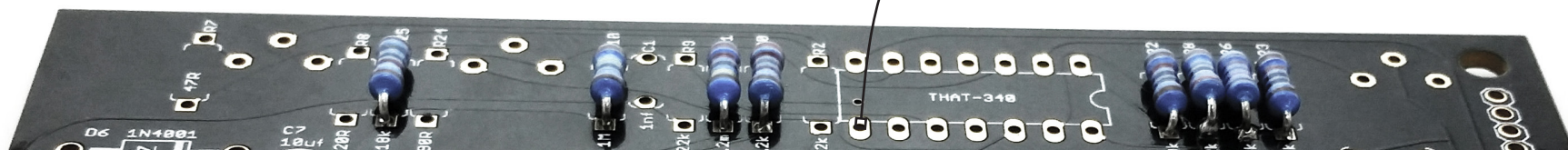
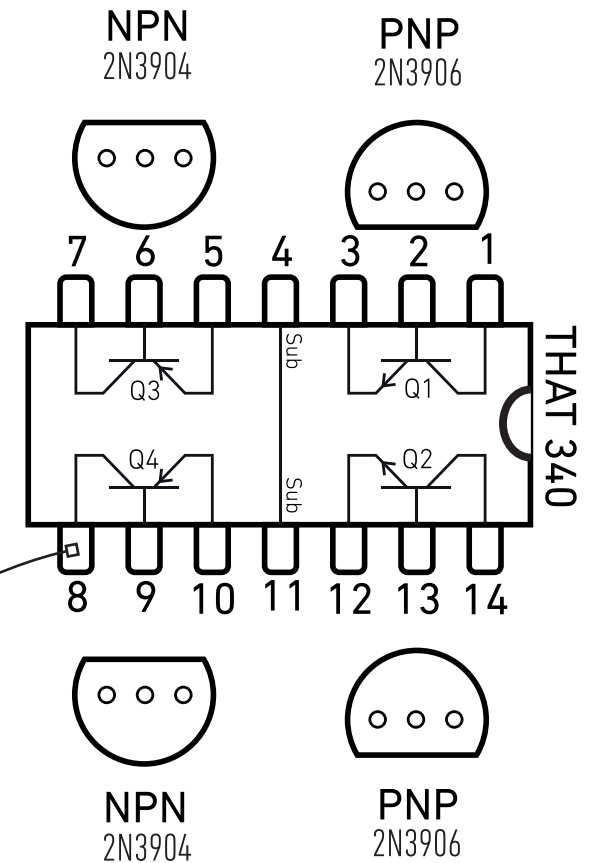
2x matched NPN-Transistors  
2N3904



Also solder your 3300ppm  
2k Tempco now.

All three parts (NPN-pair  
and tempco) need to be in  
thermal contact, which can  
be achieved by glueing them  
together with either epoxy  
or cyanacrylate.

These 4 Transistors  
form the exponential  
converter and are essen-  
tial to achieve 1V/Oct track-  
ing. Having them matched  
improves Temperature  
stability.





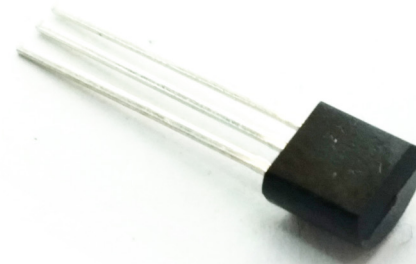
# Step 29

On components PCB:

solder the precision voltage  
reference LM4040-10.0 in.

Careful to orient them ac-  
cording to the silkscreen.

## 1x Voltage Reference LM4040-10.0





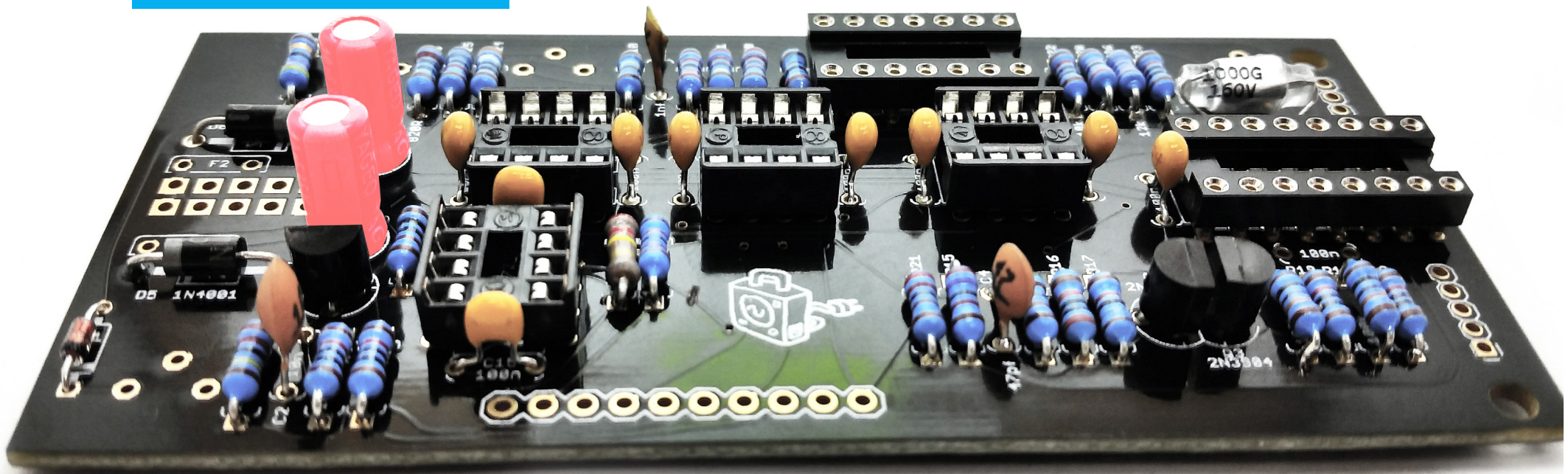
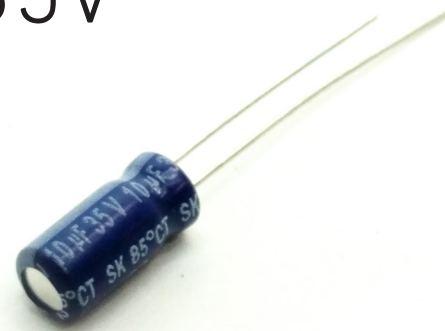
# Step 30

On components PCB:

solder the electrolytic capacitors C6 and C7 with 10uF.

Careful with the orientation. The white stripe needs to go to the round pad. The longer leg goes to the square pad. Make sure they are flat with the PCB otherwise you will run into problems later.

## 2x Electrolytic Capacitor 10uF 35V



# Step 31

On components PCB:

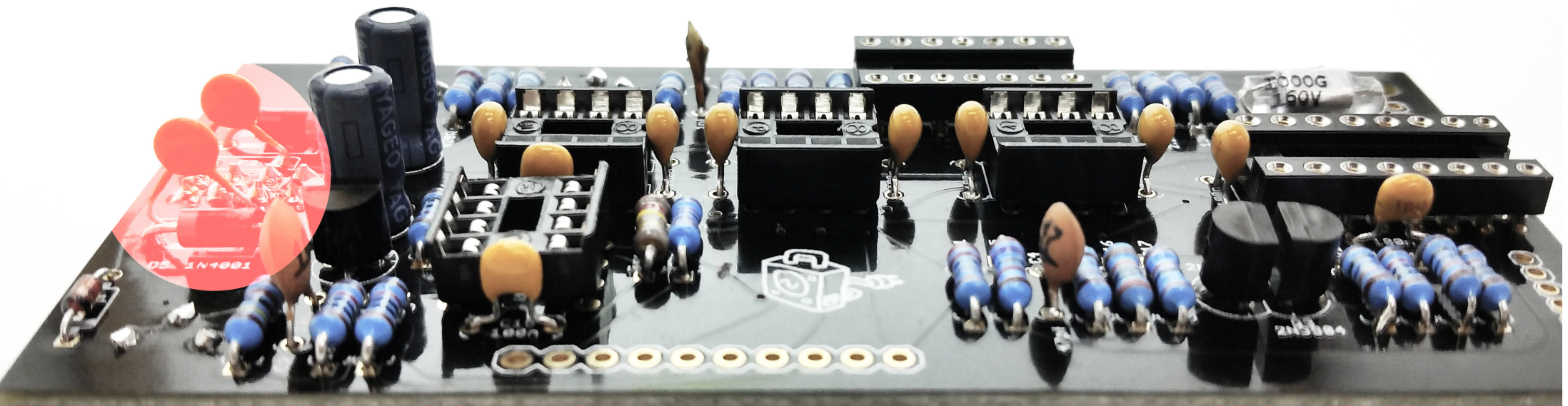
On the top side of the PCB  
solder the two polyfuses

(print says R005 BJ6S)

## 2x Polyfuses



Polyfuses are a  
resettable fuse,  
which protect  
the circuit from  
overcurrent





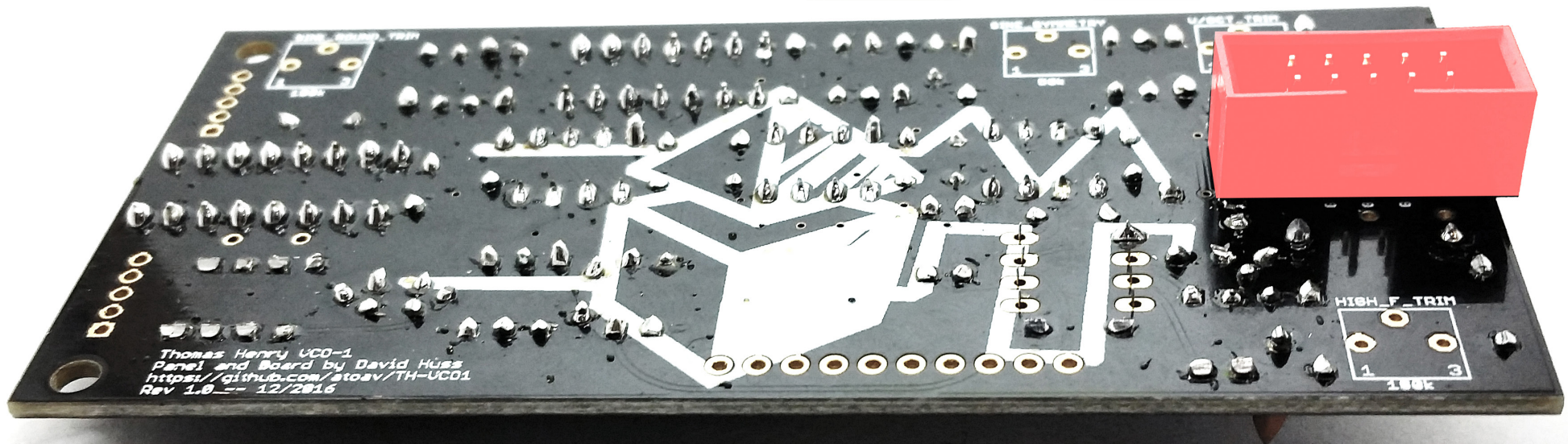
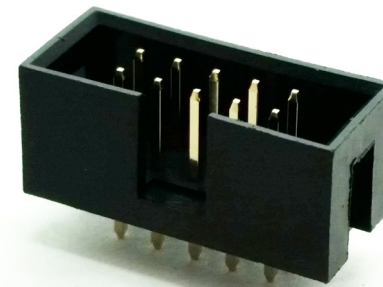
# Step 32

On components PCB:

turn the PCB around and solder one leg of the power connector to the bottom of the PCB.

Make sure the opening is pointing left.

1x Power Headers  
10-Pin, 2 Rows, 2.54mm



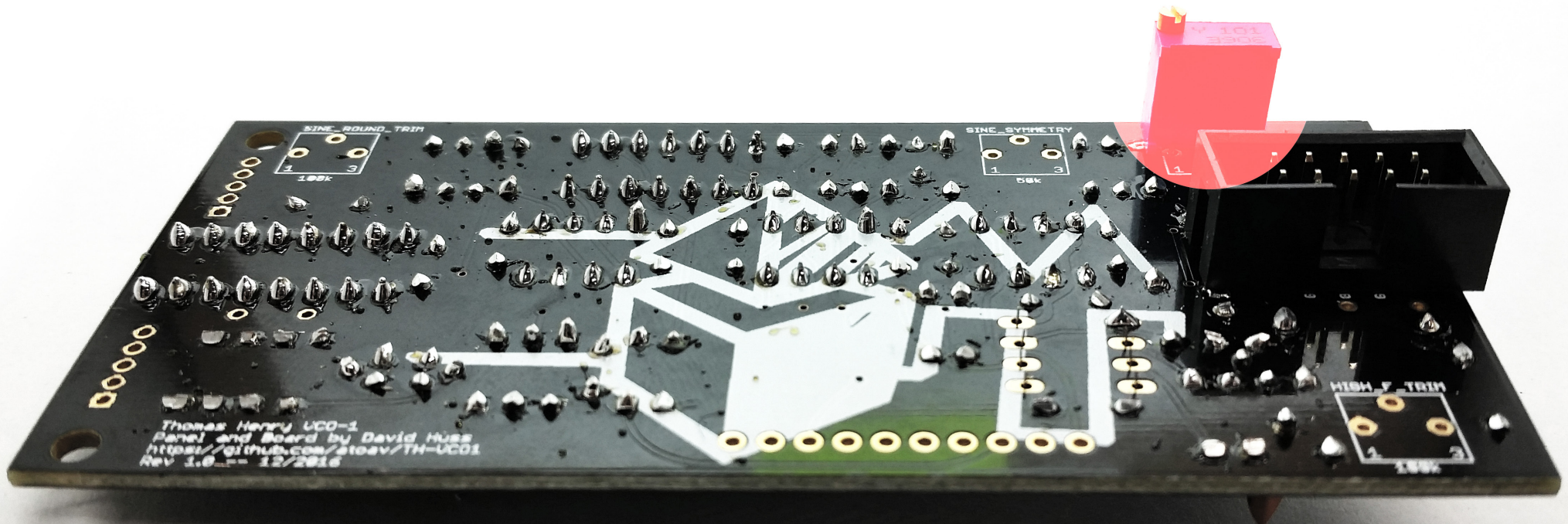
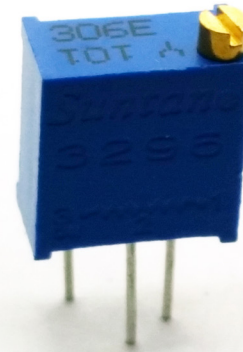
# Step 33

On components PCB:

On the bottom side of the PCB solder the 100R Trimmer (print says 101) in.

Make sure it sits firmly on the PCB.

1x Trimmer  
100R





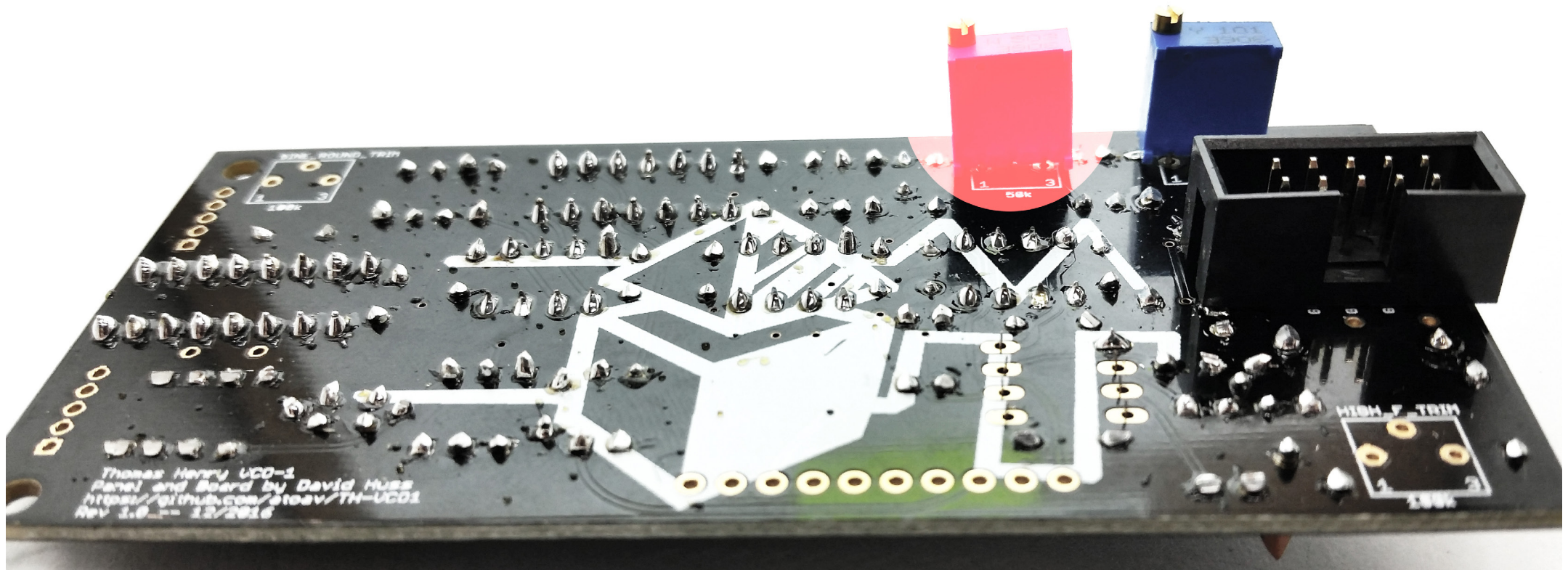
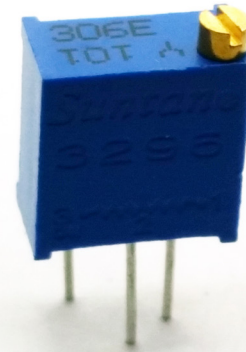
# Step 34

On components PCB:

On the bottom side of the PCB solder the 50k Trimmer (print says 503) in.

Make sure it sits firmly on the PCB.

1x Trimmer  
50k



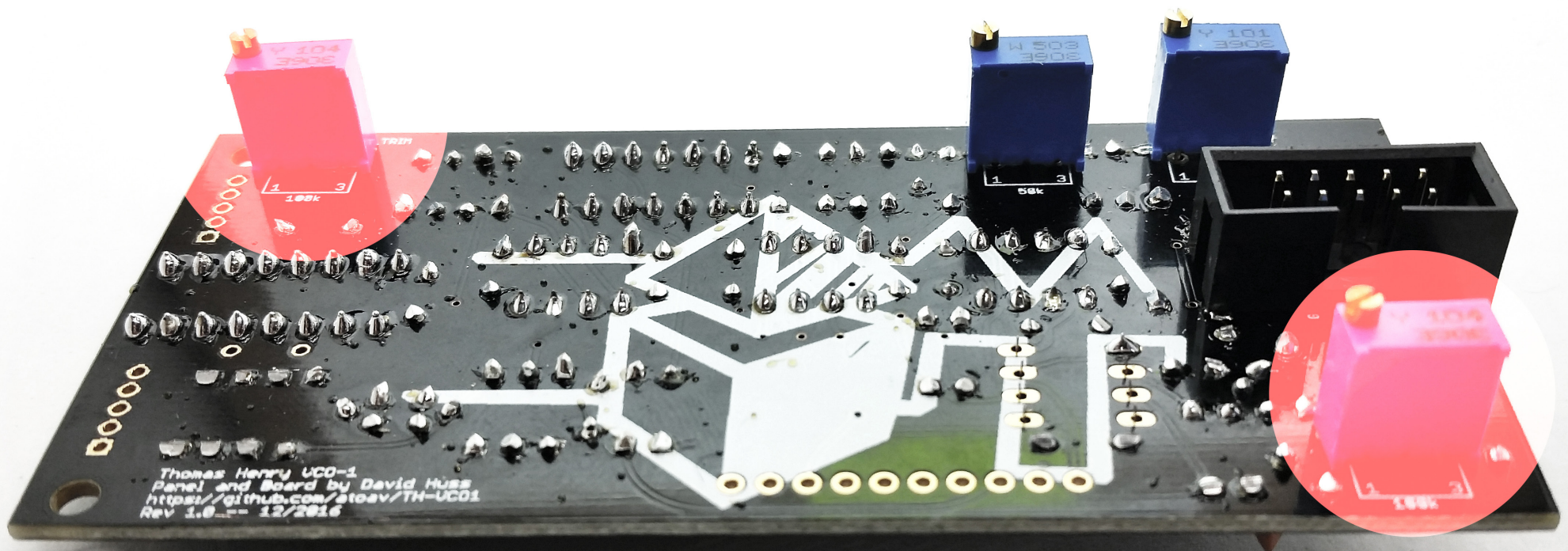
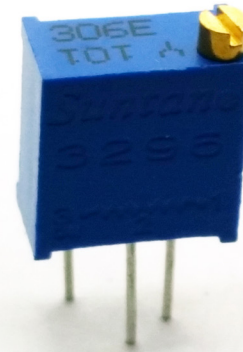
# Step 35

On components PCB:

On the bottom side of the PCB solder the two 100K Trimmers (print says 104) in.

Make sure they sit firmly on the PCB.

2x Trimmer  
100k





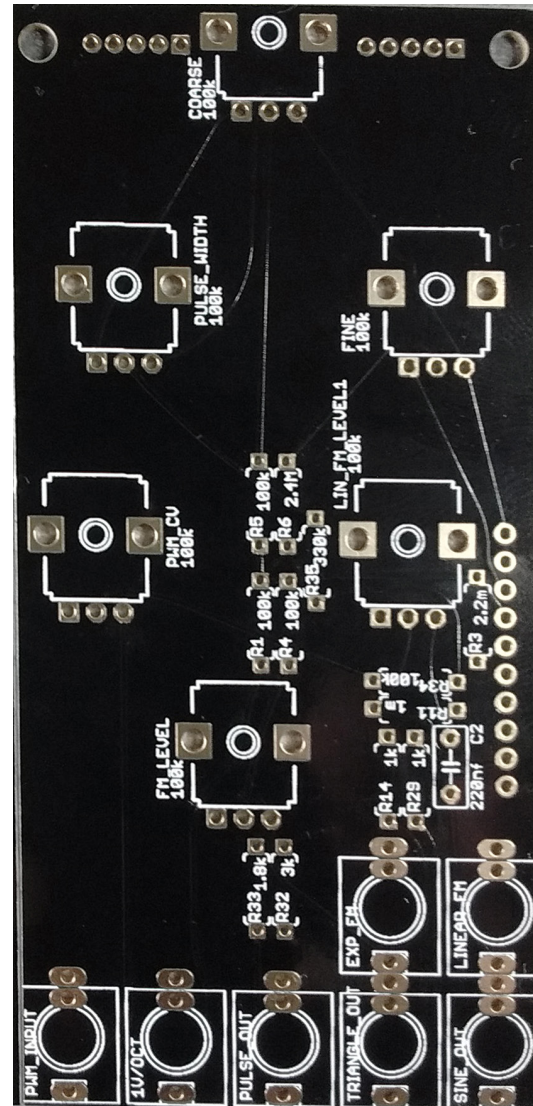
# Step 36

Take the other (pots) PCB

# Step 36

Take the other (pots) PCB

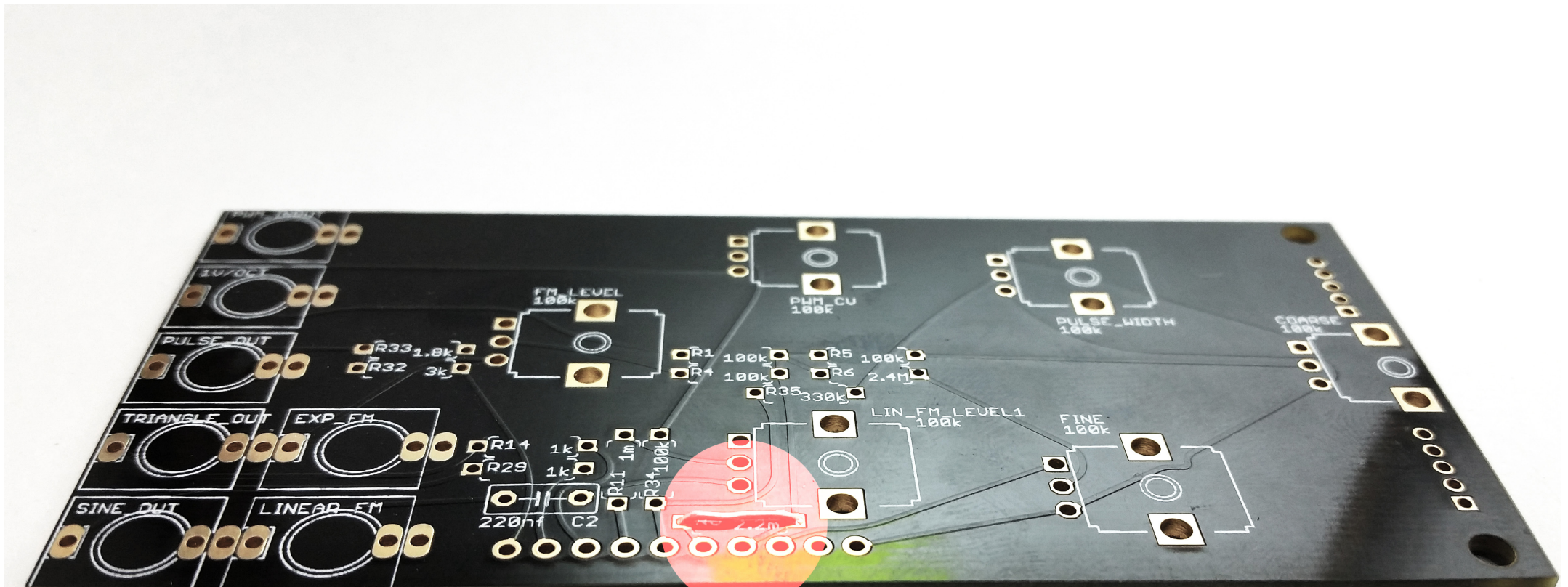
1x PCB



# Step 37

On pots PCB:

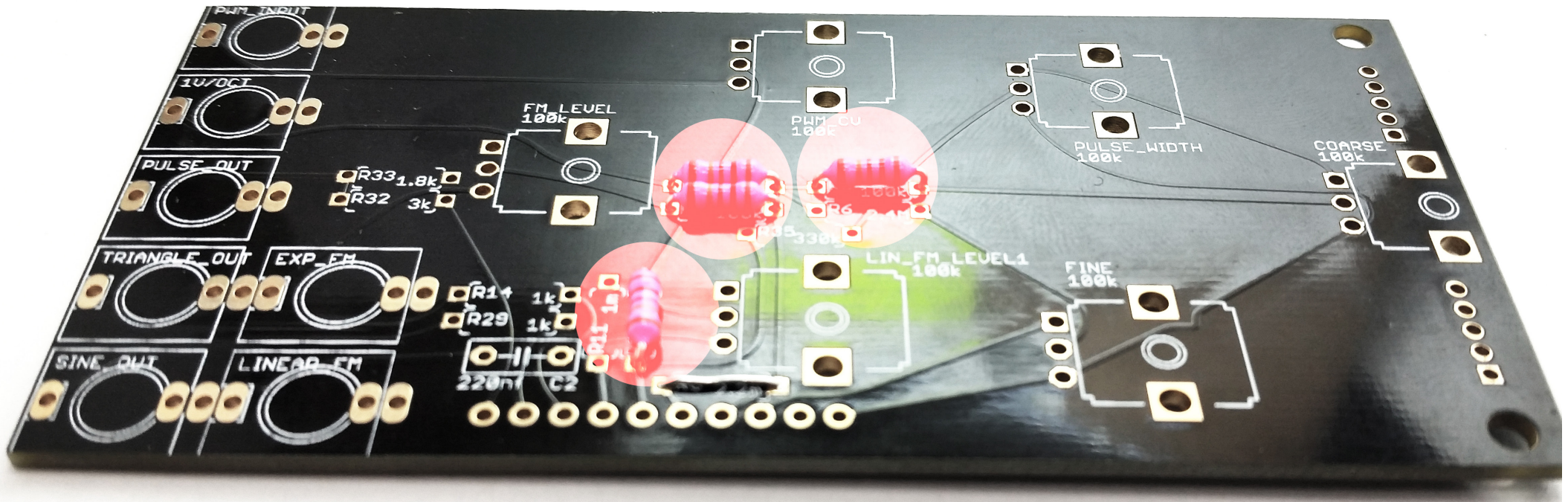
resistor R3 (2.2M) is not needed, bridge the contacts with a piece of wire (solder it there)





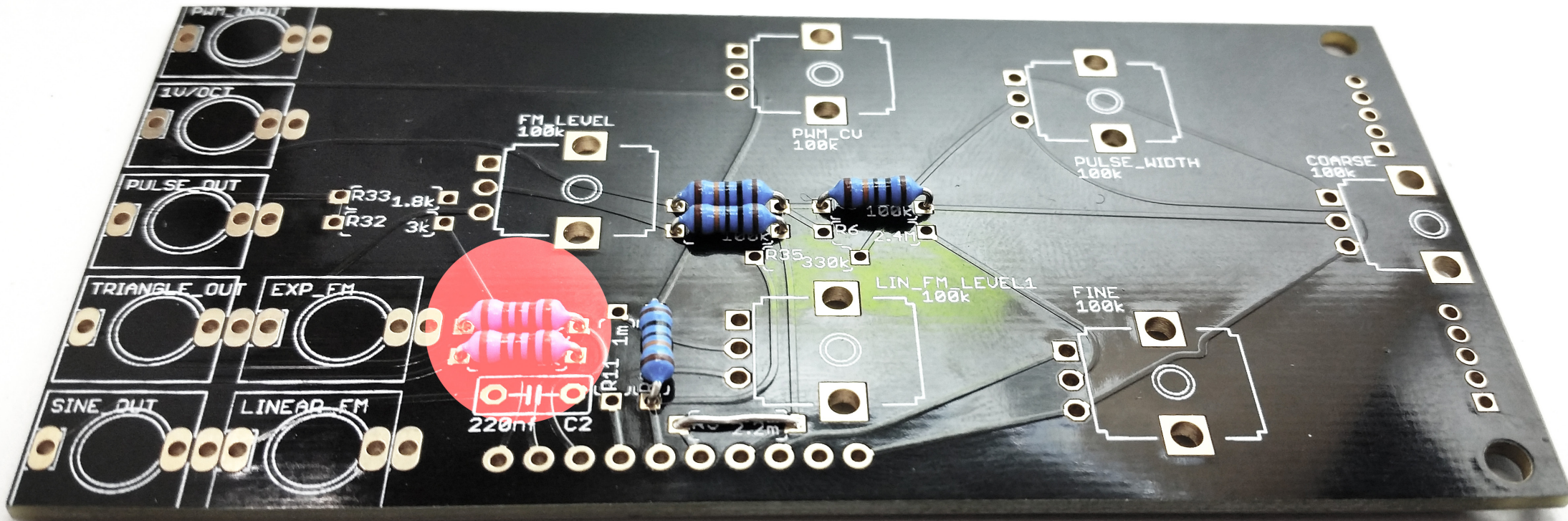
solder all 100k resistors (4 pieces) in.

4x Resistor  
100k



solder all 1k resistors (2 pieces) in.

1k



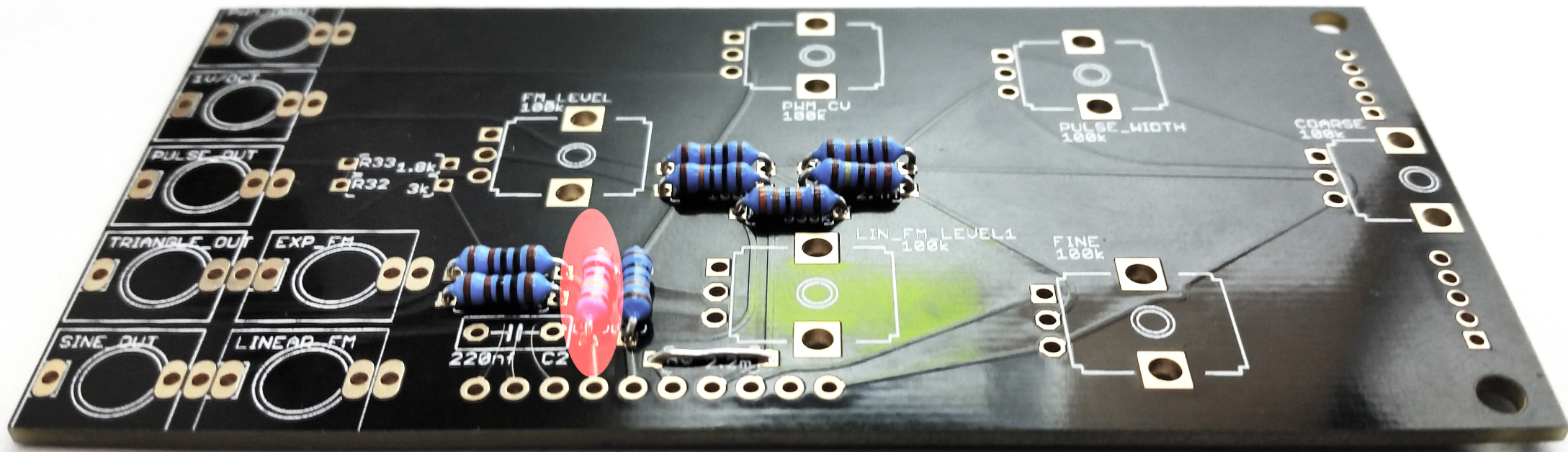


# Step 40

On pots PCB:

solder R11 with 1M

1x Resistor  
1M

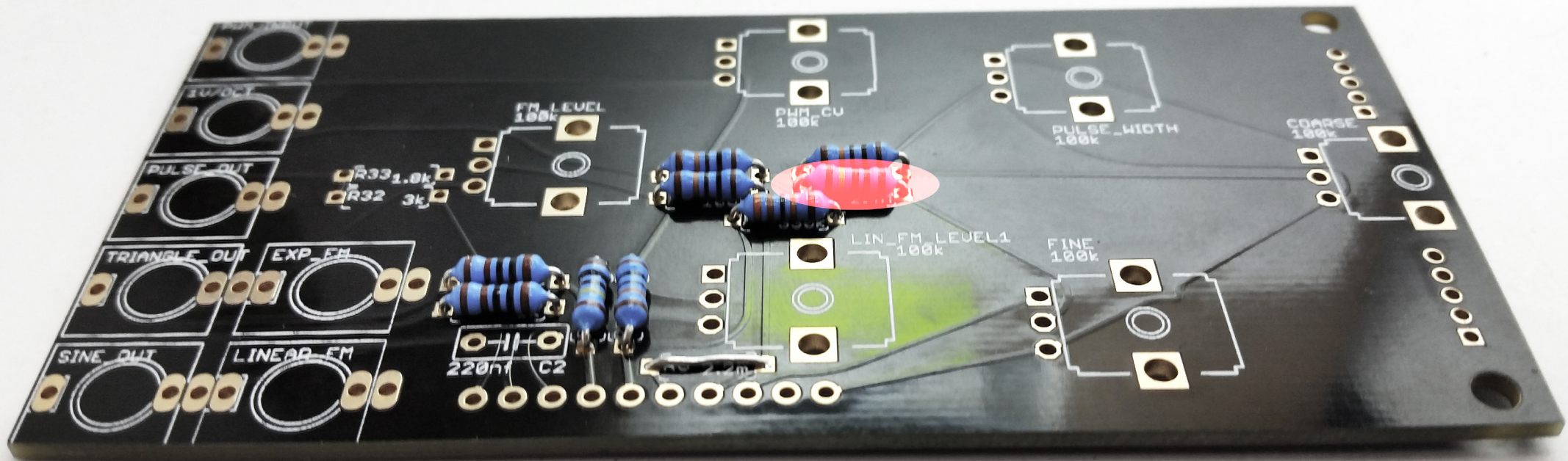


# Step 41

On pots PCB:

solder R6 with 2.4M (or 2.2M  
if no 2.4M available)

1x Resistor  
2.4M or 2.2M



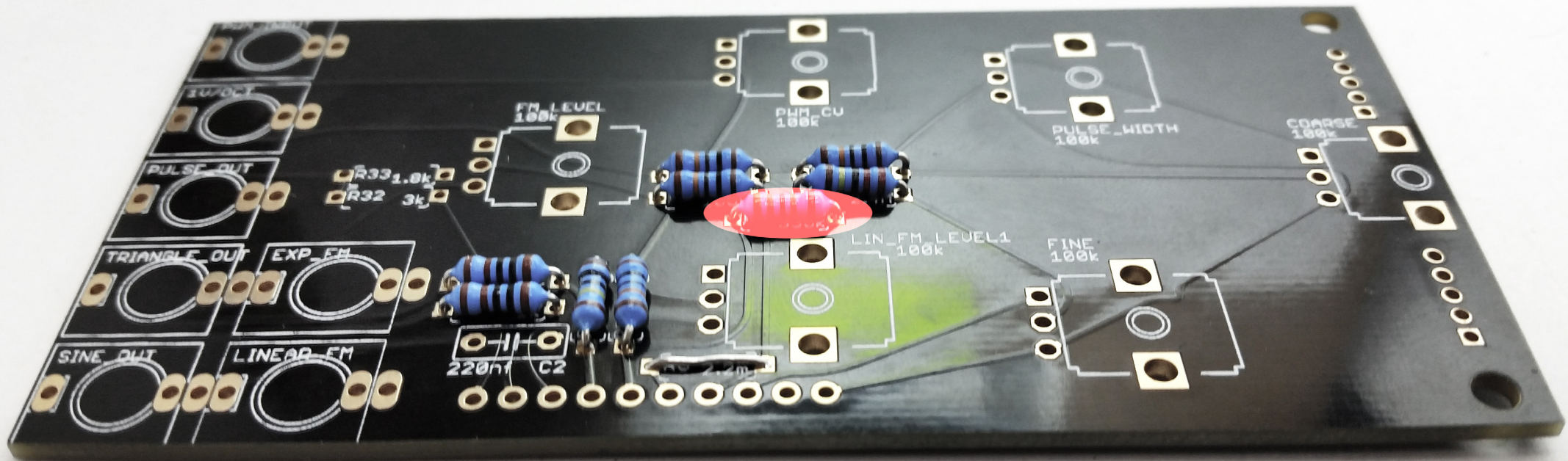


# Step 42

On pots PCB:

solder R35 with 330k

1x Resistor  
330k

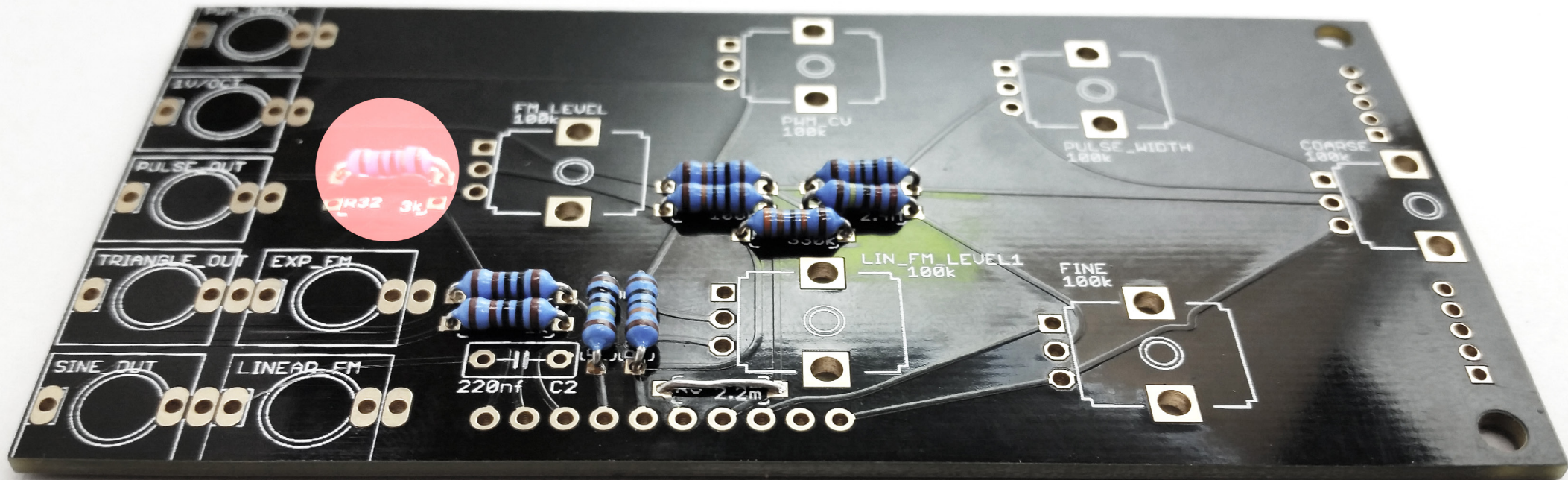


# Step 43

On pots PCB:

solder R32 with 1.8k

1x Resistor  
1.8k



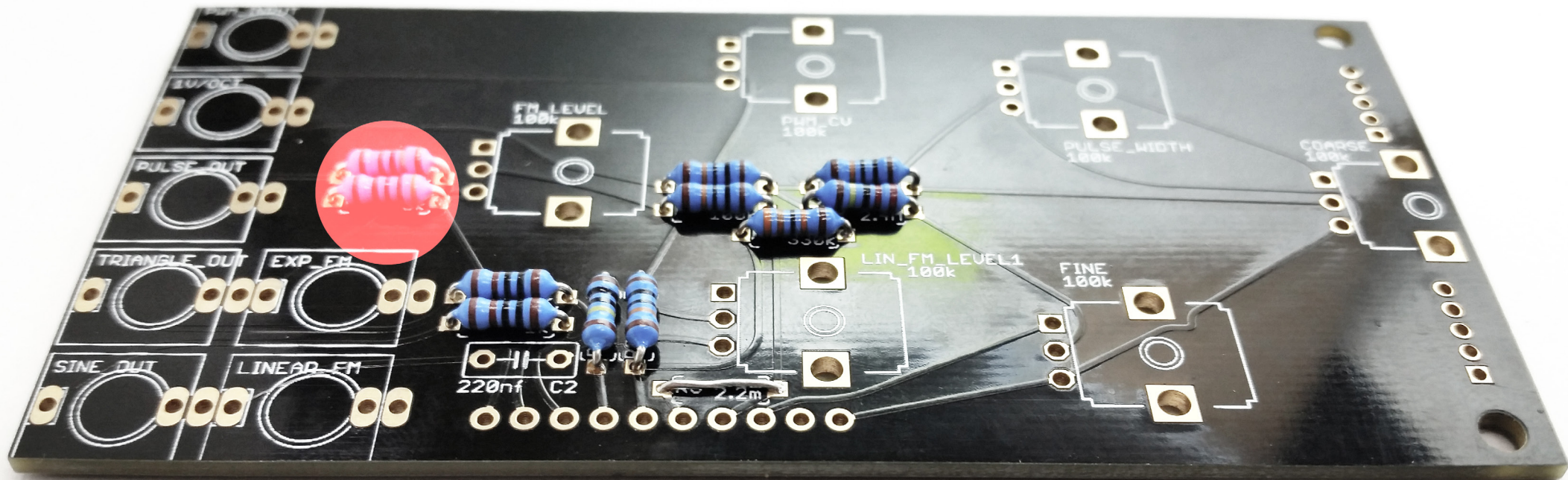


# Step 44

On pots PCB:

solder R33 with 3k

1x Resistor  
3k

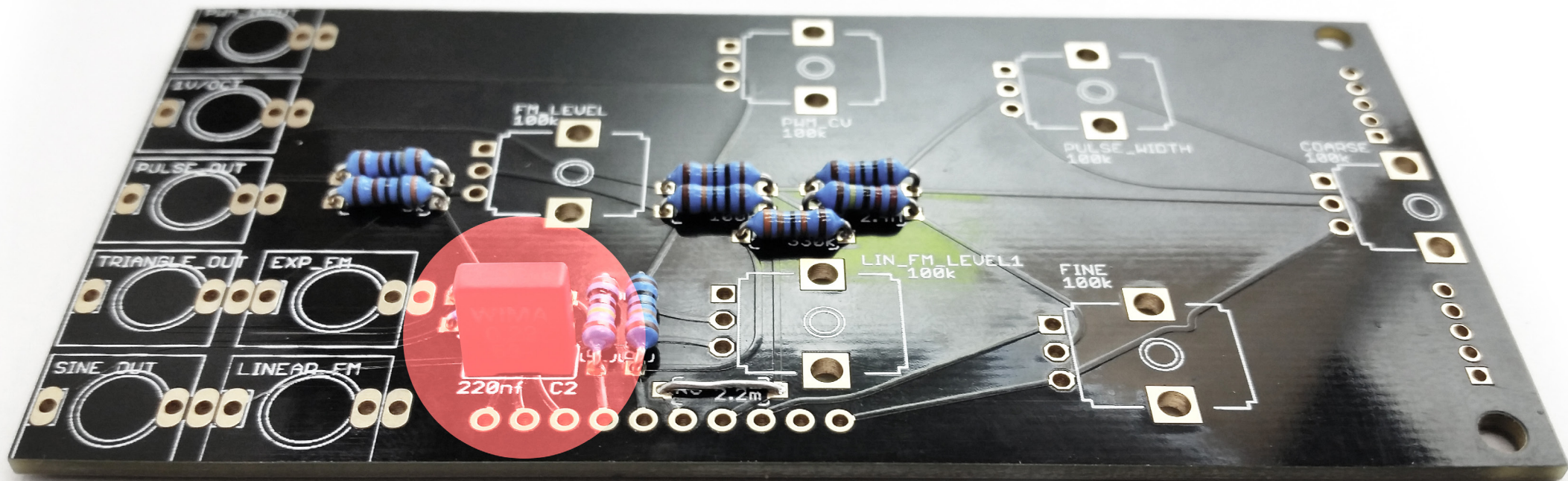
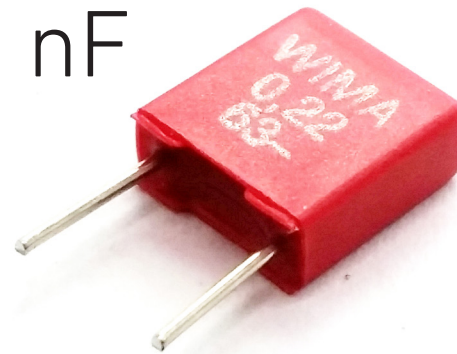


# Step 45

On pots PCB:

solder C2POTS with 220nF  
(print says 0,22 63-)

1x DC Block Capacitor  
220 nF





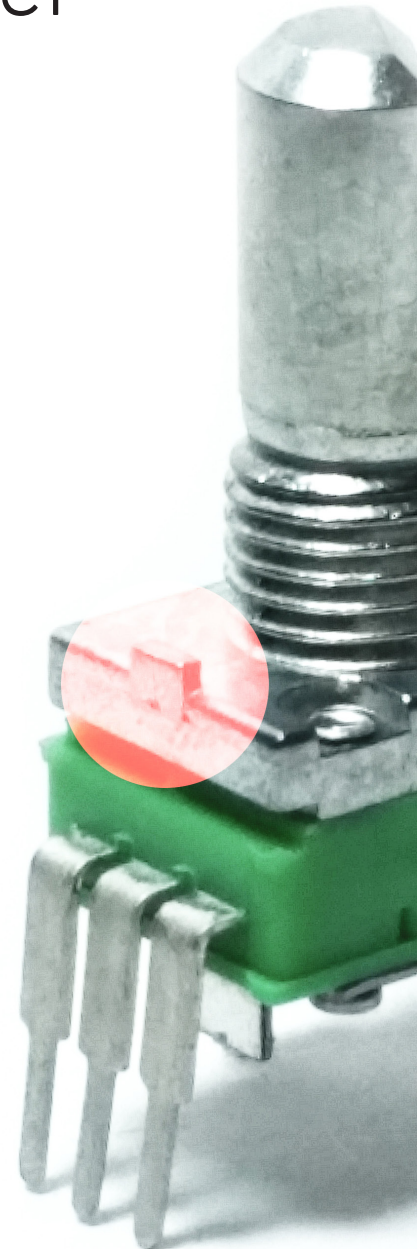
# Step 46

On pots PCB:

If your pots have a small metal piece, break it off with pliers (twisting usually works well).



## 5x Potentiometer 100k



# Step 47

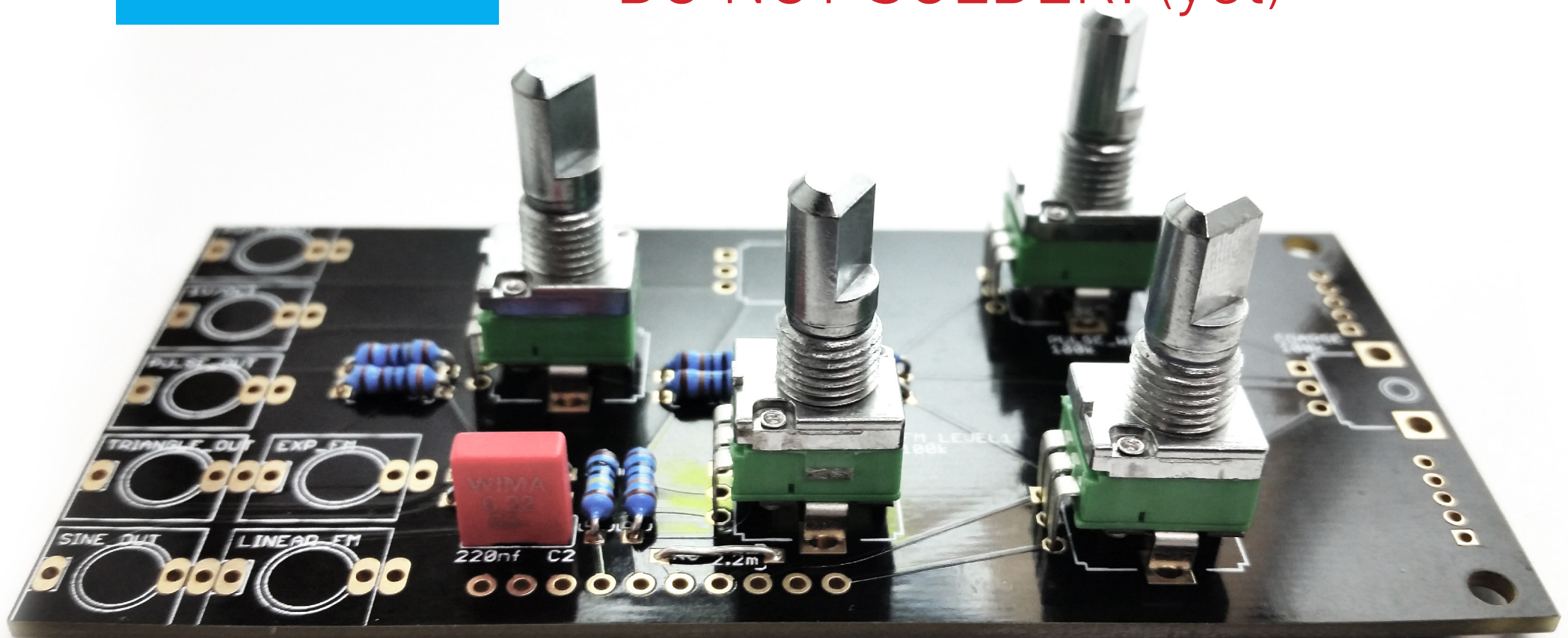
On pots PCB:

Do not solder: push in the green pots (except PWM\_CV!)

5x Potentiometer  
100k



DO NOT SOLDER! (yet)



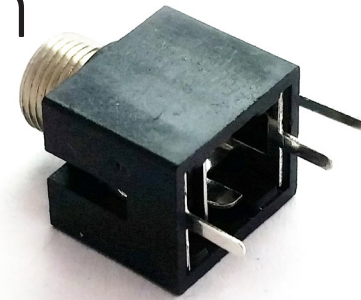


# Step 48

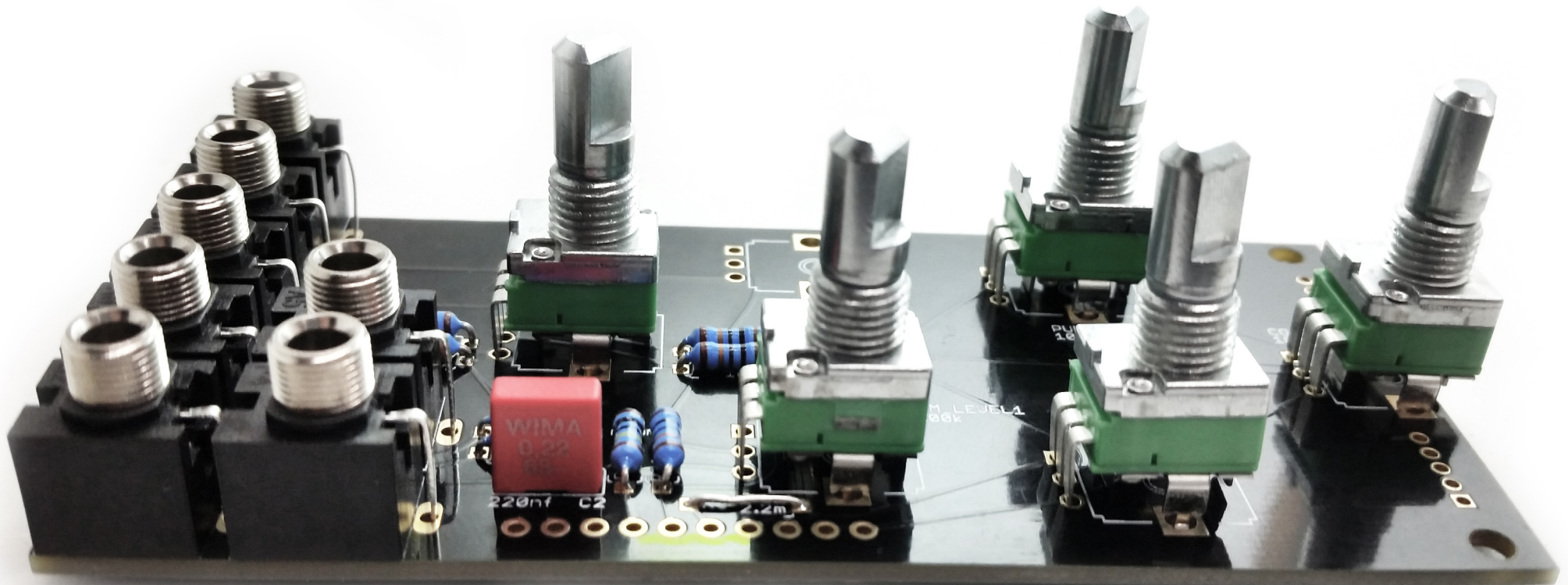
On pots PCB:

Do not solder: push in the  
Jacks

7x Jack  
3.5mm



DO NOT SOLDER! (yet)



# Step 49

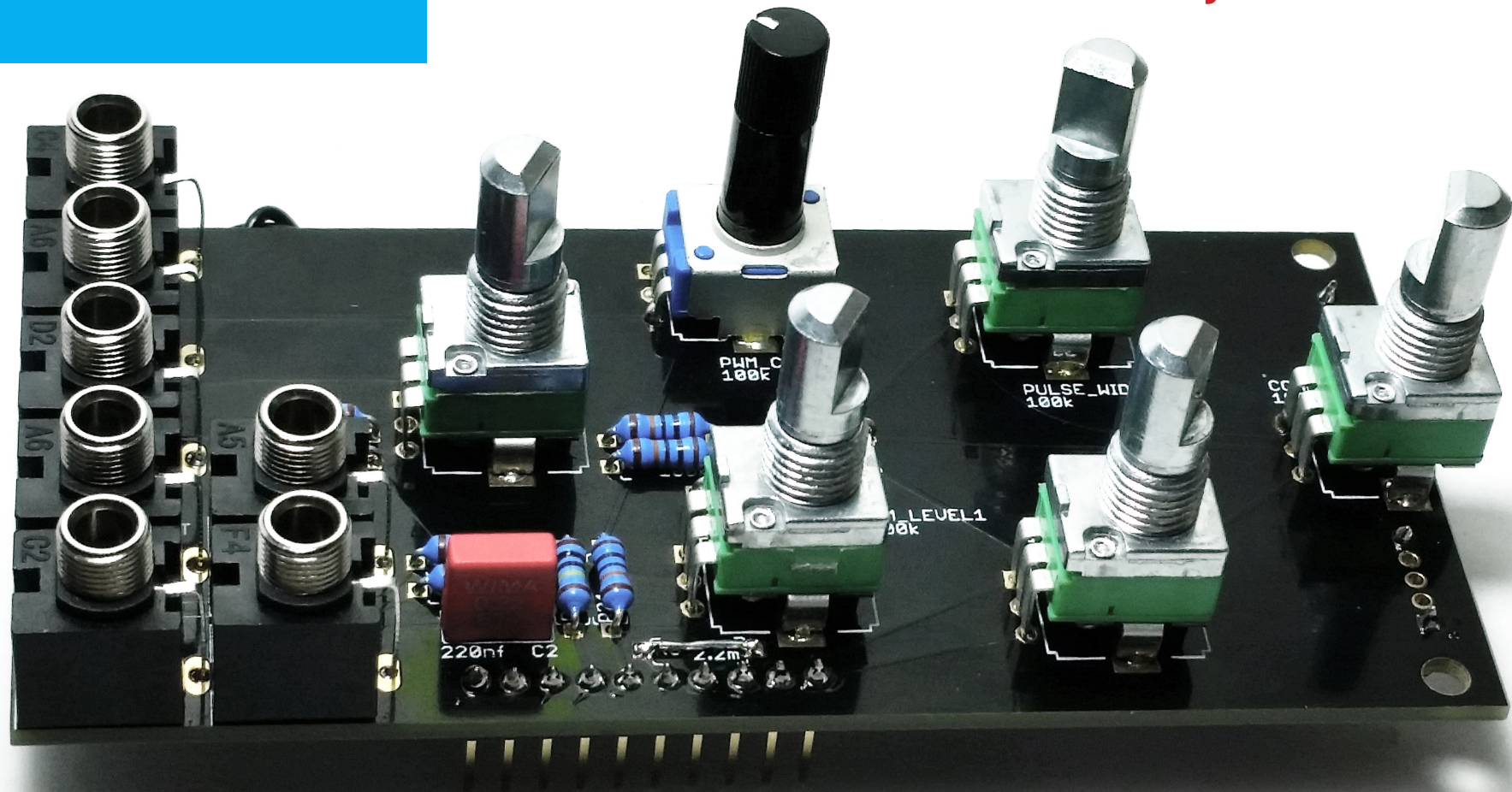
On pots PCB:

Do not solder: push in the  
pot with plastic shaft

1x Potentiometer  
100k



DO NOT SOLDER! (yet)





# Step 50-52

On pots PCB:

Put the Panel onto Pots and Jacks. Screw it on with the fingers

Solder Pots and jacks (make sure everything sits tightly, before soldering the first things). Be careful not to apply too much solder at the jacks.

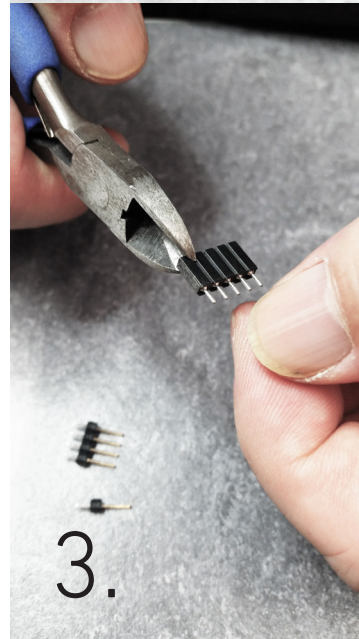


# Step 53

On pots PCB:

Sadly the 5-Pinheaders have the wrong spacing : (  
Split the two 5-pin headers (2x male and 2x female) into four 1 pieces.

Use a cutter first, then use two pliers to break them appart.



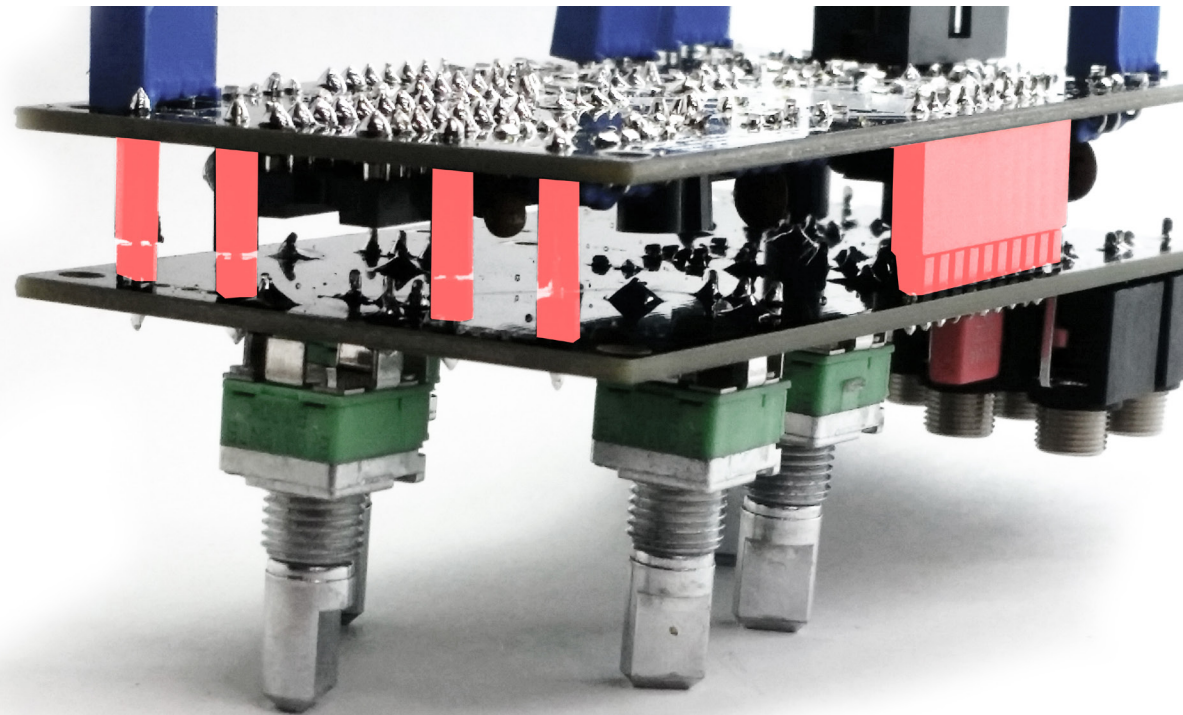


# Step 54

Take the Panel off again.

Stack both PCBs together with the pinheaders inbetween and solder the pinheaders (if you are not sure they are straight, only solder one pin).

For the single Pins, solder only one first.



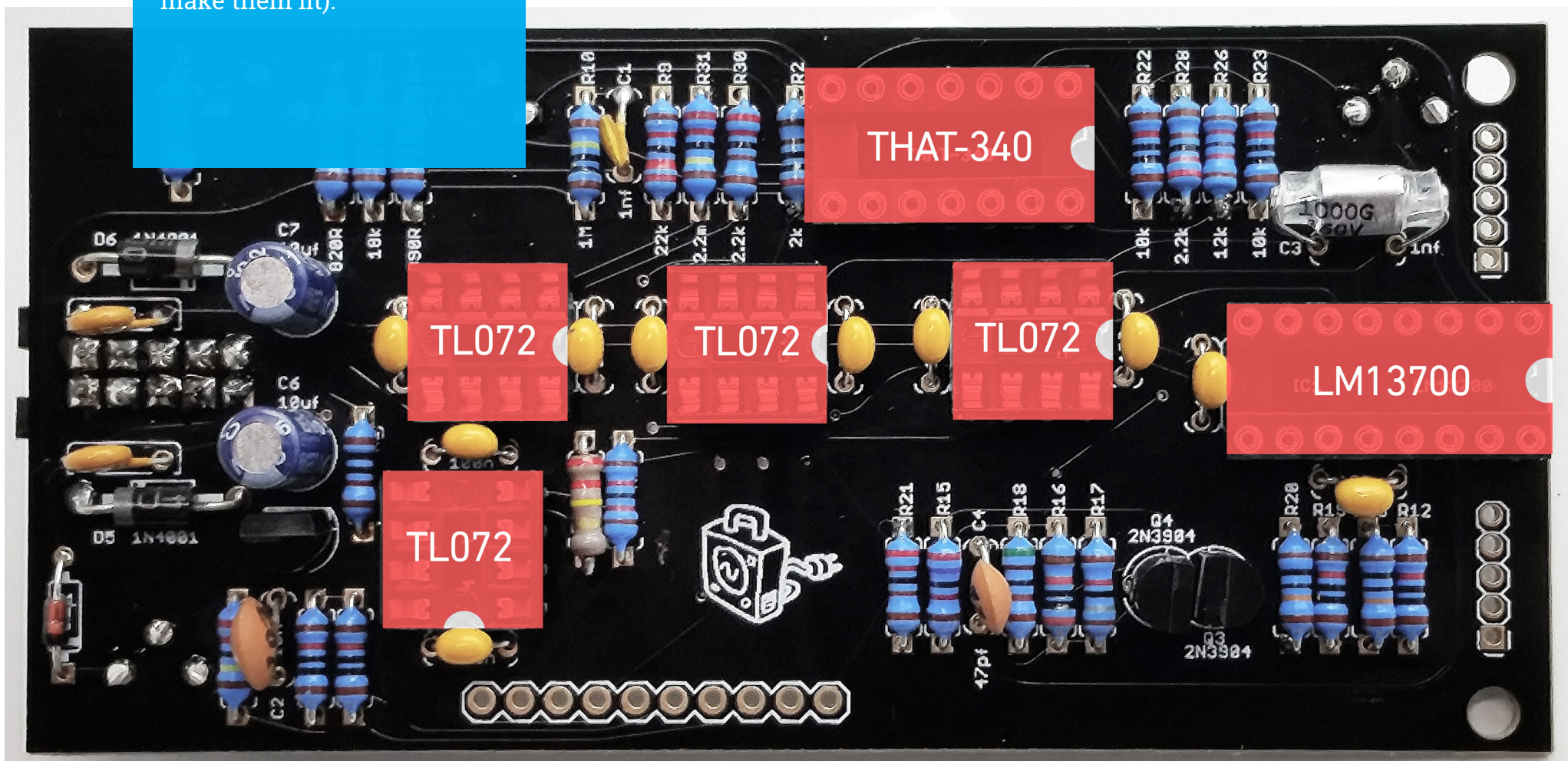
# Step 55

On components PCB:

Put the ICs into the sockets

(WATCH FOR THE ORIENTATION! you may have to carefully bend the IC legs to make them fit).

TL072 are dual Opamps, the LM13700 is a OTA and the THAT340 contains mono-lithic matched transistors

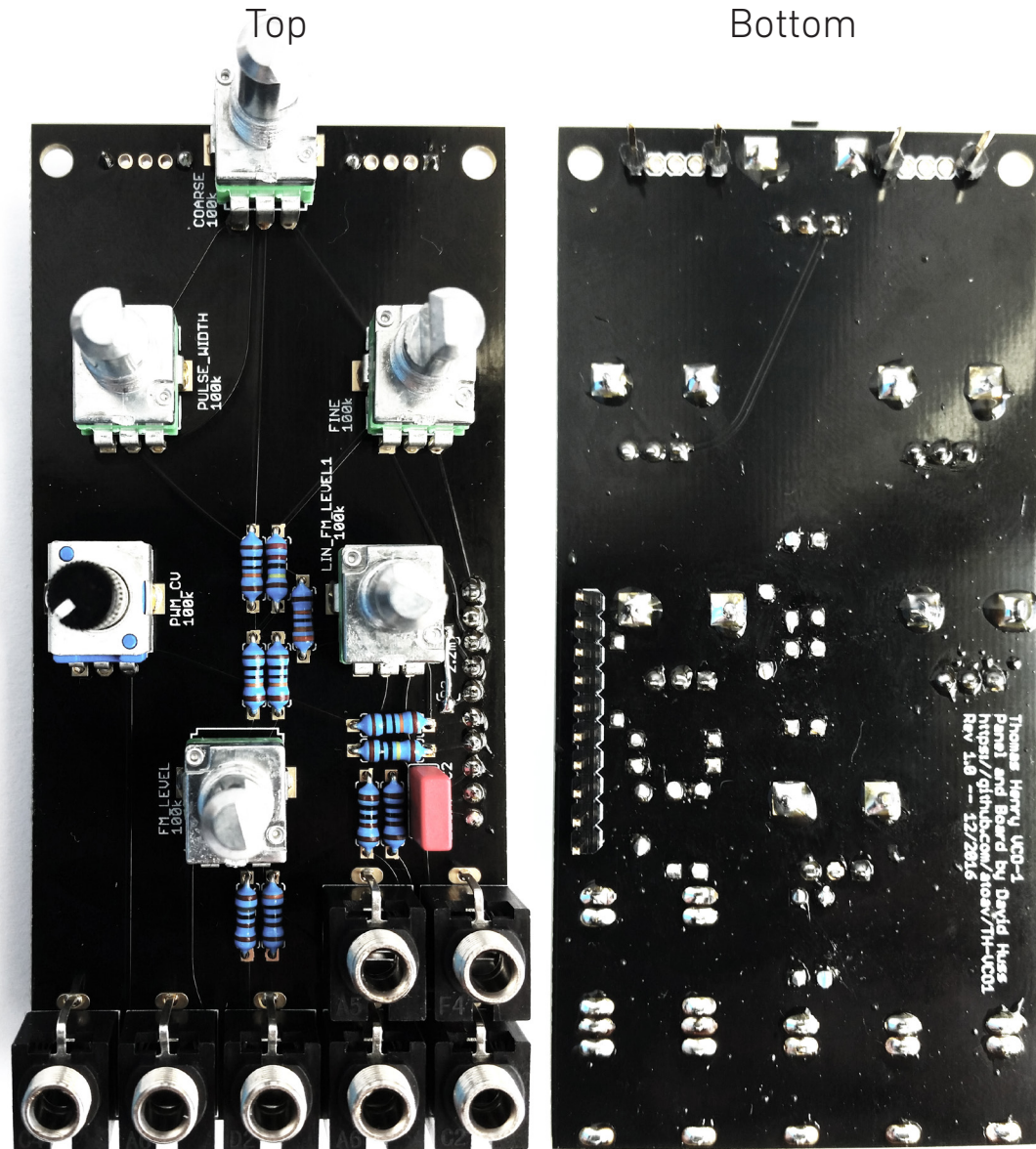




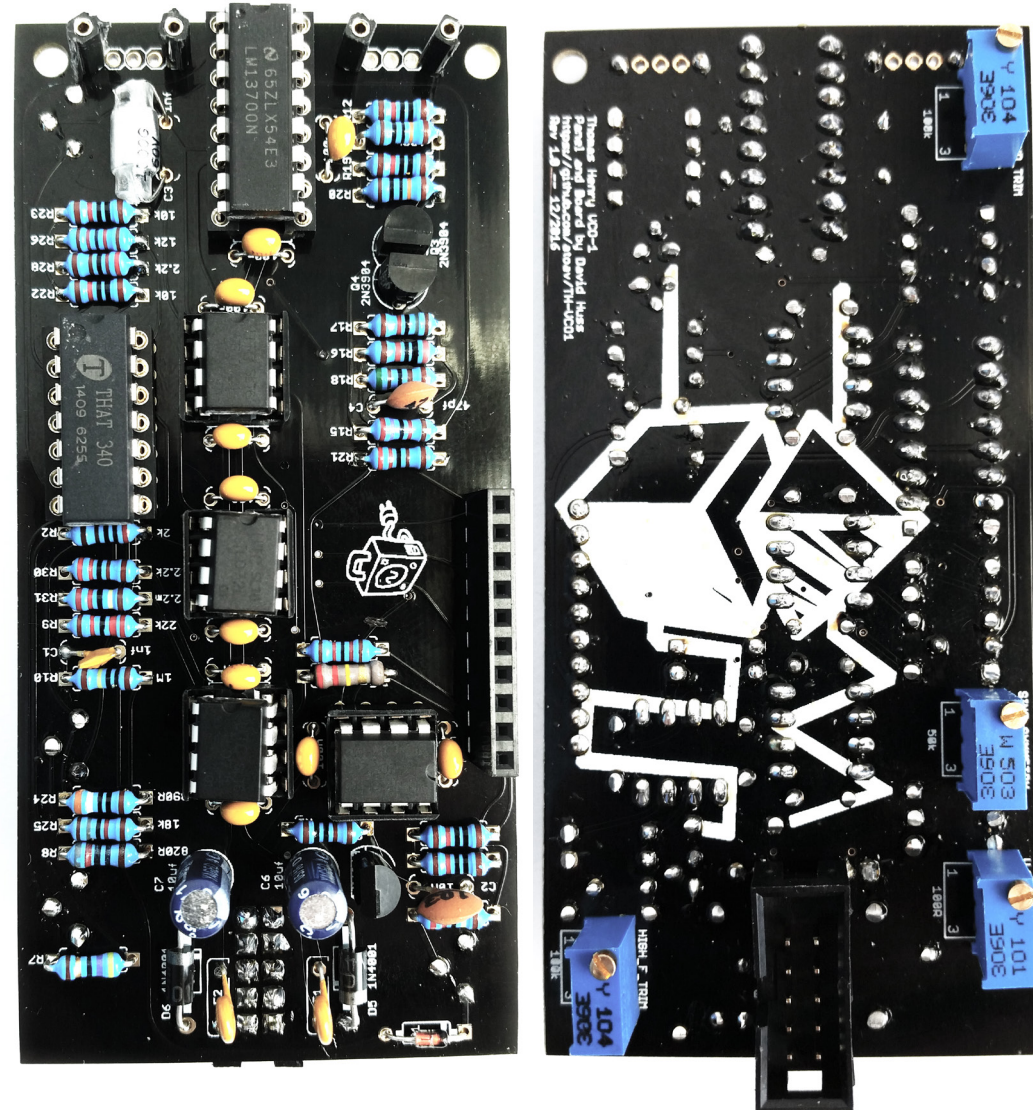
# Step 56

Control all Boards. They should look somewhat like this. Note that the brown resistor on this board has a different value than your's.

Pots PCB  
Top Bottom



Components PCB  
Top Bottom



# Step 57

Stack everything back together. Screw the Panel on.

Push on the knobson . If the knob is too loose, put a little piece of paper, isolating tape or plastic on the shaft before pushing the knob on.





# Step 58

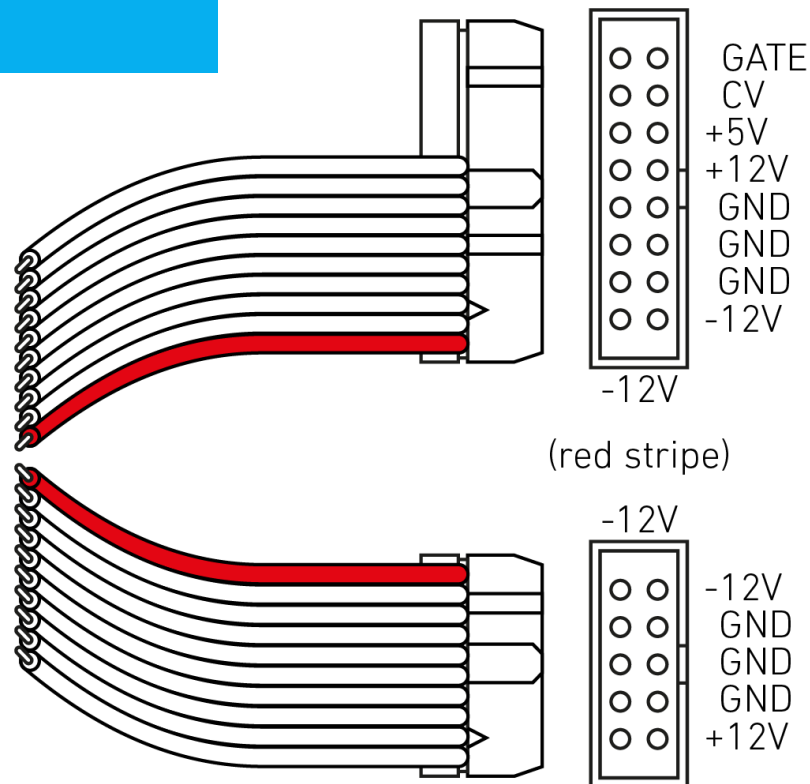
Assemble the powercable.

Make sure every wire is at the right place and press the connector together.

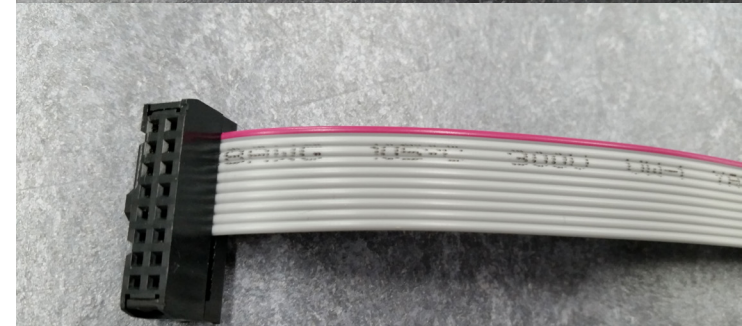
If possible measure for connectivity and shortcuts before using it! Make sure the pinout fits your busboard!

Make a conductivity test with a multimeter.

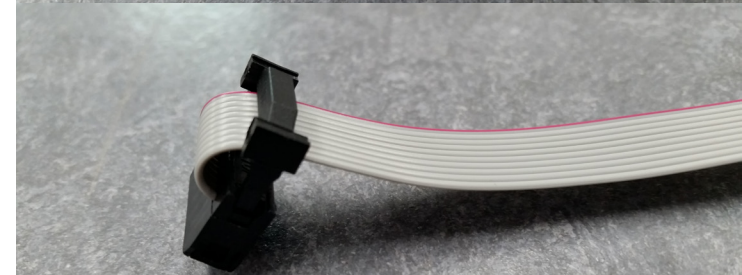
-12V should be connected only to -12V, +12V should be connected only to +12V and Ground should be only connected to Ground.



1. Place ribbon cable and press together



2. Place ribbon cable and press together



3. Fold cable around, clip in strain relief



4. Fold cable around, clip in strain relief

# TESTING

Congratulations! You have new VCO now. But does it work? If you have a lab power supply (with the possibility to limit current to roughly 55mA for each rail), try your module with limited current first.

Make sure the polarity of the power cable is right (if in doubt – measure! searching for broken parts afterwards is harder than being careful from the start!)

If your module smokes, delivers no output or draws way to much current, switch off your Powersupply and search for Errors :)

For Calibration check the Calibration Instructions.

