Din Sync .Info

Dear customer,

thank you for purchasing the VCF303 partial kit. I hope you have fun building and using it. Before you rush off and start soldering please take a moment to read through this manual to get a feel for what's ahead.

During the build, please take your time and double check everything before committing to soldering. Of course it will take slightly longer but I honestly think extra time spent now to make sure everything is correct is the best way. It's certainly much better than rushing the build and then spending an unknown amount of time trying to find a problem this caused. I suppose my point here, take your time, it'll be worth it.

I really hope the completed module brings you many hours of enjoyment in the future.

regards

Paul

# A little background and some notes about the module itself.

To date this has probably been the single best selling module from DinSync.info and I personally built each one currently on the market by hand myself. It has been out of production for a couple of years since I have since moved to more automated processes so now seems like a good idea to open it up to the DIY crowd.

Many have commented that it's the most authentic TB-303 sounding eurorack filter and this might be true, much of this will be down to part selection since essentially it is the same circuit (albeit adapted for eurorack) The only real difference is that a typical eurorack system has a very clean and stable power supply, this wasn't the case in the TB-303 and would probably account for the remaining differences in tone.

A few things to note about the build, at the time of designing the module I wasn't able to find one single manufacturer of potentiometers that had all the values I needed (I also did not have the funds to have them custom made in small numbers) So this why is there is a little mix and match there.

The use of metal film polyester caps was to match the original ones used in the TB-303. To my ears this makes a difference in the tone, in actual use it may only be subtle. Personally I feel these small differences add up. For example I used 945 npn transistors as used in the later production runs of the TB compared to for example the x0xb0x which uses 536 npn transistors as used in the earlier runs of the TB. The 945 to my ears sounds more acidic and metallic and my reasoning is that most people would probably prefer the more acidic sounding model than the more realistic bass guitar tone as was intended.

If you have the original matched pairs of course use them. However the matched pairs supplied in the kit probably are as good as using the originals. Unfortunately the originals are now grossly overpriced. I spent some time working out the best pairs as substitutes so you shouldn't worry too much if you decide to use them, every commercially built VCF303 on the market uses them also.

Finally the output is inverted, this is quite an amusing mistake I made that was never fixed. I was at the time toying with the idea of a high pass mode. Of course there is no discernable difference in the sound in normal use but with a bit of patching you can easily make the filter into a highpass one. To do this take a multiple of the input signal and mix it with the output.

# Preparing for the build

A basic set of tools, you could get by with less but this would be comfortable to work with.  From left to right we have,
nutdriver (for the jack nuts)
socket driver for the m3 nuts
solder
small screwdriver for fixing the knobs
2.5mm hex wrench
small pliers
wire clipper (flush cut)
panavise jr
dremel tool
tip cleaning pad (dry)
soldering iron

a larger selection of tools, this includes more stands, and some removing tools in case you needed to repair some mistakes. However you don’t need these extras, although the electrical tape will be needed later on.

# Do you have everything?

Ok before we start let's make sure you have everything you need.

Your partial kit should have four parts,

1: Aluminium front panel and pcb set (wrapped in newspaper)

2: One static protection bag containing
- 2SA733P x 4
- 2SC945P x 15
- LED x 1
- 2291A/B discrete pair (red) x 1
- 1583A1/B1 discrete pair (blue) x2

3: One small bag containing
- INLINE JACK x 7
- KNURLED NUT x 7
- 10MM NYLON STANDOFF x 8
- NYLON WASHER x 12

4: One large bag containing
- 16MM HEX HEAD BOLT x 4
- 10 PIN HEADER M x 1
- 20 PIN HEADER M x1
- 20 PIN HEADER F x 1
- DUAL POT x 1
- POWER CABLE
- M3 SCREWS x 8

if anything is missing, please contact me immediately on info@dinsync.info and I'll take care of it for you.

Additionally to this you should also have sourced the contents of both the TOP-BOM and BASE-BOM (or just the contents of TOTAL-BOM which is the top and base BOMs combined for convenience) These are supplied with this document as PDF and xlsx for your convenience.

# One last thing

Right I’m sure you are itching to get started but before that a few words. I’ve tried to make the build manual as easy as possible, in all steps you will see a placement drawing, some descriptive text and if needed for clarification a photograph..

Some level of soldering competence is assumed, if you’ve never soldered before you should probably practise some before attempting the kit. There are some great soldering tutorials online if you search for them. I recommend you take a look at some if you’ve never soldered before, at least if only to learn some best practises.

Ok let’s get on with the build, but before we get started we need to do some preparation

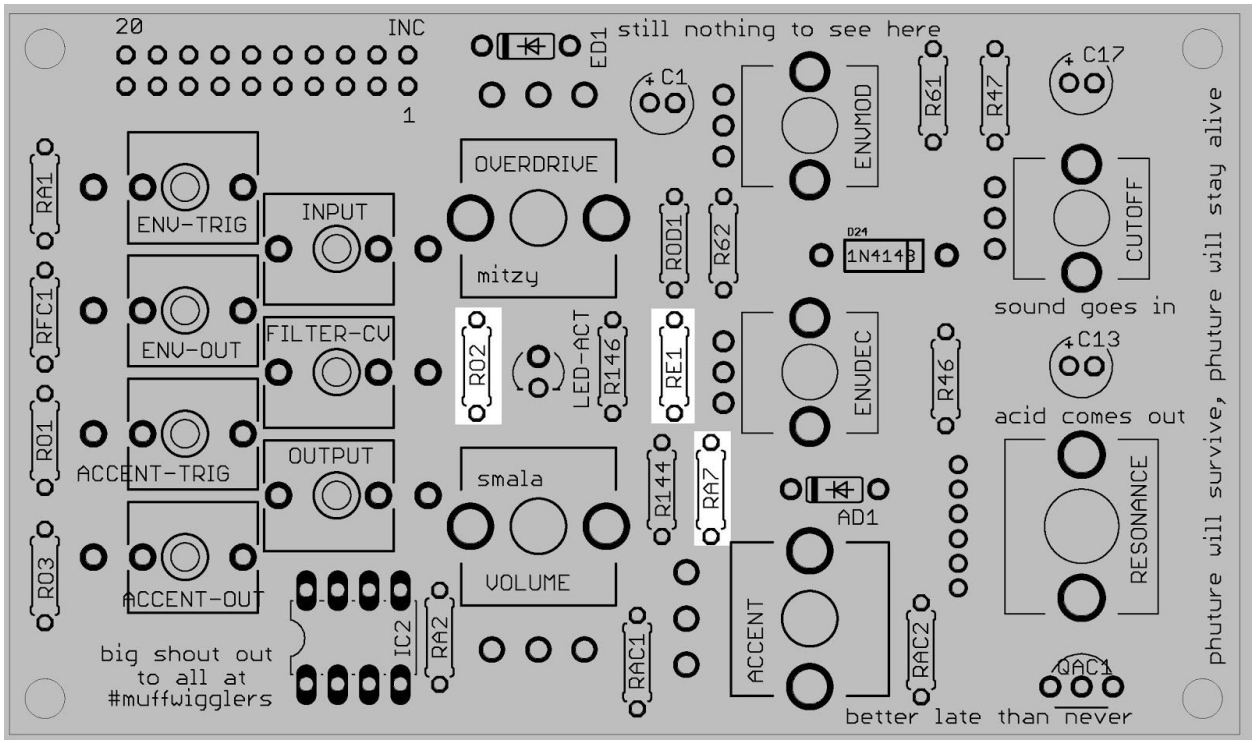
The first thing to do is to trim the DUAL pot (that's the one with 6 pins that came with the partial kit) down to the size we need. The easiest way to do this is with a dremel like tool, hold the shaft (not the pot body!) and just cut the top of the shaft off where the knurls end. If you don't have such a tool you could try cutting with a sharp blade or cutter, just make sure you hold the shaft and not the pot body when you do.

USE EYE PROTECTION IF YOU ARE USING POWER TOOLS, NO EXCEPTIONS, EVER! (can't stress that enough really, even if it is common sense)

here’s what it should look like after trimming (ok there’s four here since we stole the picture from the modseq build manual but it’s exactly the same procedure here. For this build you only need one of course)

Ok let’s start with the TOP pcb (this is the control pcb with the potentiometers and jacks on it)

the best way to go about things here are to place all the low level parts first then build up to the larger hardware.



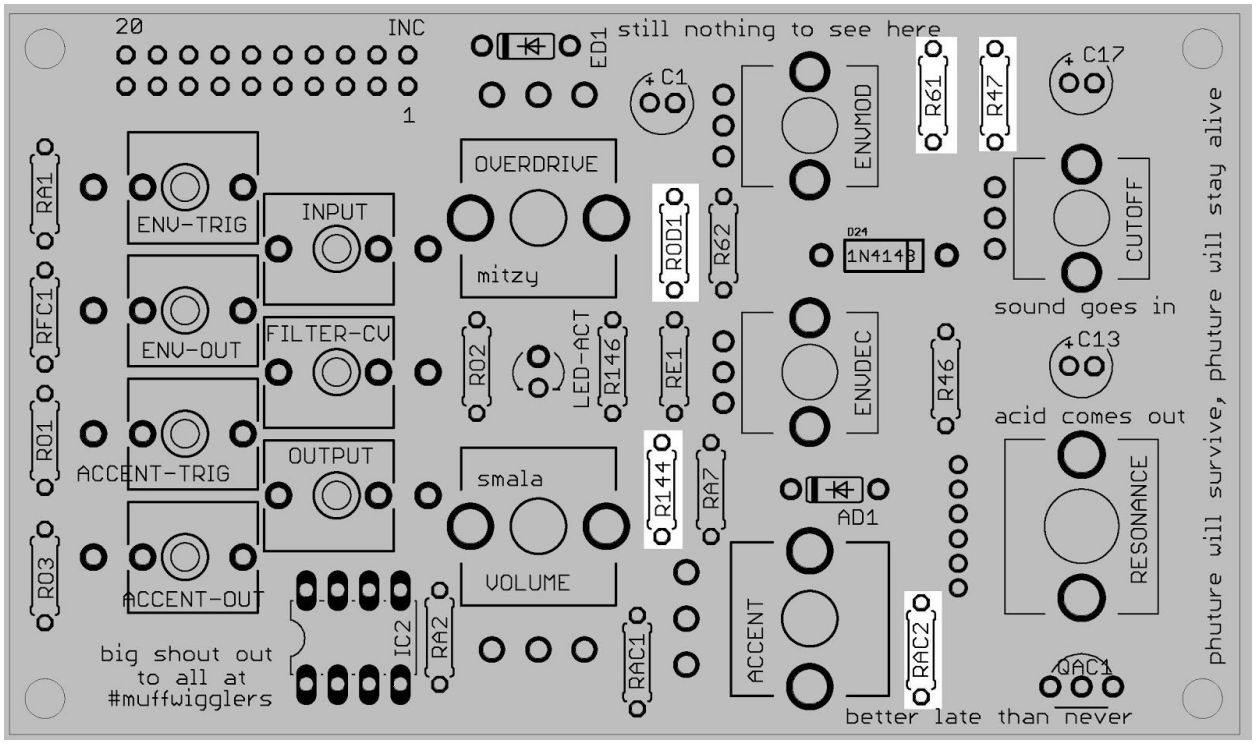

we will therefore start with the resistors. Grab your TOP pcb and place three 1K resistors in the places shown above.

Make sure they are snug to the board, it's not super important but it certainly looks nicer.

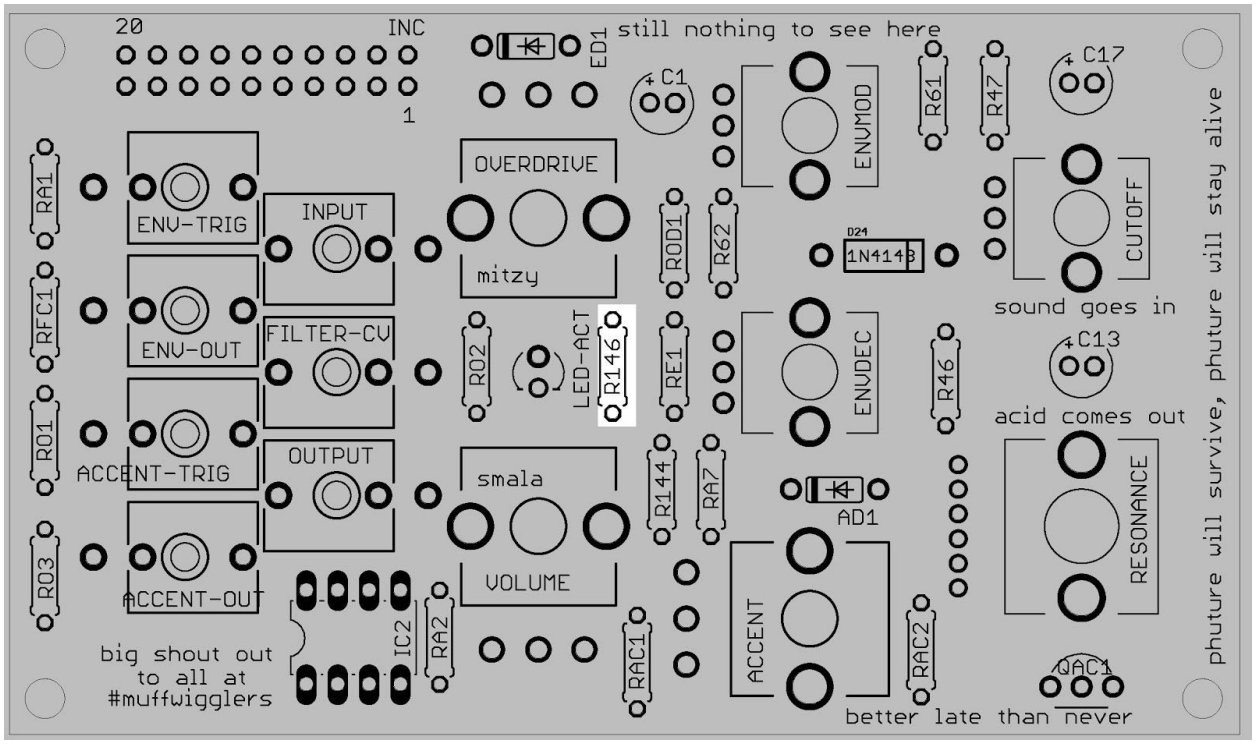
Right so solder those three in and clip the legs, that's it for the first step.

For the most part this is how we shall proceed in the build. You will see a diagram like above, you place the part, flip the board, solder and clip legs and move to the next step. It's that simple, if you take your time on each step and double check before you commit to solder and take a slow careful methodical approach then the build should be very smooth.

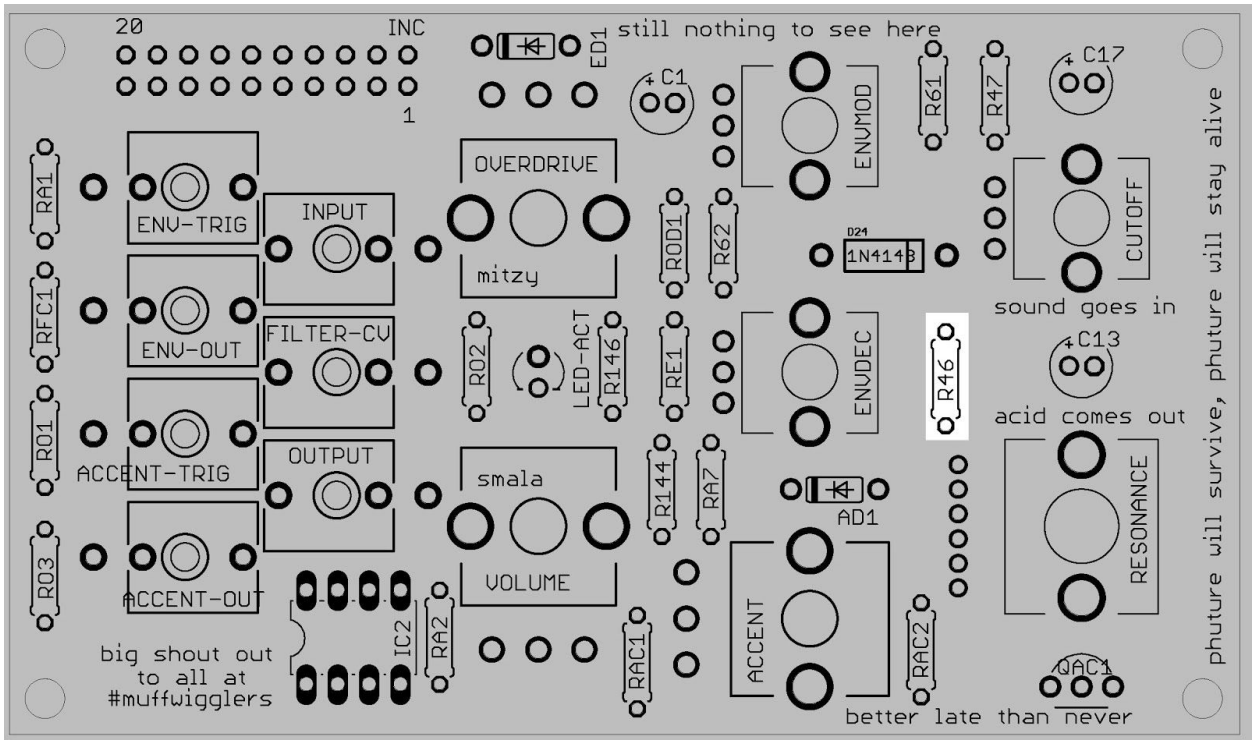
Ok without further ado, here are the next stages.



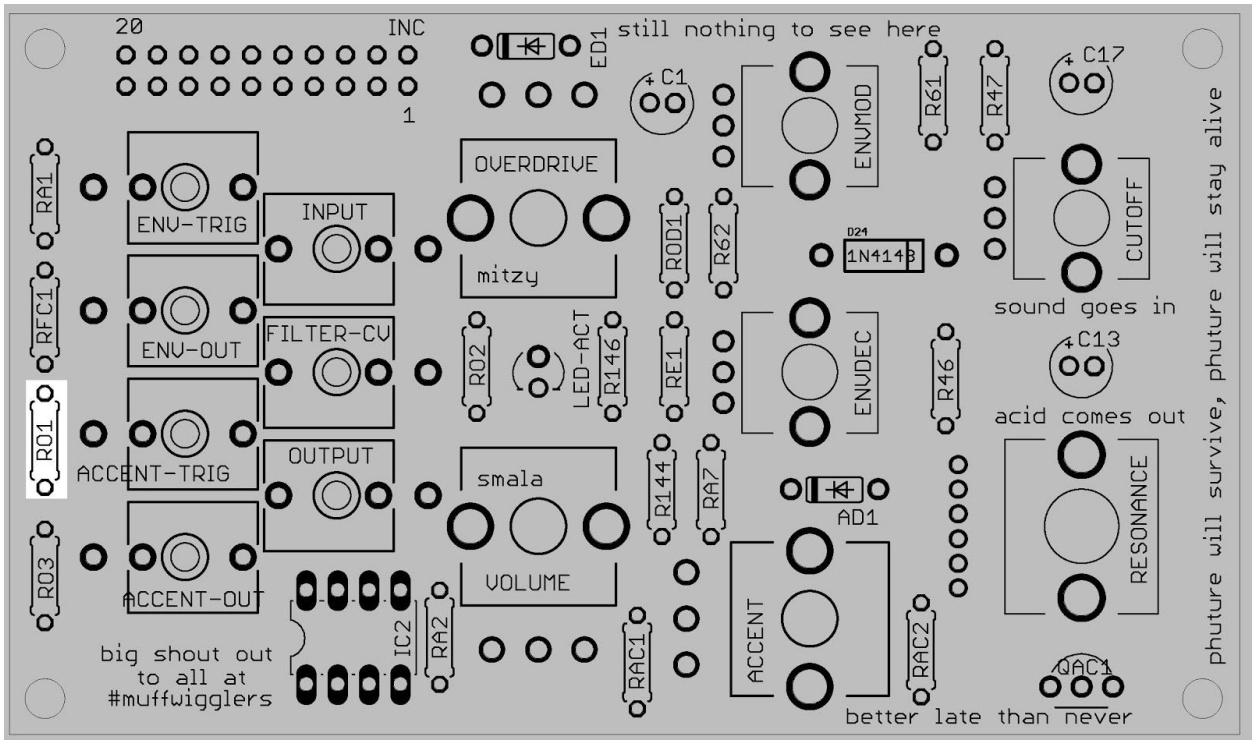

Place and solder five 10K resistors.



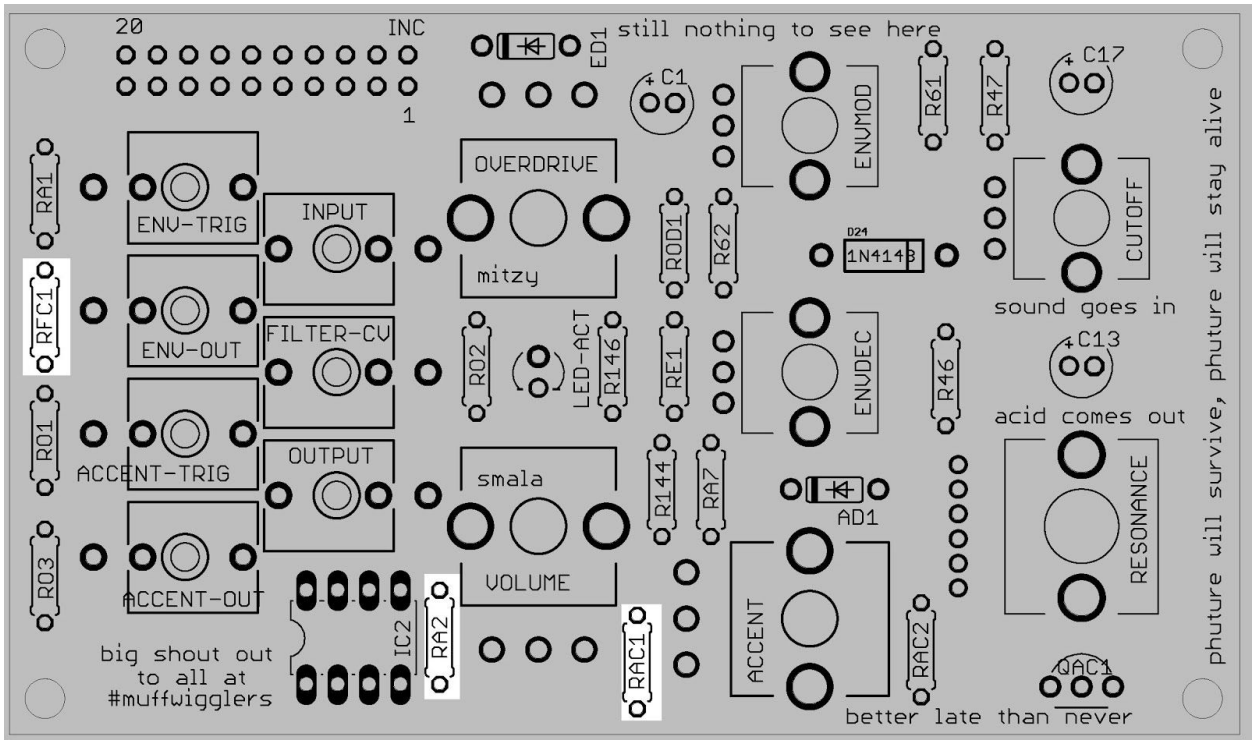

Place and solder one 22K resistor.



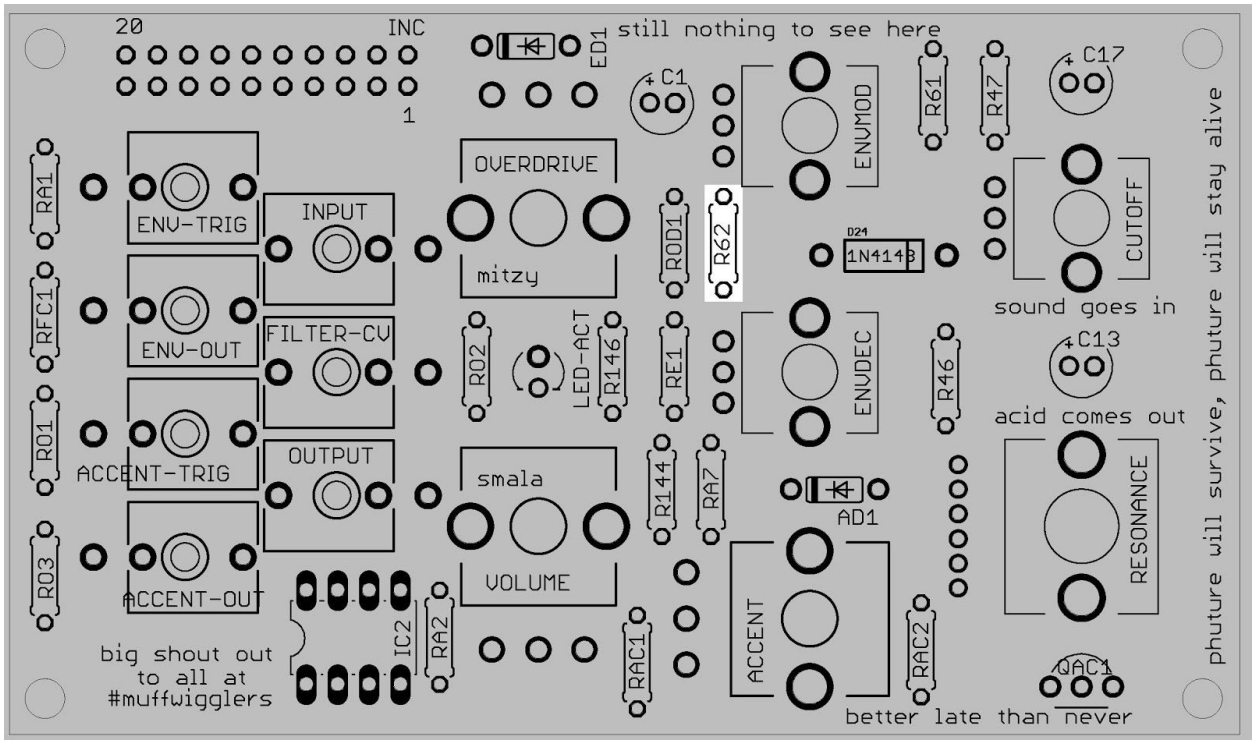

Place and solder one 47K resistor.



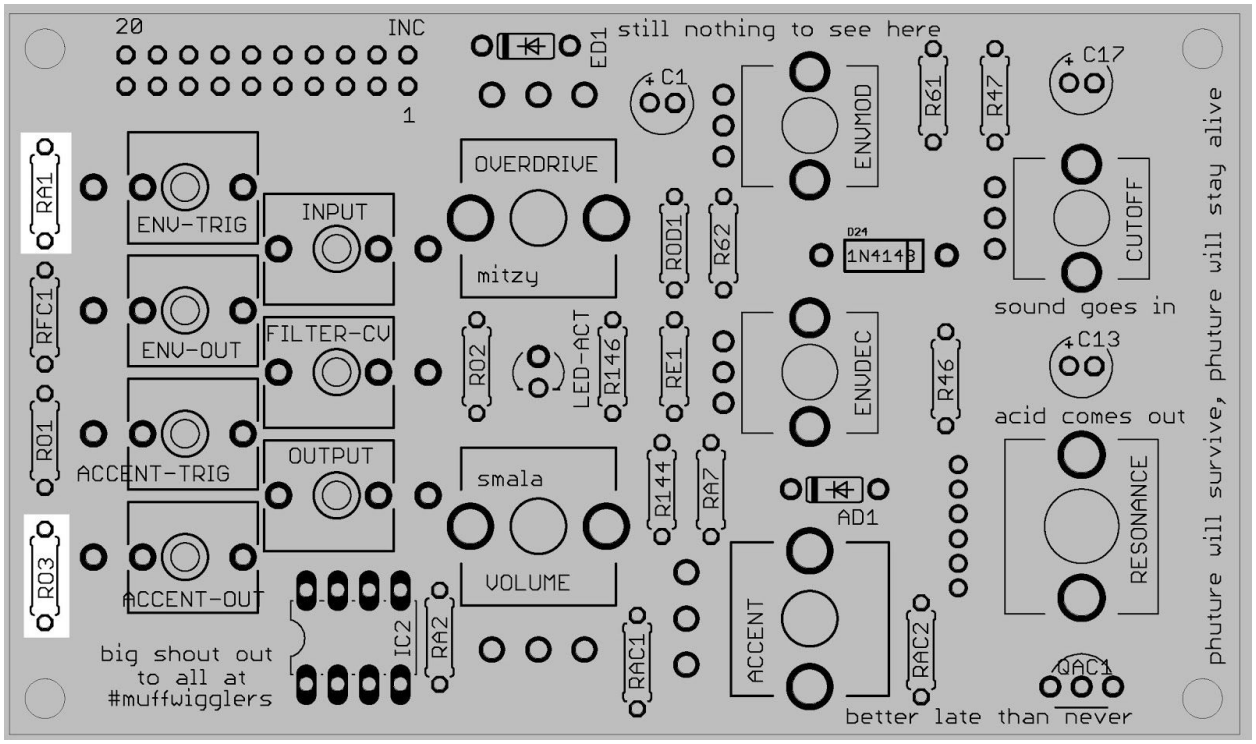

Place and solder one 56K resistor.



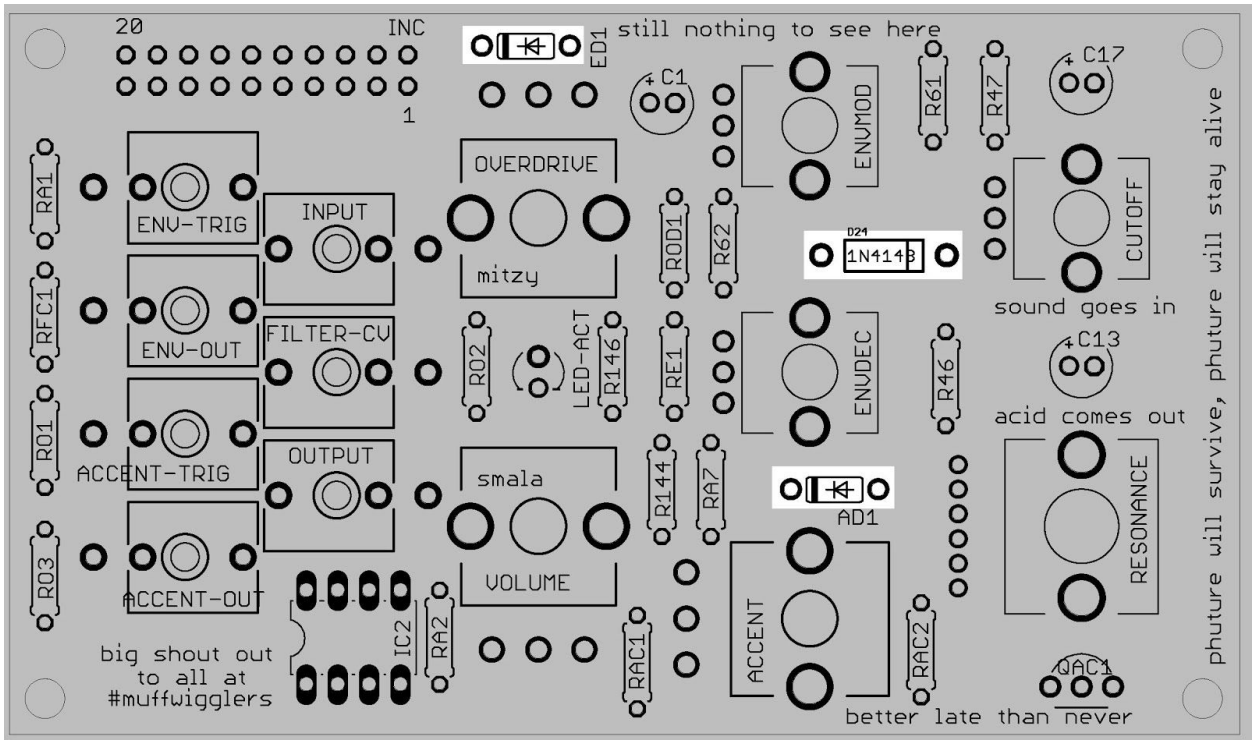

place and solder three 100K resistors.




place and solder one 220K resistor.

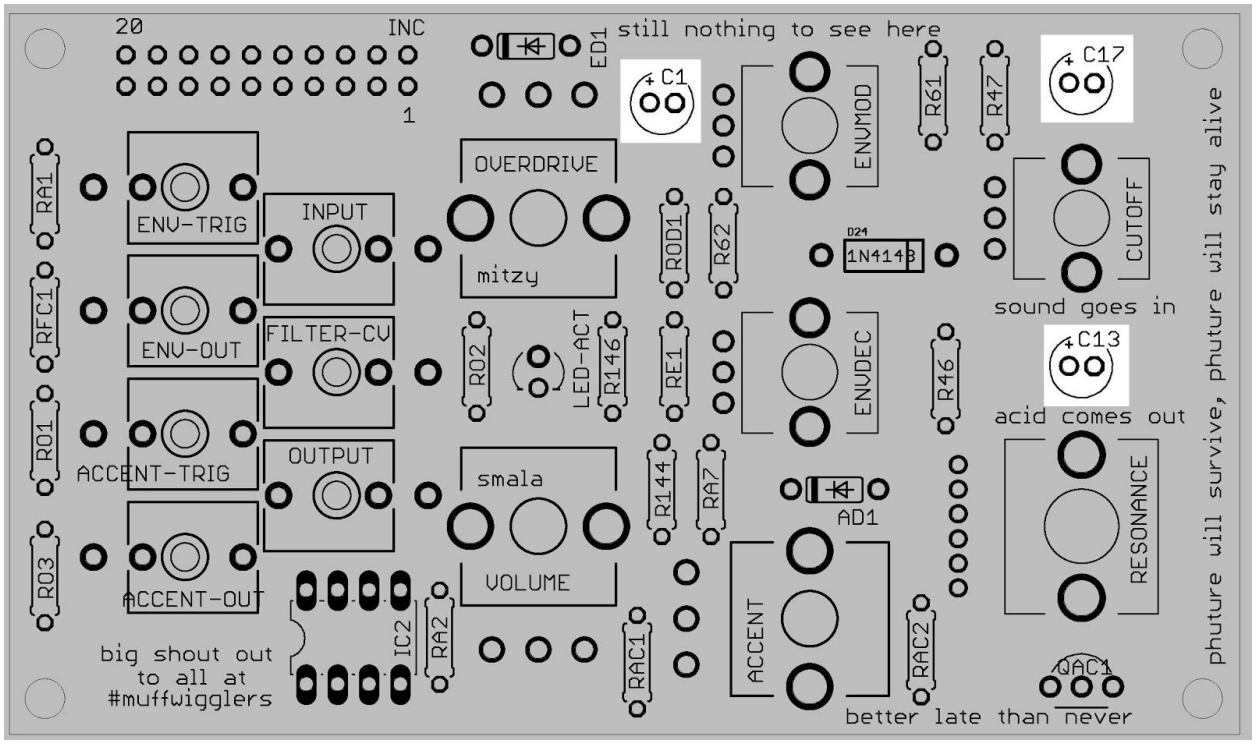



place and solder two 1M resistors.



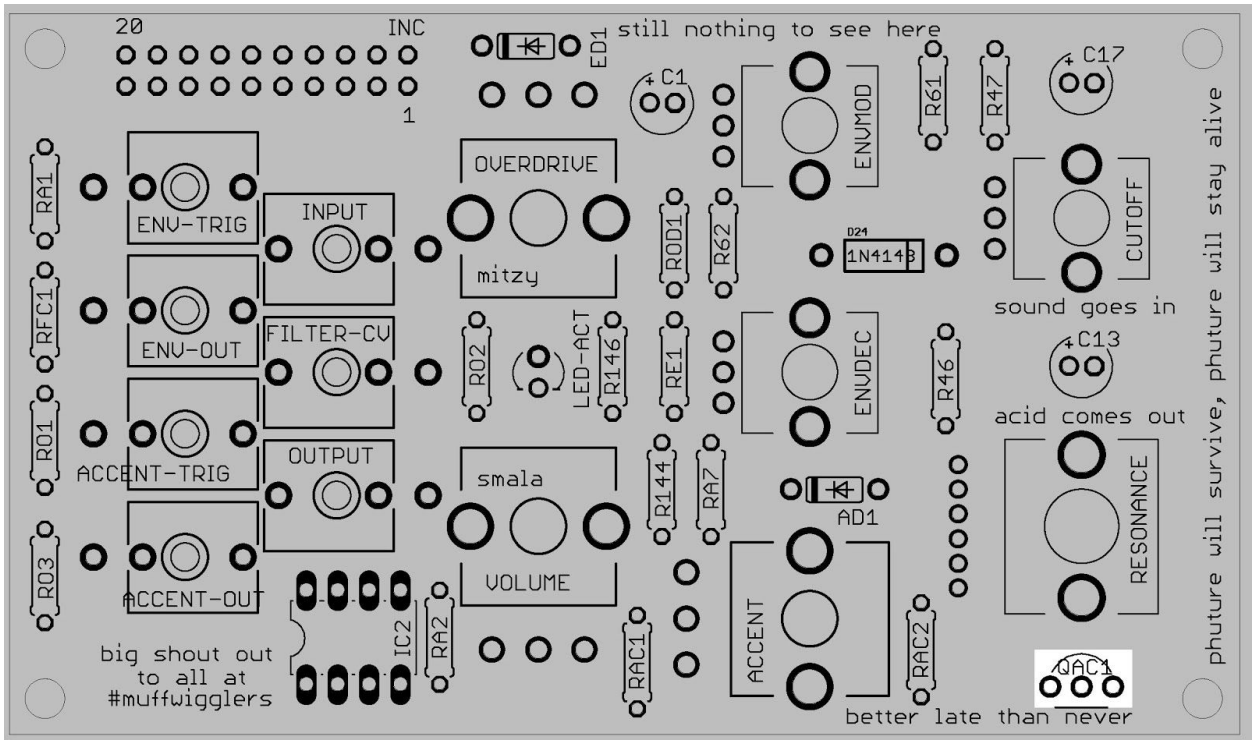

place the 1N4148 diodes, be sure to align the band on the part the same as the silkscreen.

The silkscreen here has a small “error”, where two different symbols were used by accident, but the ones shown are all 1N4148.

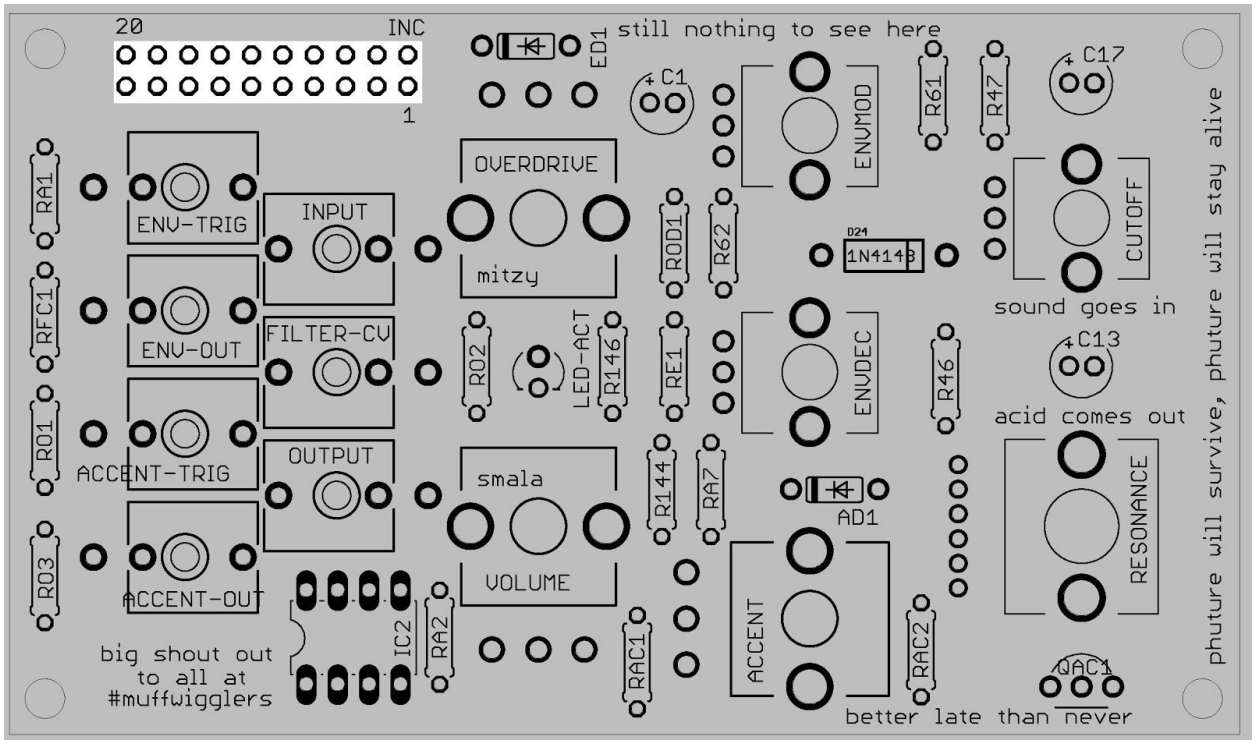



place and solder the “mini” 1uf Electrolytic caps,  these parts are polarised. Make sure the long leg of the capacitor goes into the hole marked with a plus sign.

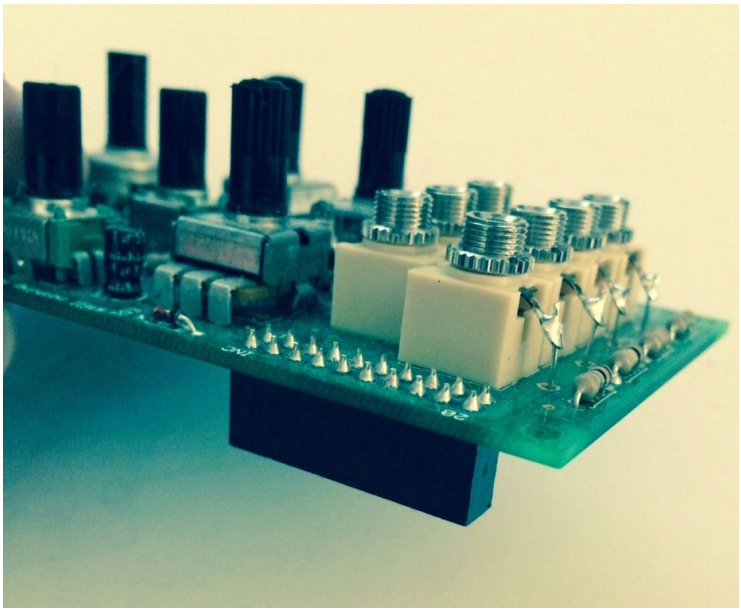
The shorter leg (also usually marked on the side of the can is negative).

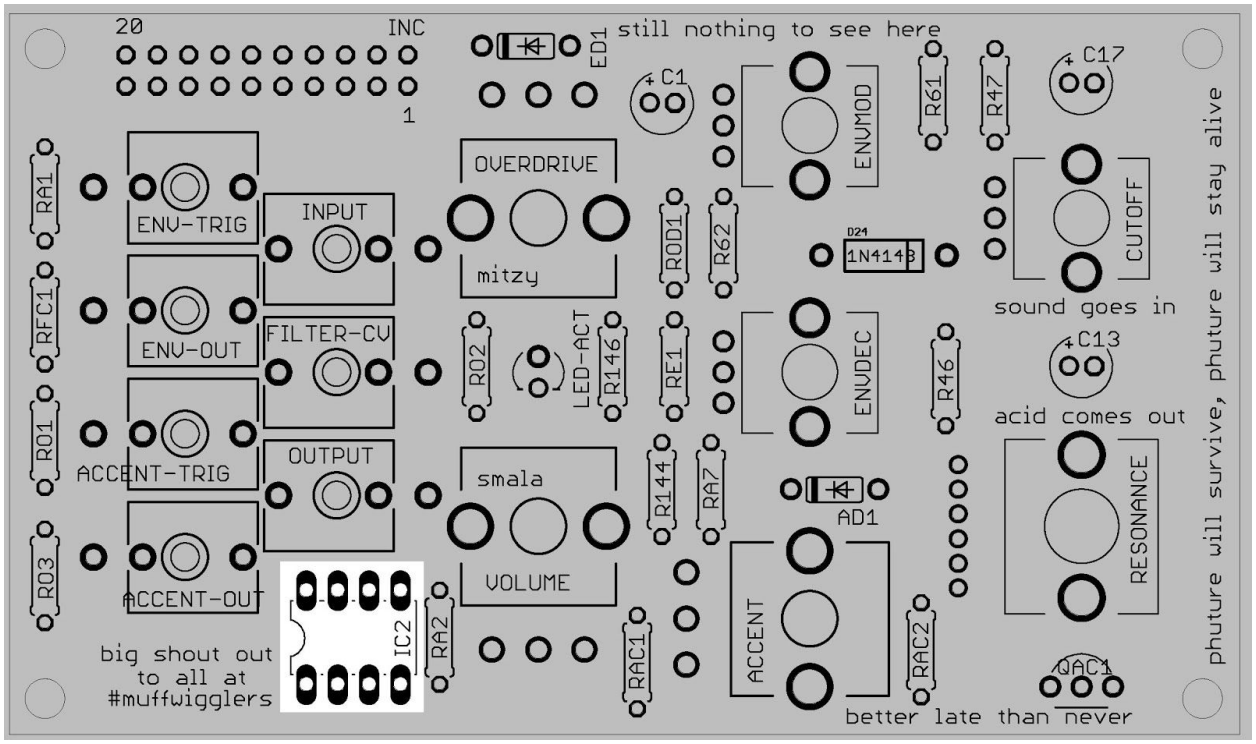



place one 2SC945P and solder,  it's important not to push the part all the way to the pcb as it is heat sensitive to soldering. Leaving longer legs will help alleviate that, however be sure that the top of the transistor isn’t also too high so as to prevent you placing the front panel on the completed module.



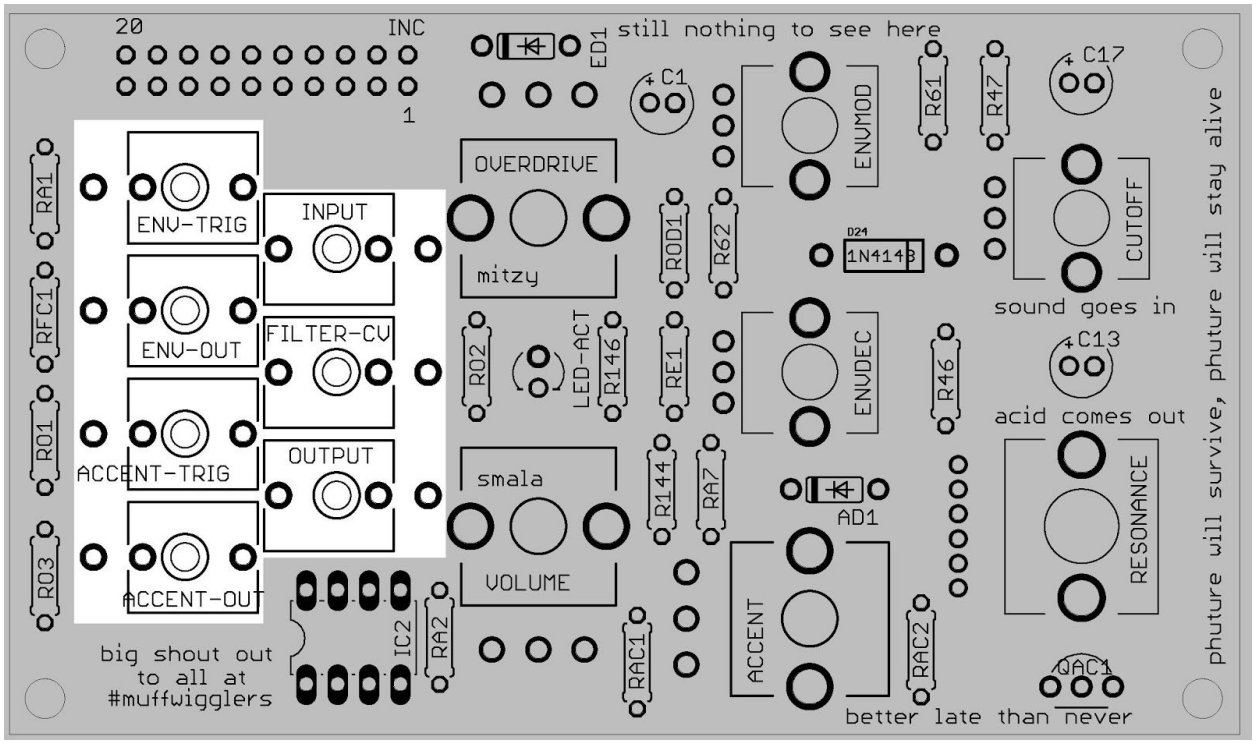

place the female 20 pin header, this is soldered on the other side! Check the reference picture below for the correct orientation before soldering (please double check this as it will be next to impossible without a solder station to remove this part once it is in place)



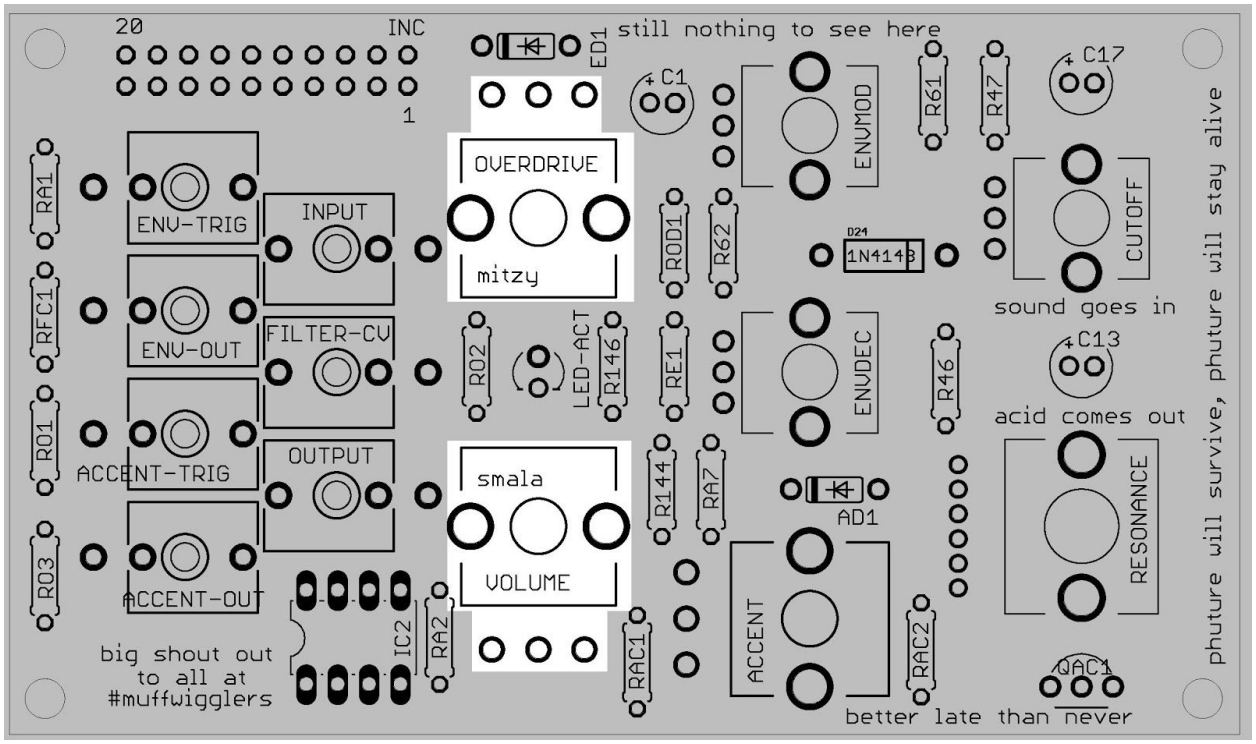



if you have an 8PIN IC socket place it now, if not place and solder your TL082

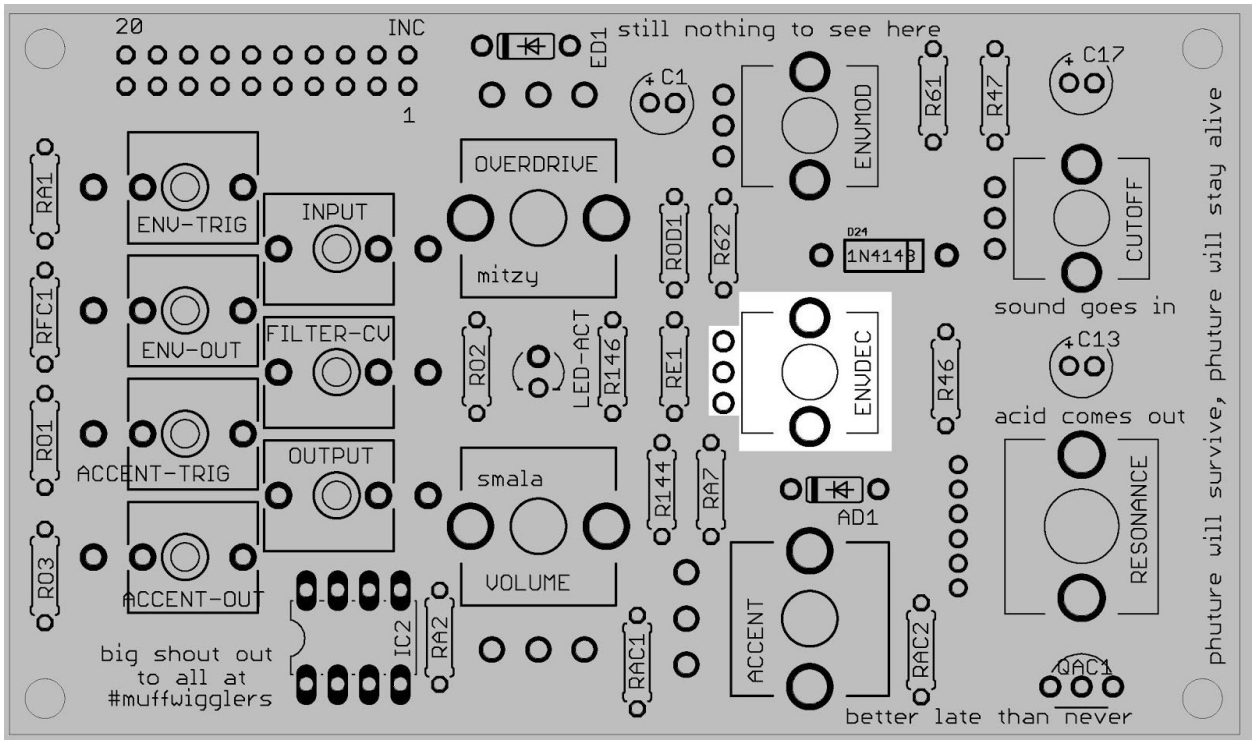
Be sure to align the socket/chip with the indented notch on the silkscreen as this part must be placed correctly and **will fail if reversed**.




place and solder the jacks.

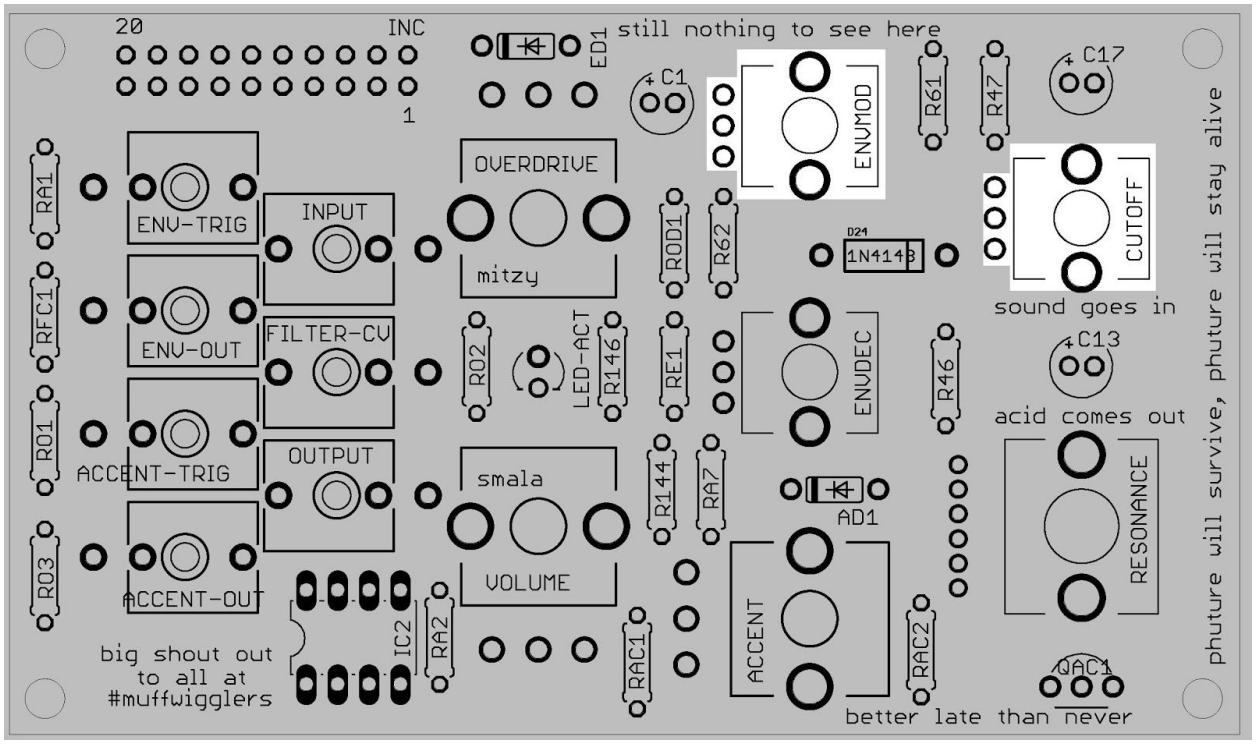



place and solder the 1M LIN pots.  p.s. Mitzy and Smala are the names of my cats ;-)

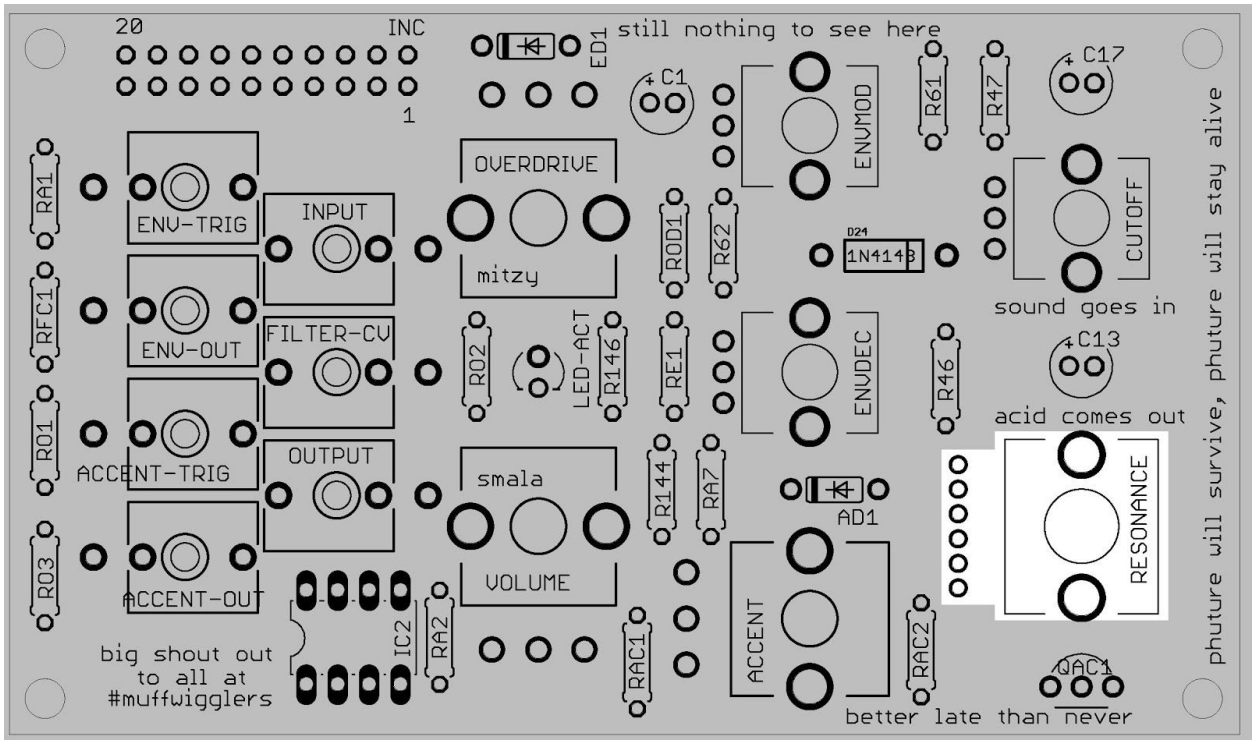



place and solder the 1M LOG pot.

place and solder the 50K LIN pot.




place and solder the 50K LOG pots.

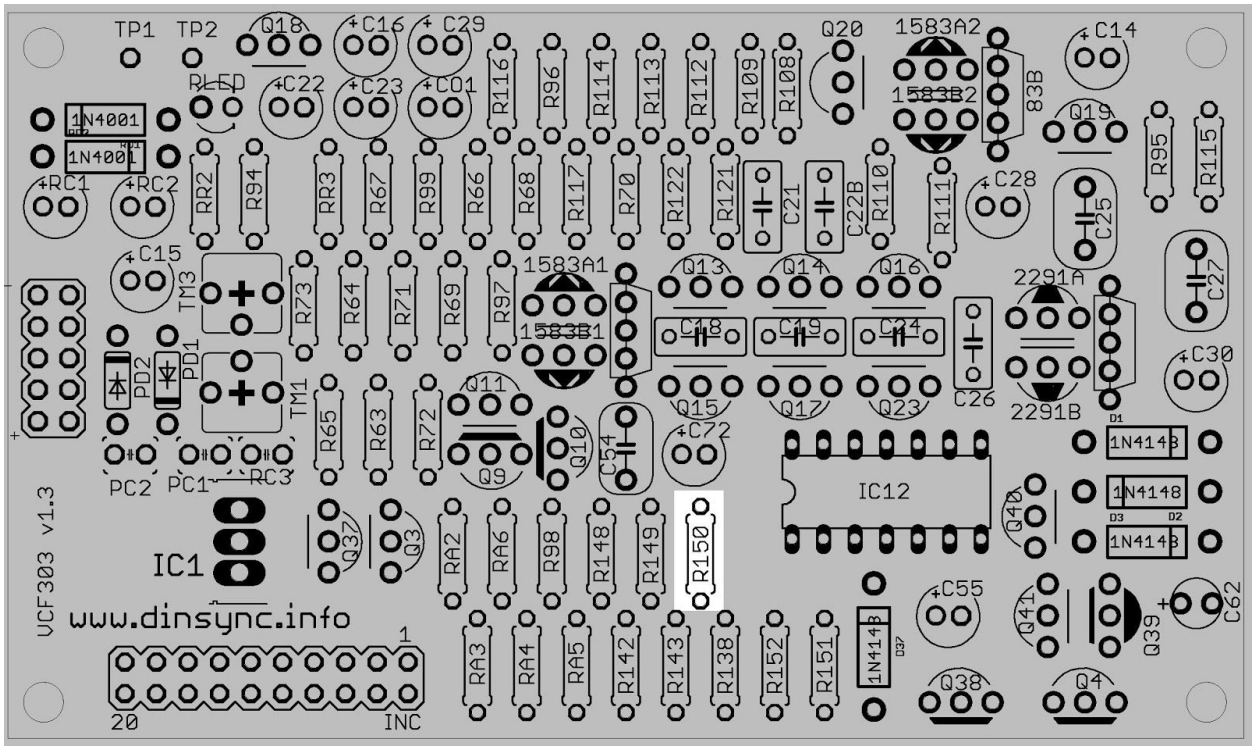



place and solder the 50K DUAL LIN pot that came with the kit. Take your time on this one and double check the pins for solder bridges as they are quite fine.

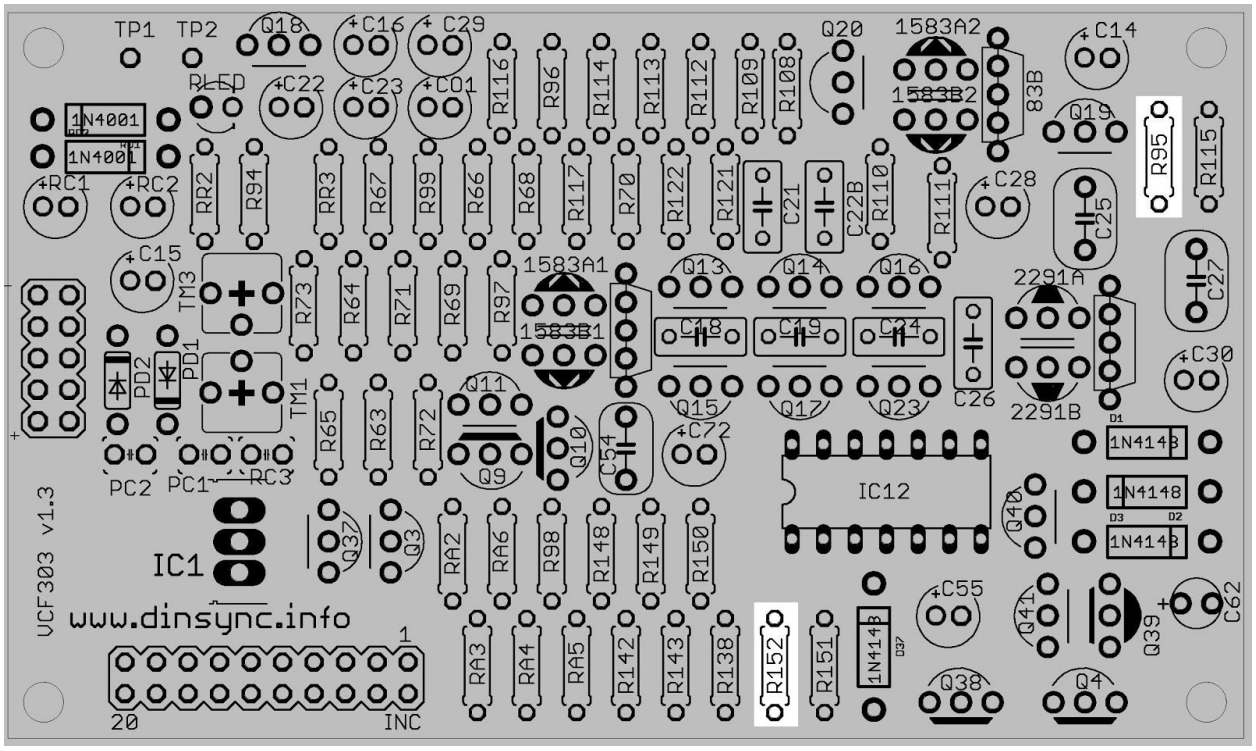
That's it for now, we didn't solder the LED in yet as we will do that at the final assembly stage. For now place your TOP board to one side and grab the BASE pcb.

Now before you start, how about a small break, clear the mind, stretch the legs walk the wife and kiss the dog, etc…

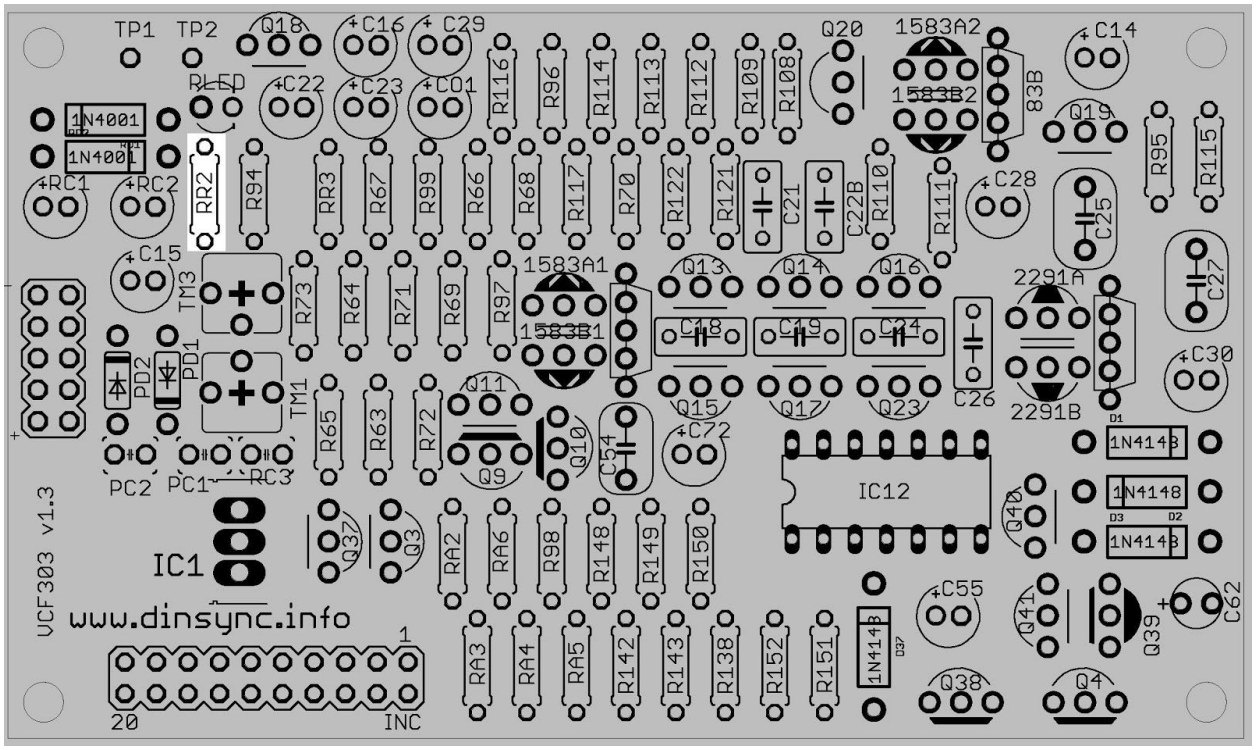
ok all rested? let's get on with the BASE pcb.



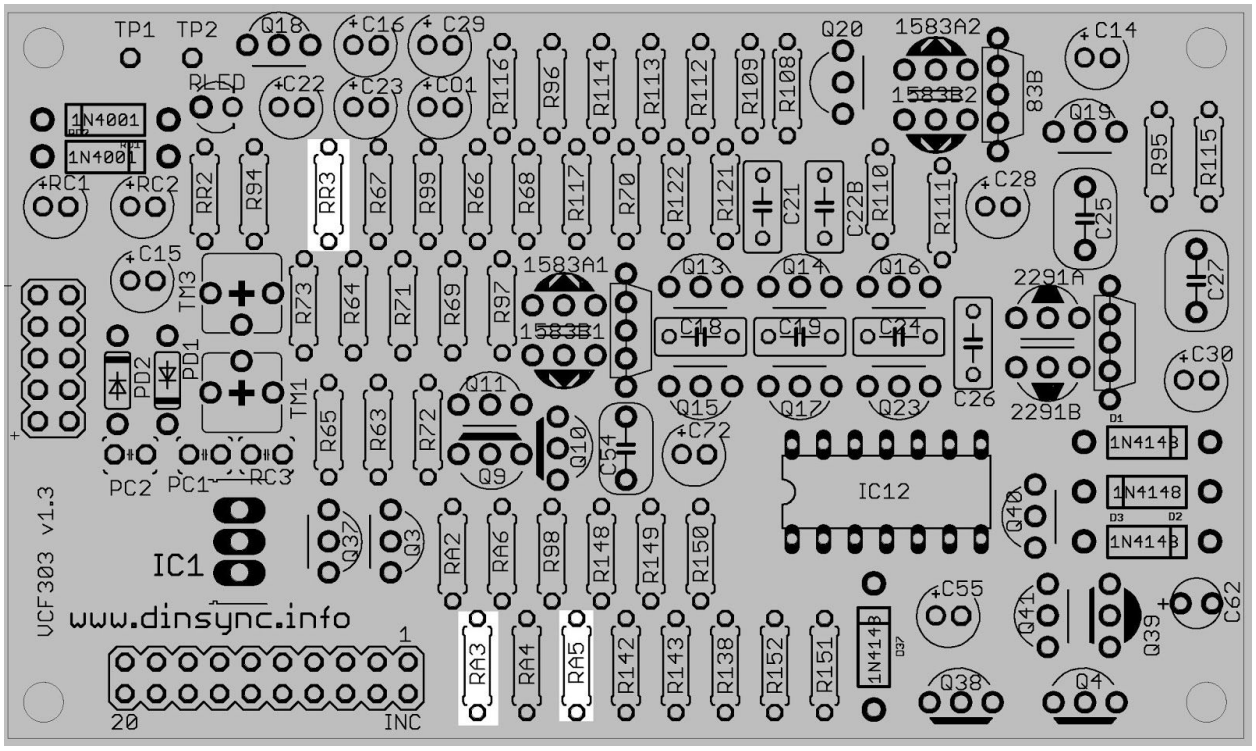

place and solder one 22 ohm resistor.



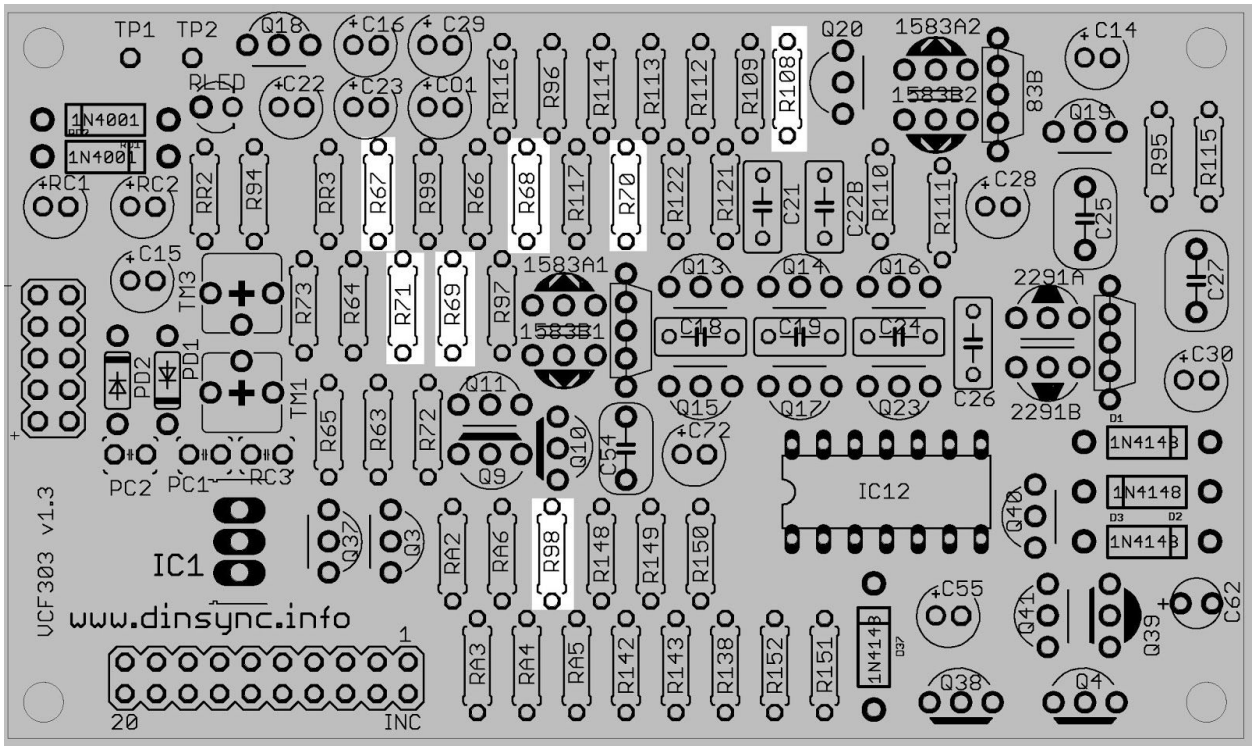

place and solder two 100 ohm resistors.



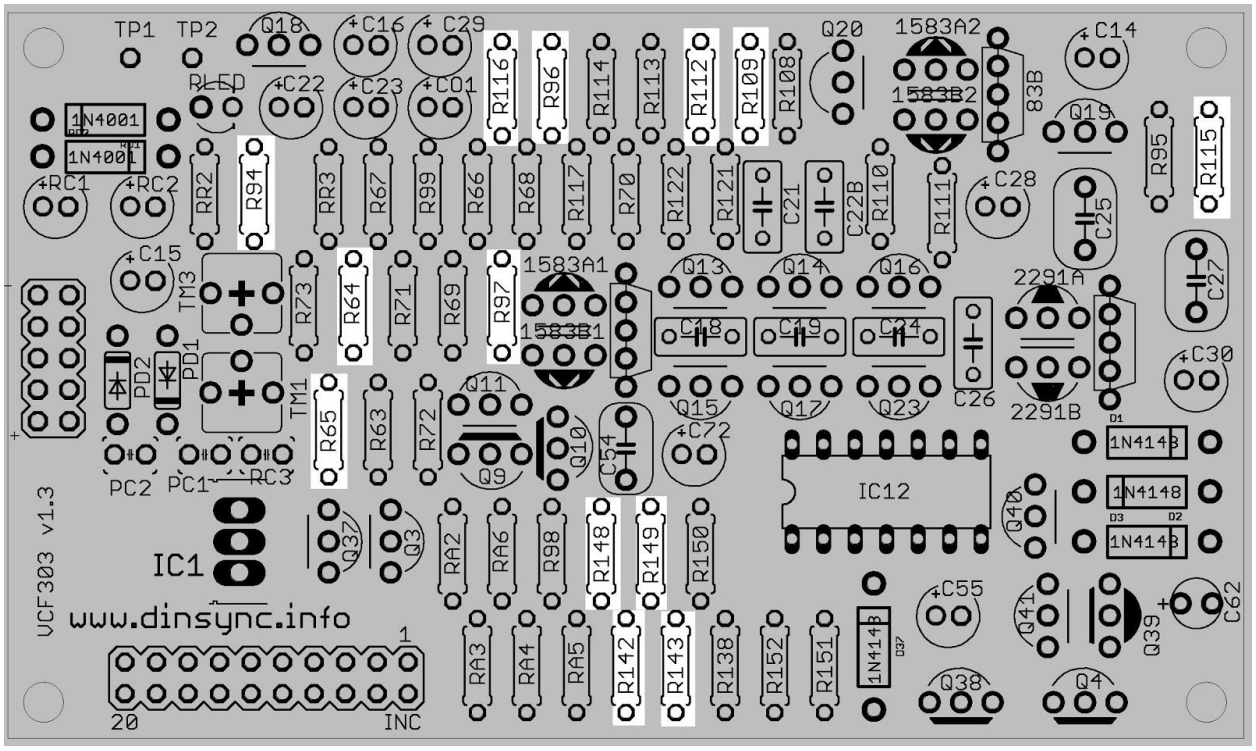

place and solder one 240 ohm resistor.



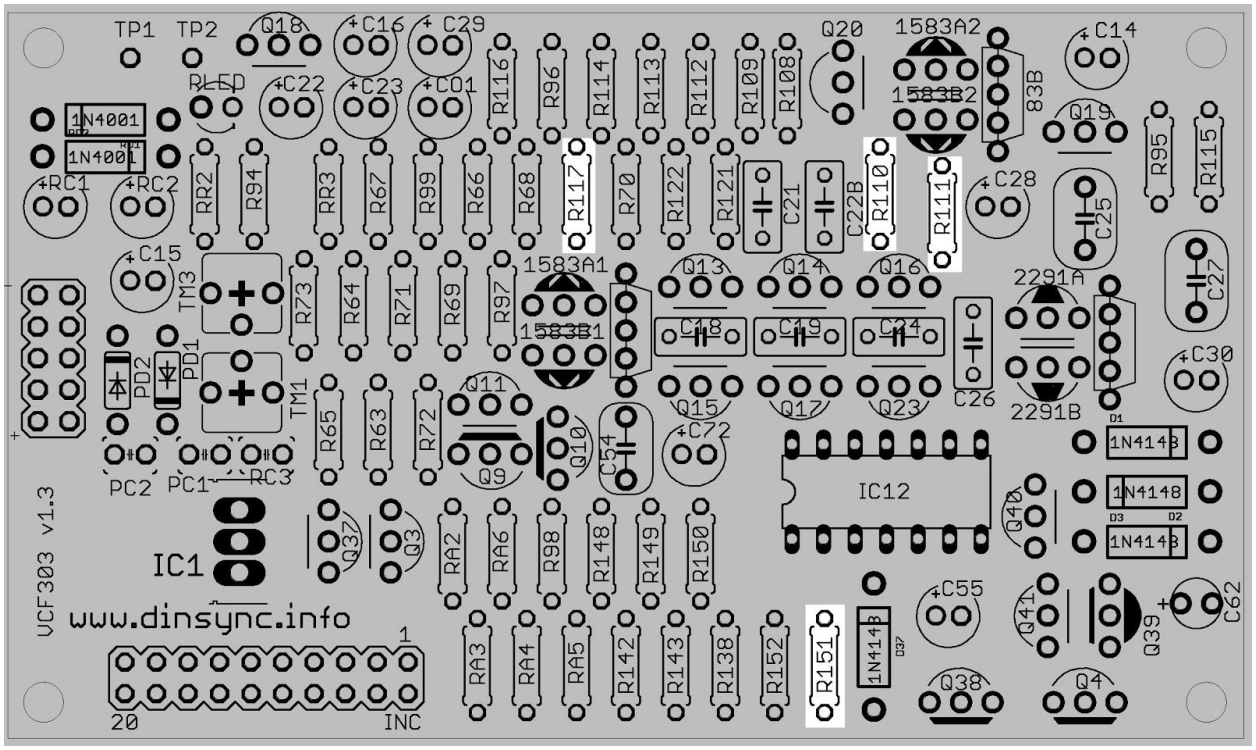

place and solder three 1K resistors.



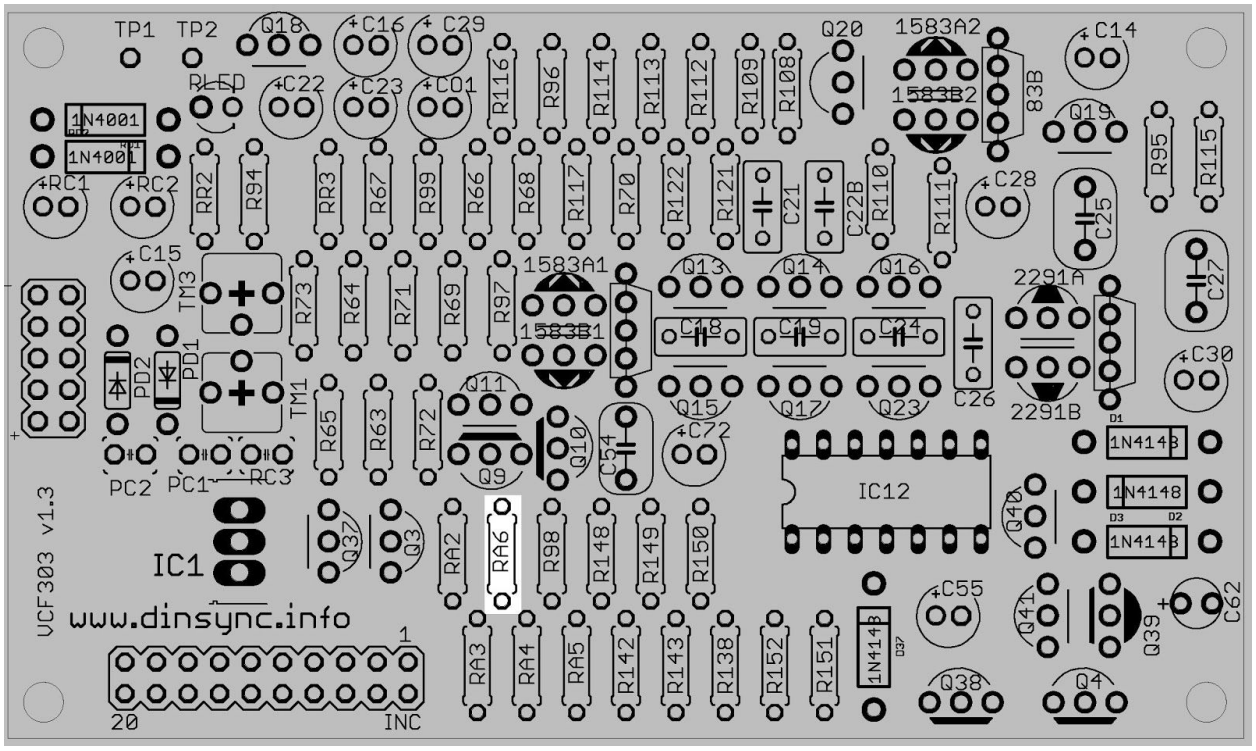

place and solder seven 2.2K resistors.



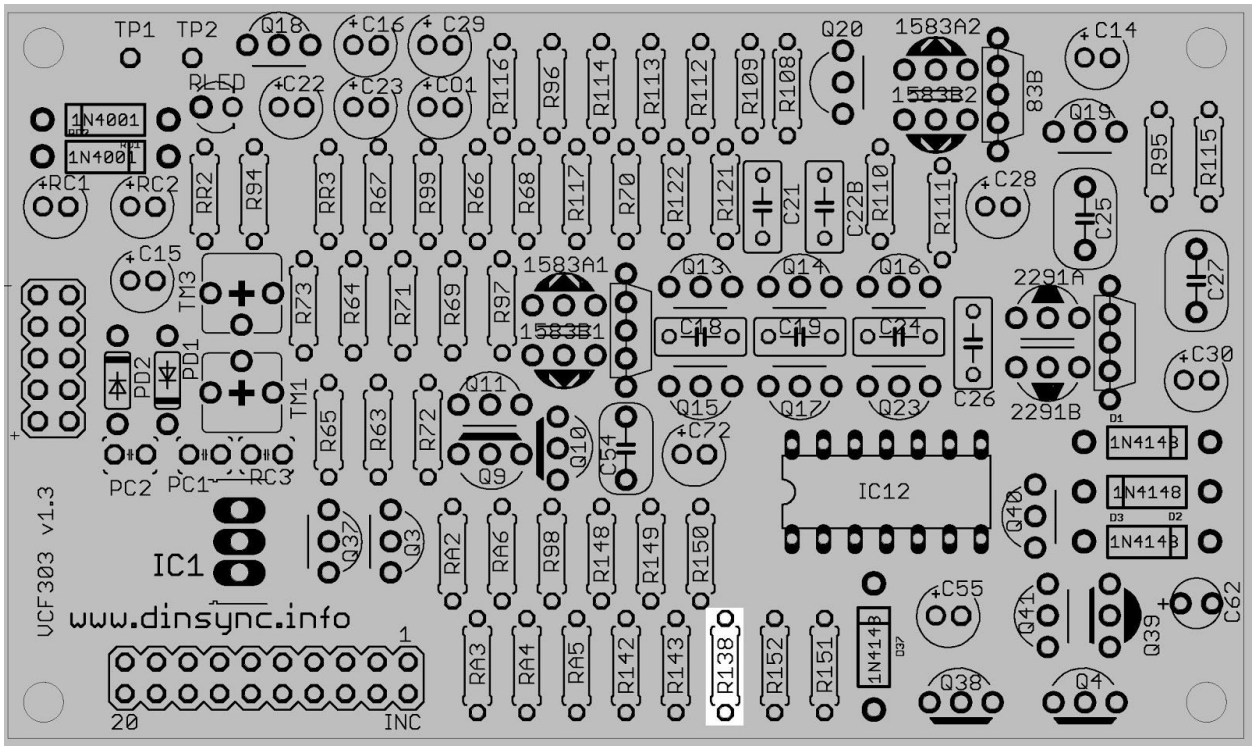

place and solder thirteen 10K resistors.



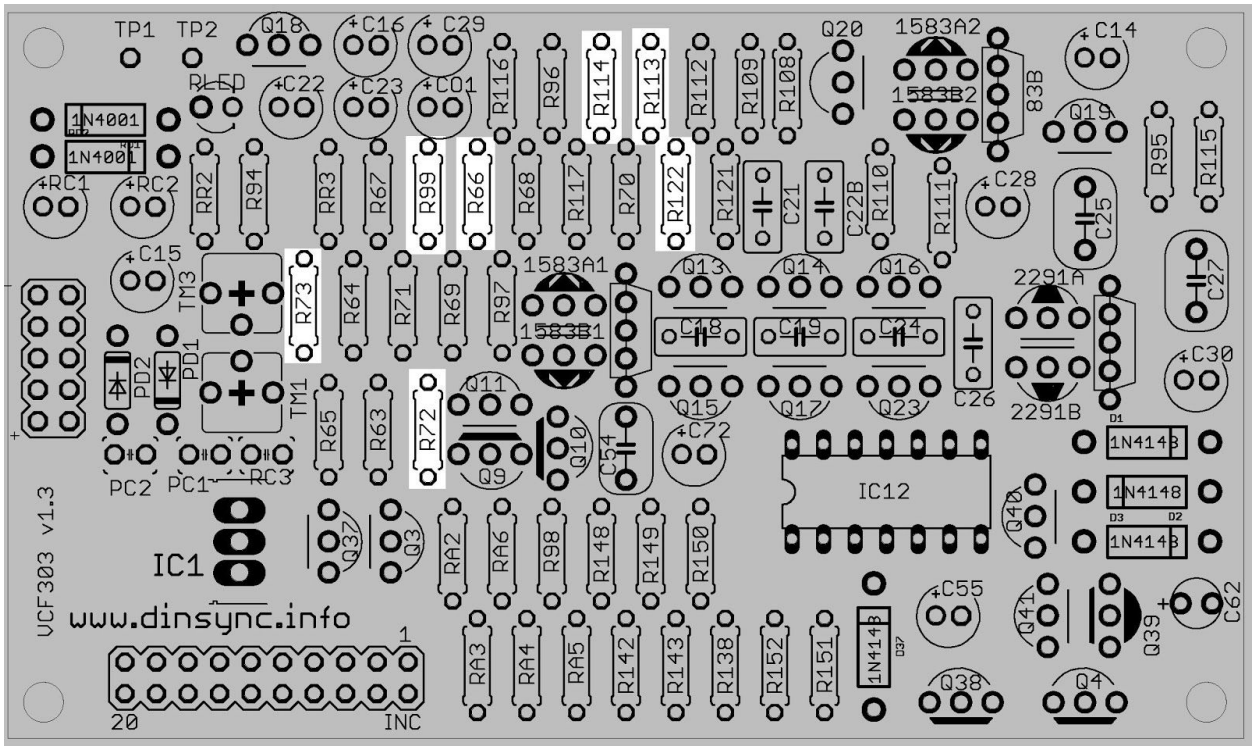

place and solder four 22K resistors.



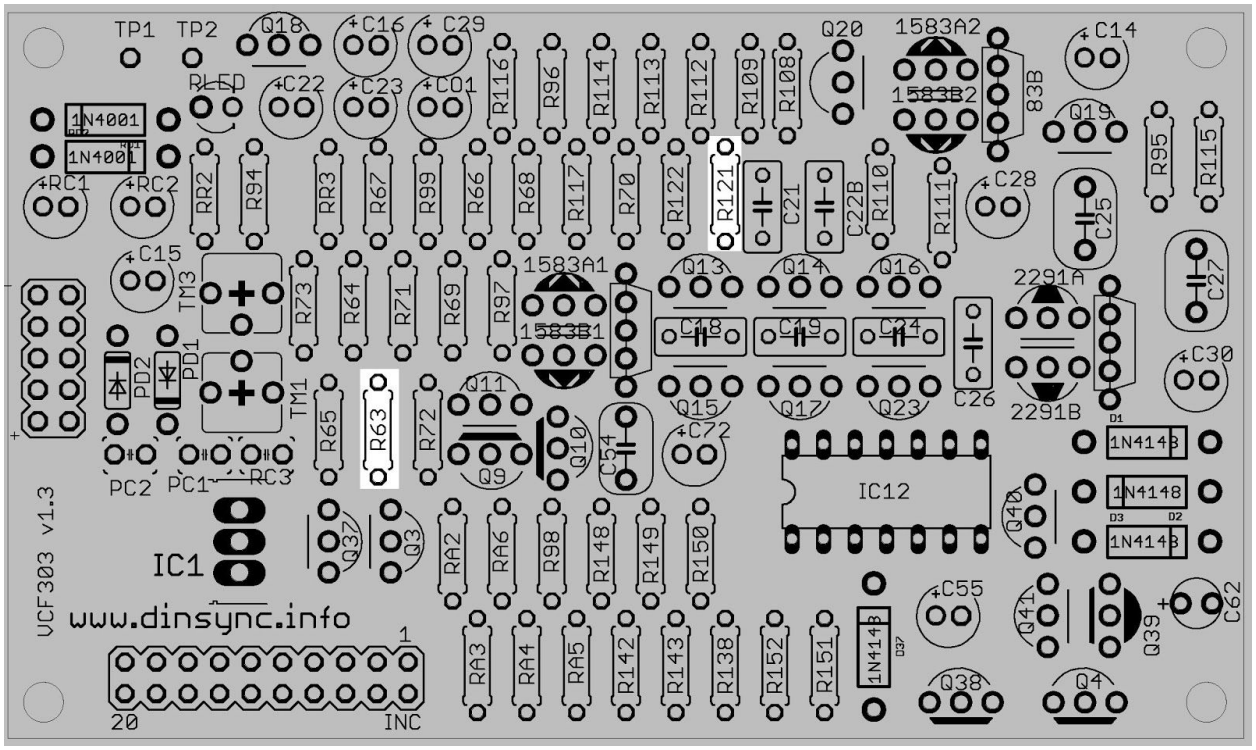

place and solder one 47K resistor.



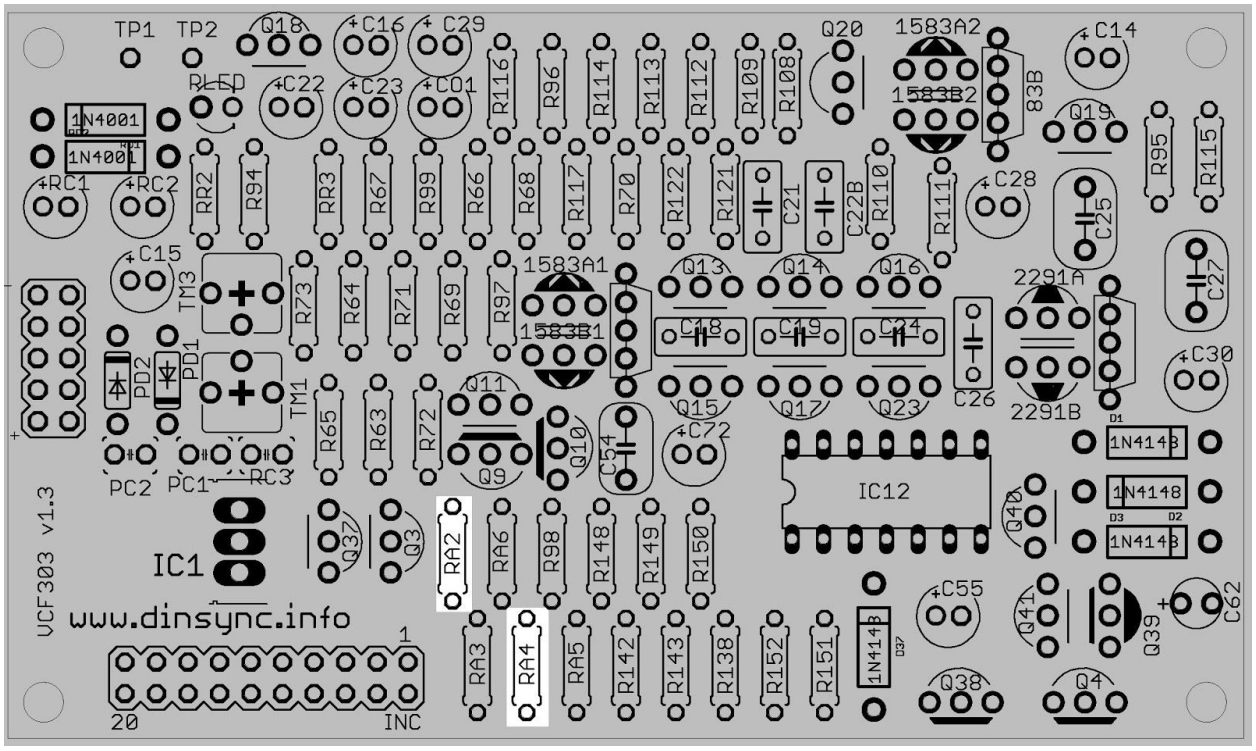

place and solder one 68K resistor.



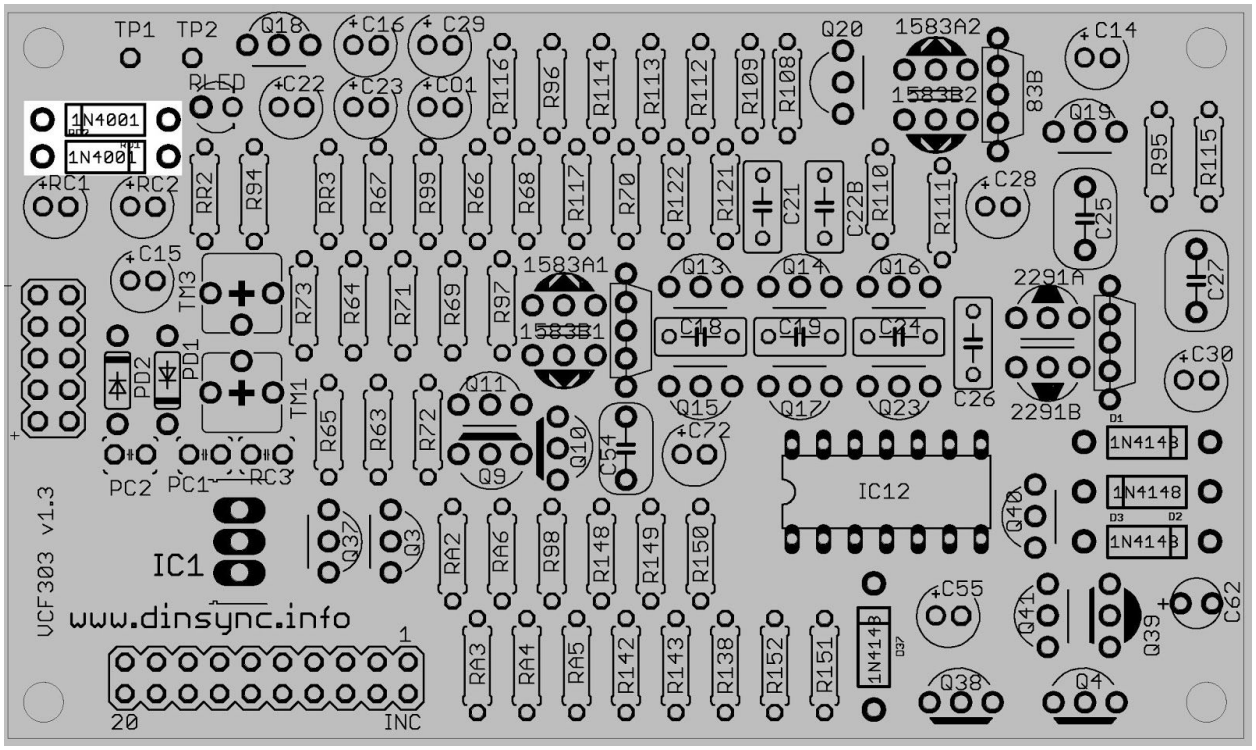

place and solder seven 100K resistors.



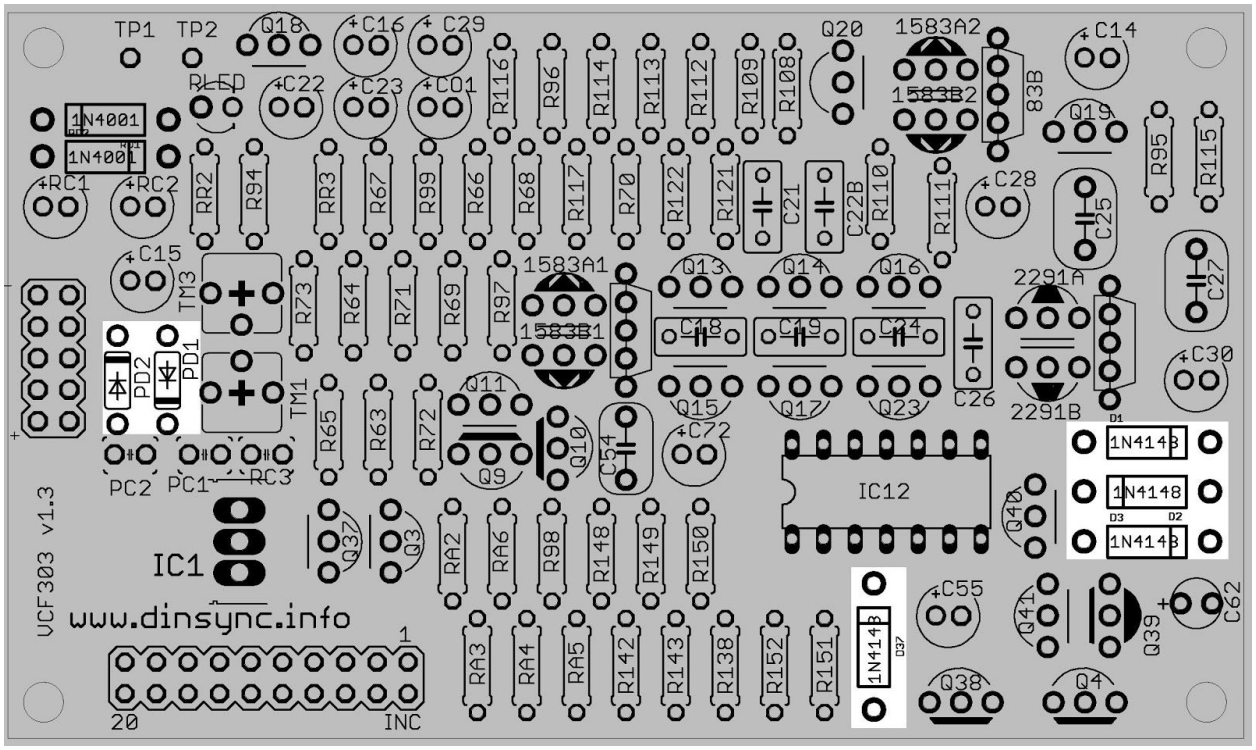

place and solder two 220K resistors.




place and solder two 1M resistors.

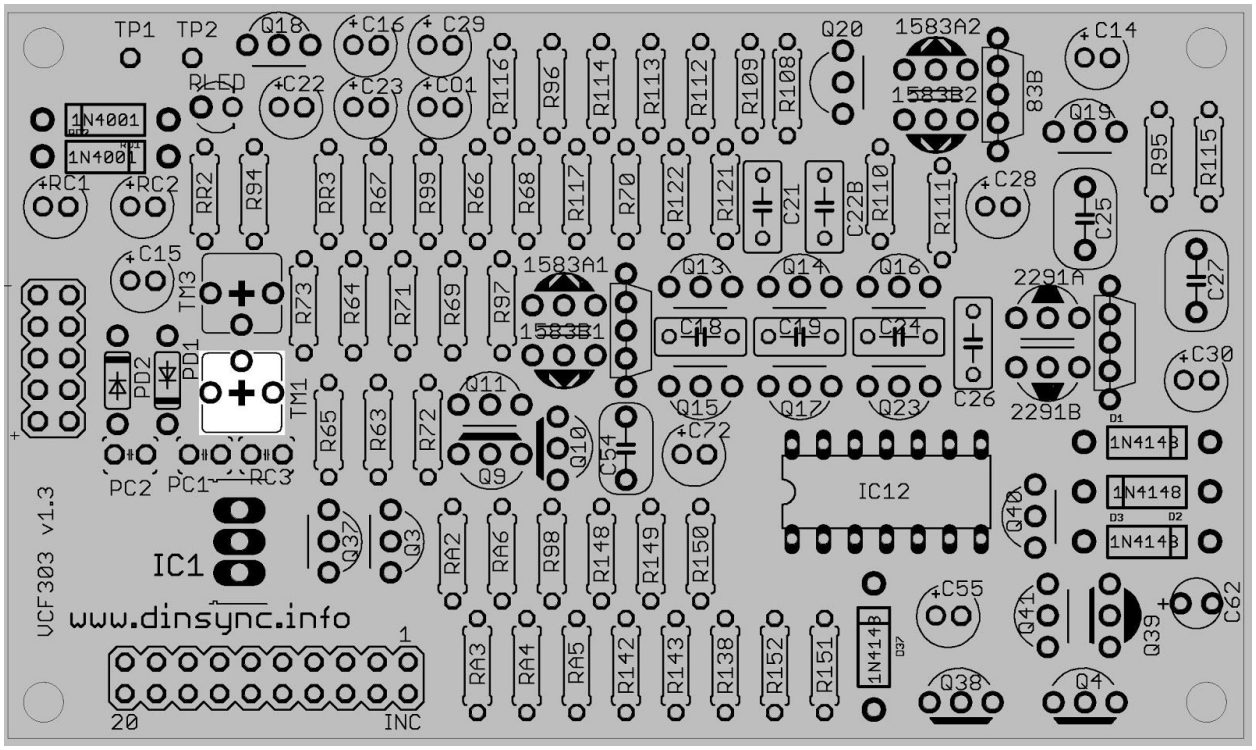



place and solder two 1N4001 diodes. Be sure to align the band to the silkscreen.

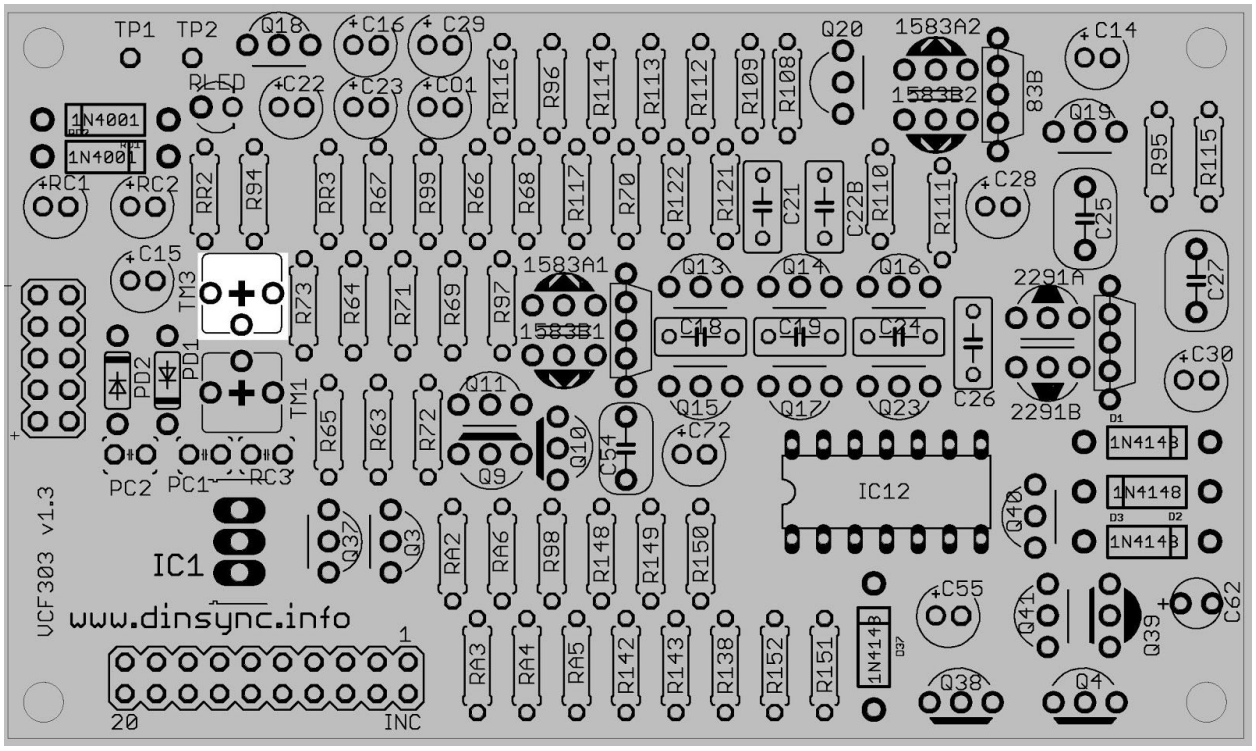



place the 1N4148 diodes, be sure to align the band on the part the same as the silkscreen.

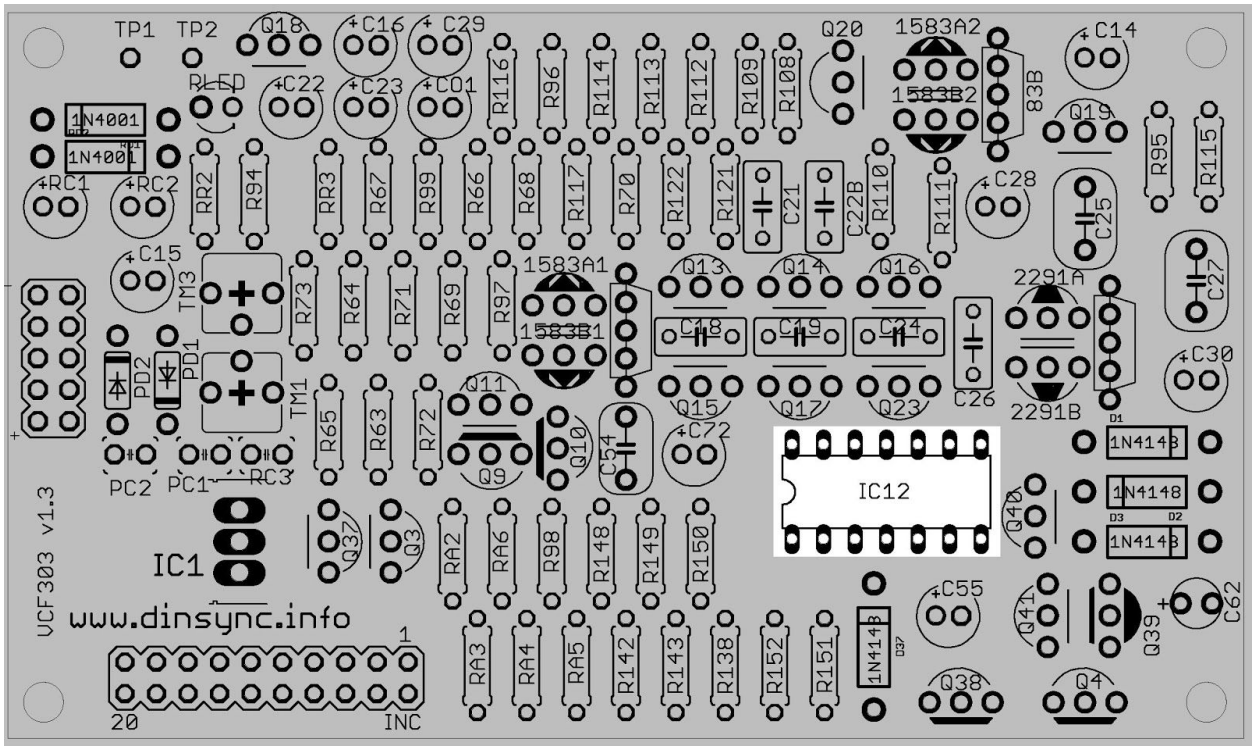
The silkscreen here has a small “error”, where two different symbols were used by accident, but the ones shown here are all 1N4148.



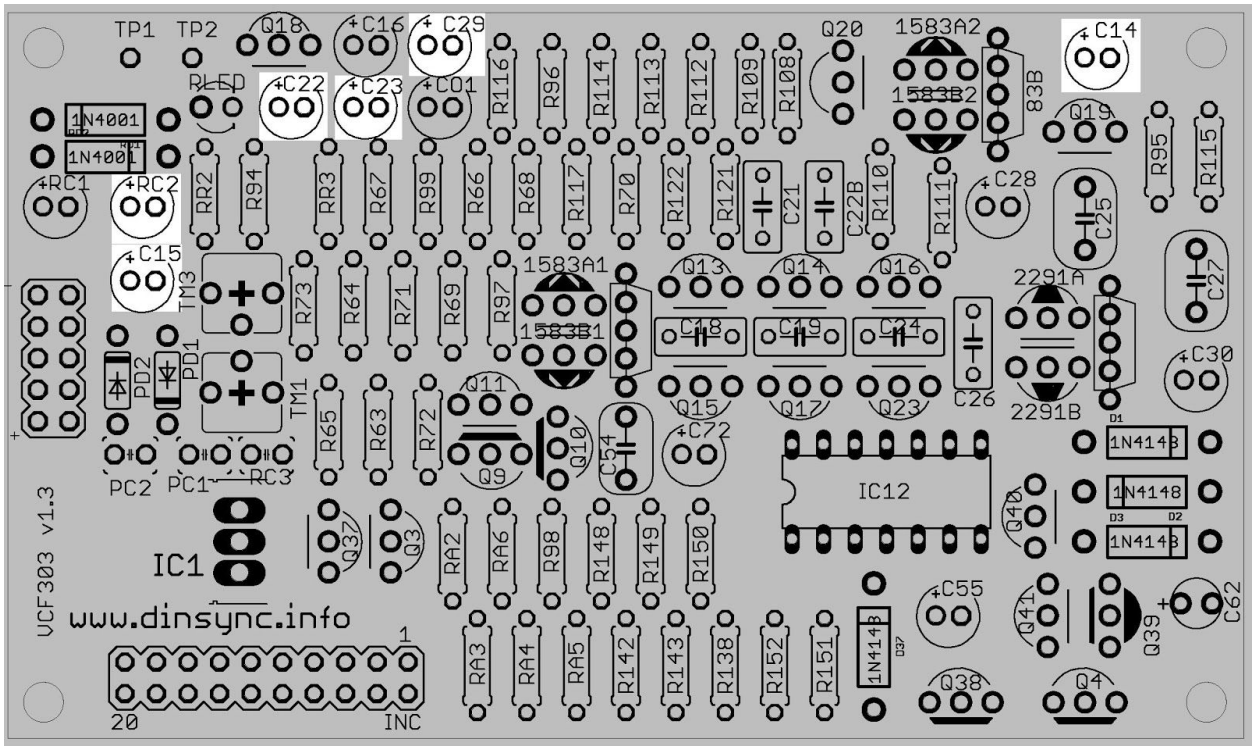

place and solder the 5K trimmer, check the position before soldering it in the wrong slot. ;)



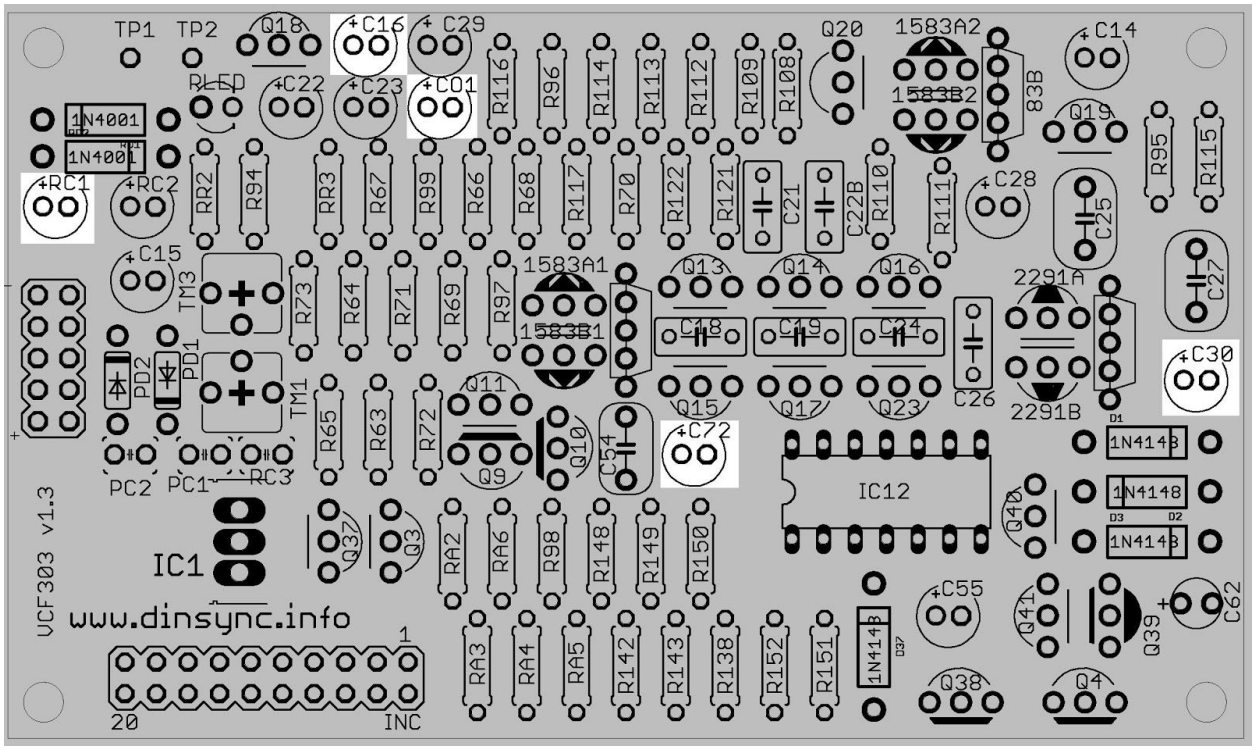

place and solder the 500K trimmer.



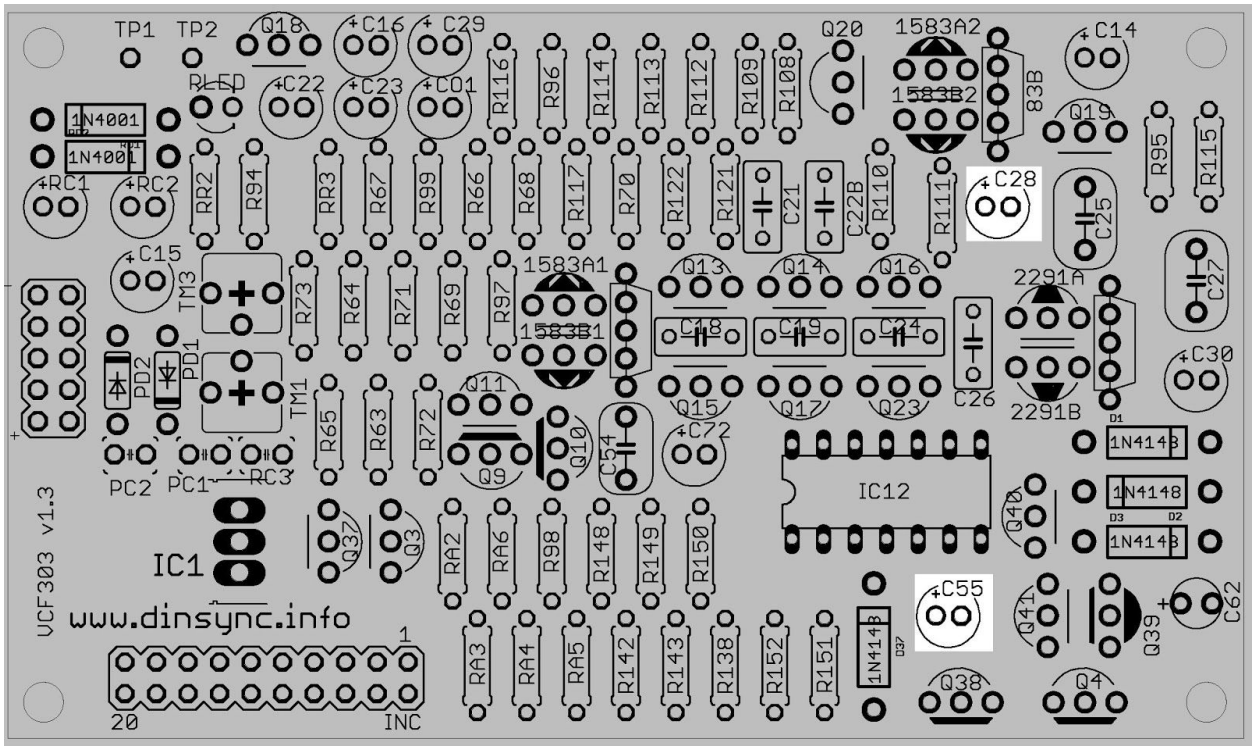

if you have a 14PIN IC socket place it now, if not place and solder your CD4066.



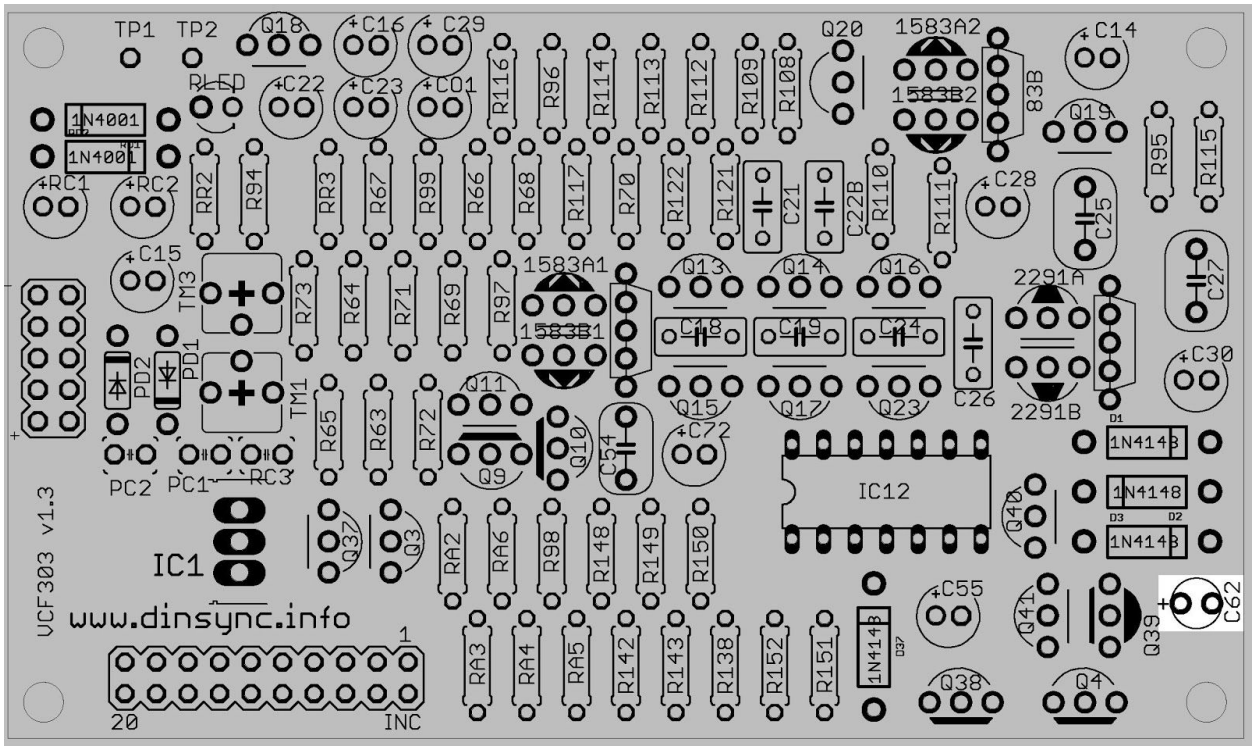

place and solder six 1uF electrolytic capacitors, again these are polarised.



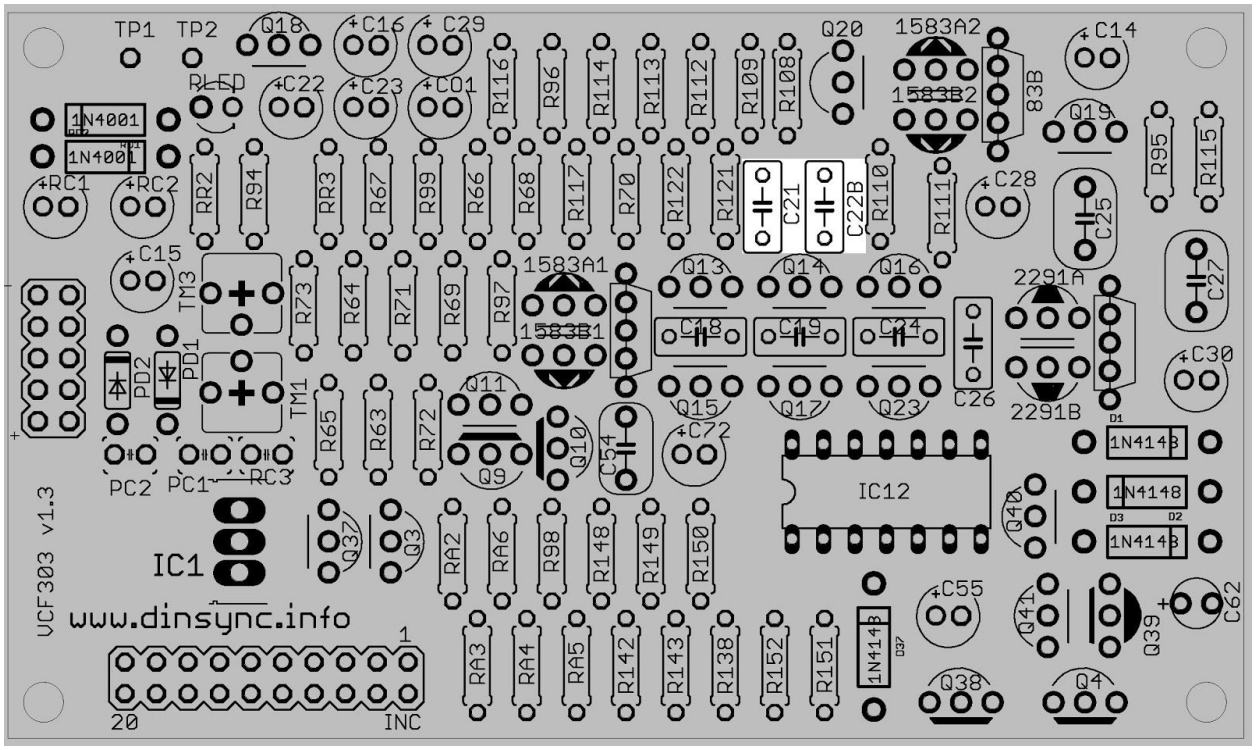

place and solder five 10uF electrolytic capacitors. Check orientation.



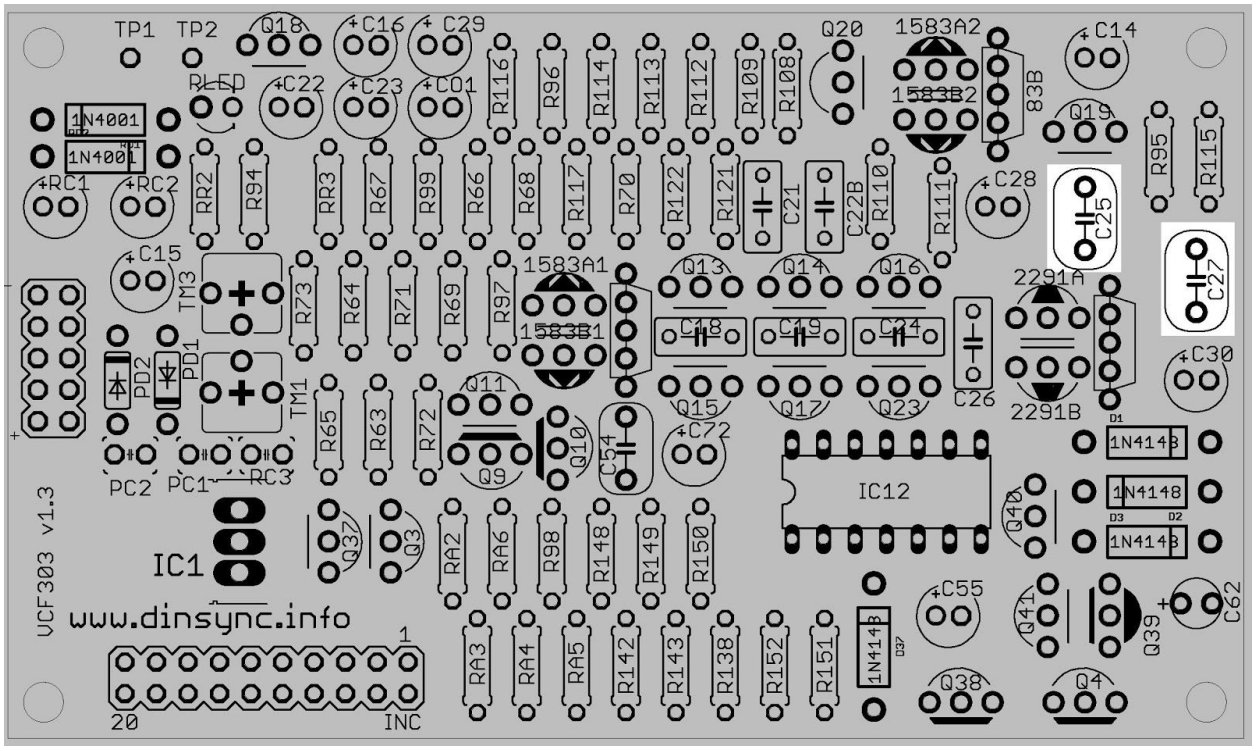

place and solder two 47uF Electrolytic capacitors. Again orientation should be checked.



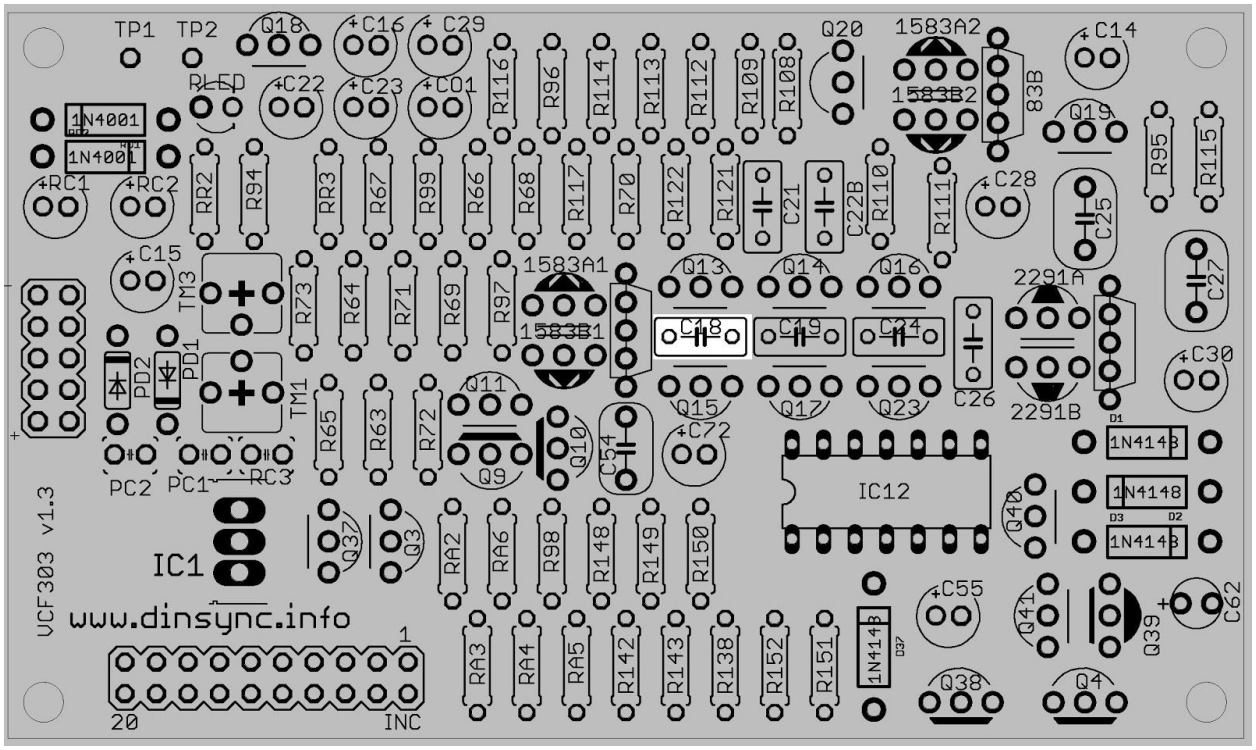

place the 1uF tantalum cap. Please note that the positive leg is marked on the body of the component (this is important to note since it is different to the way electrolytic capacitors are marked) and should be aligned to the hole with the plus sign. If this part is placed wrong, it could explode and possibly even catch fire!



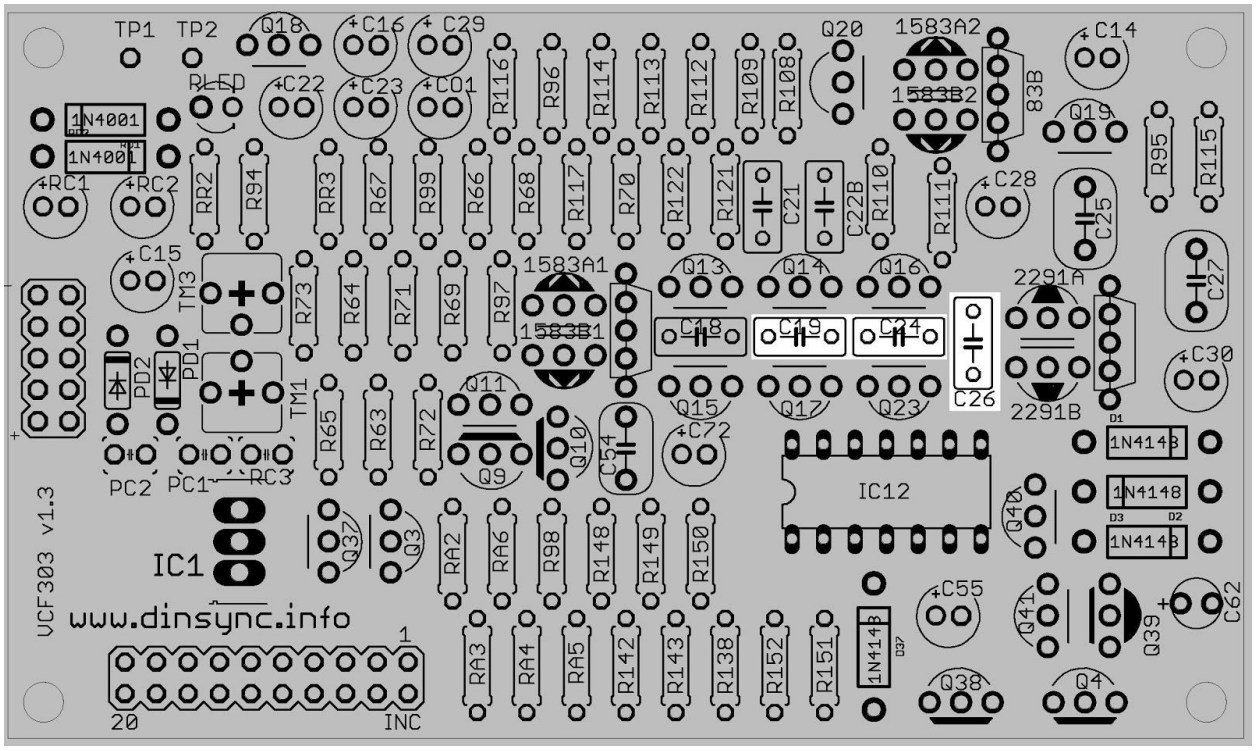

place and solder two 0.01uF poly capacitors



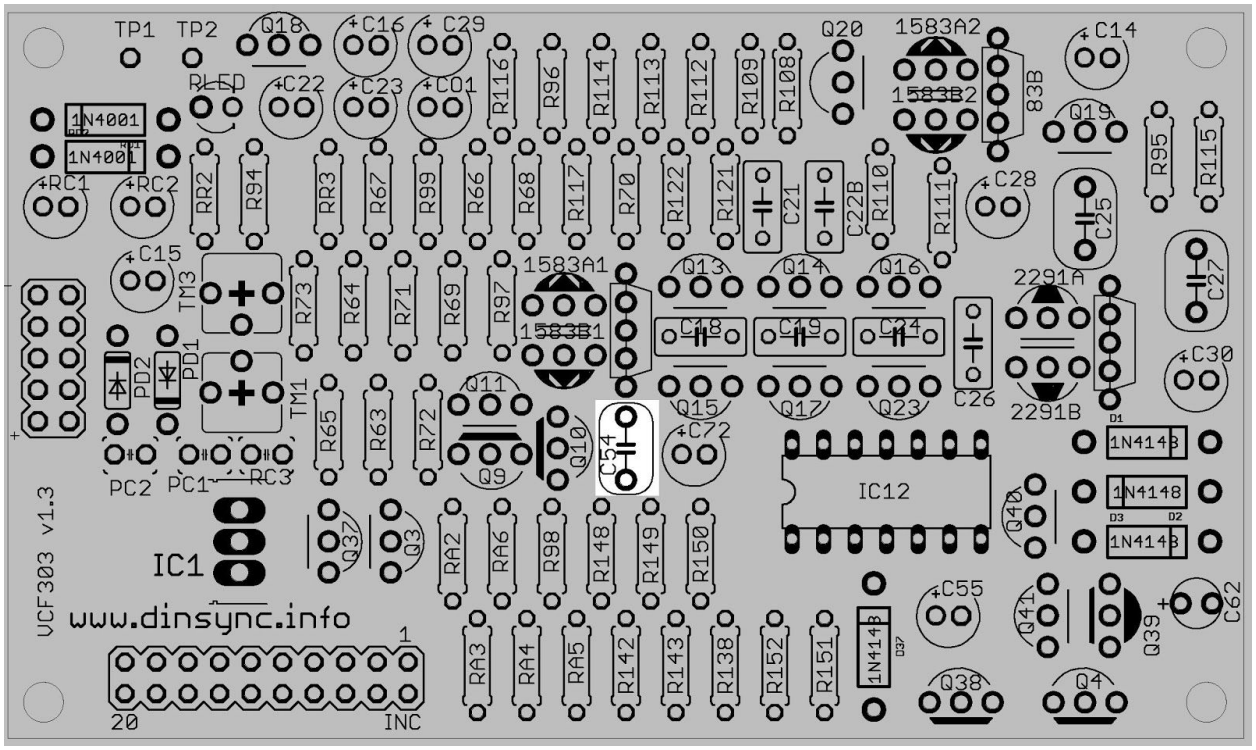

place and solder two 0.1uF poly capacitors (do not confuse with the ceramic 0.1uF)



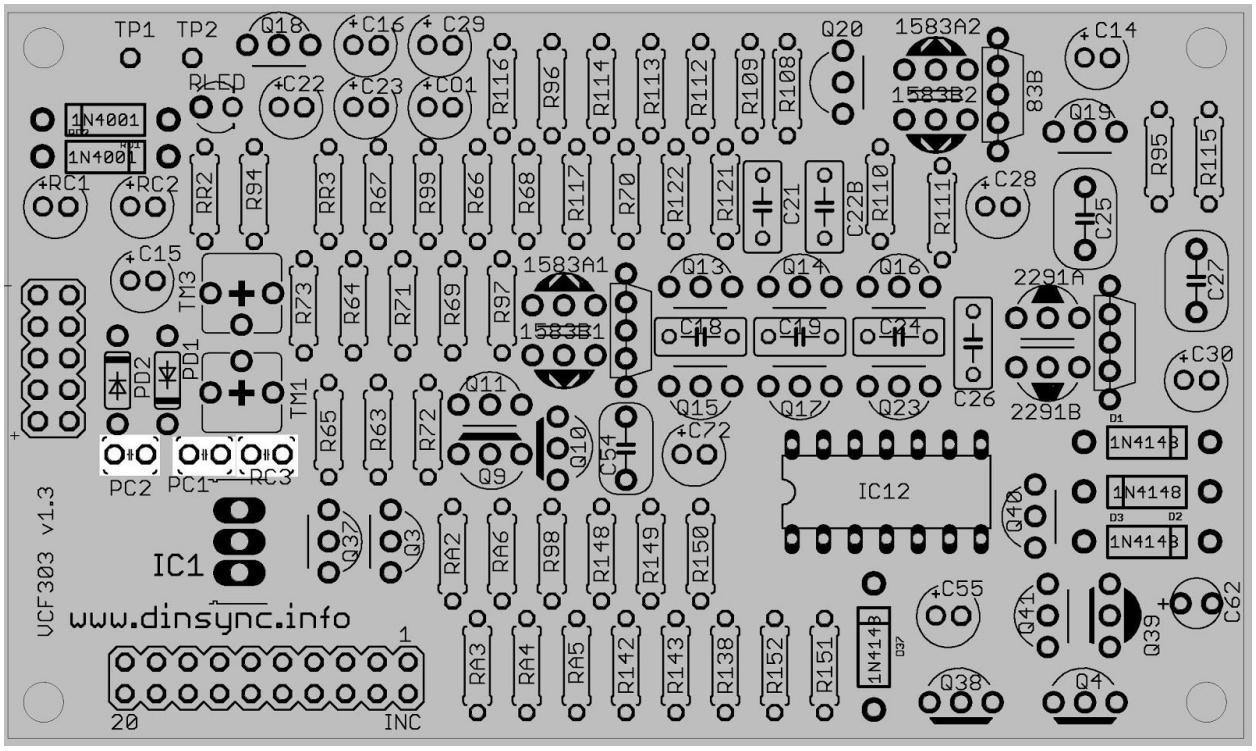

place and solder one 18nF poly capacitor.



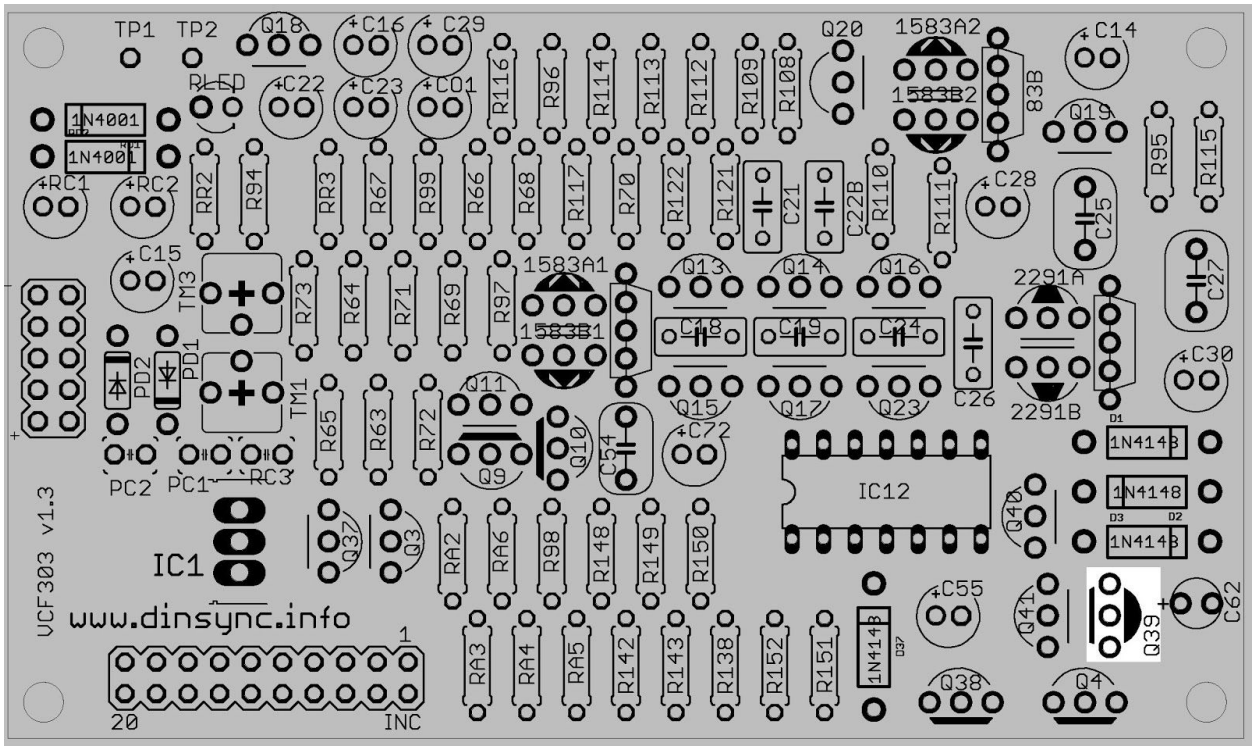

place and solder three 33nF poly capacitors.



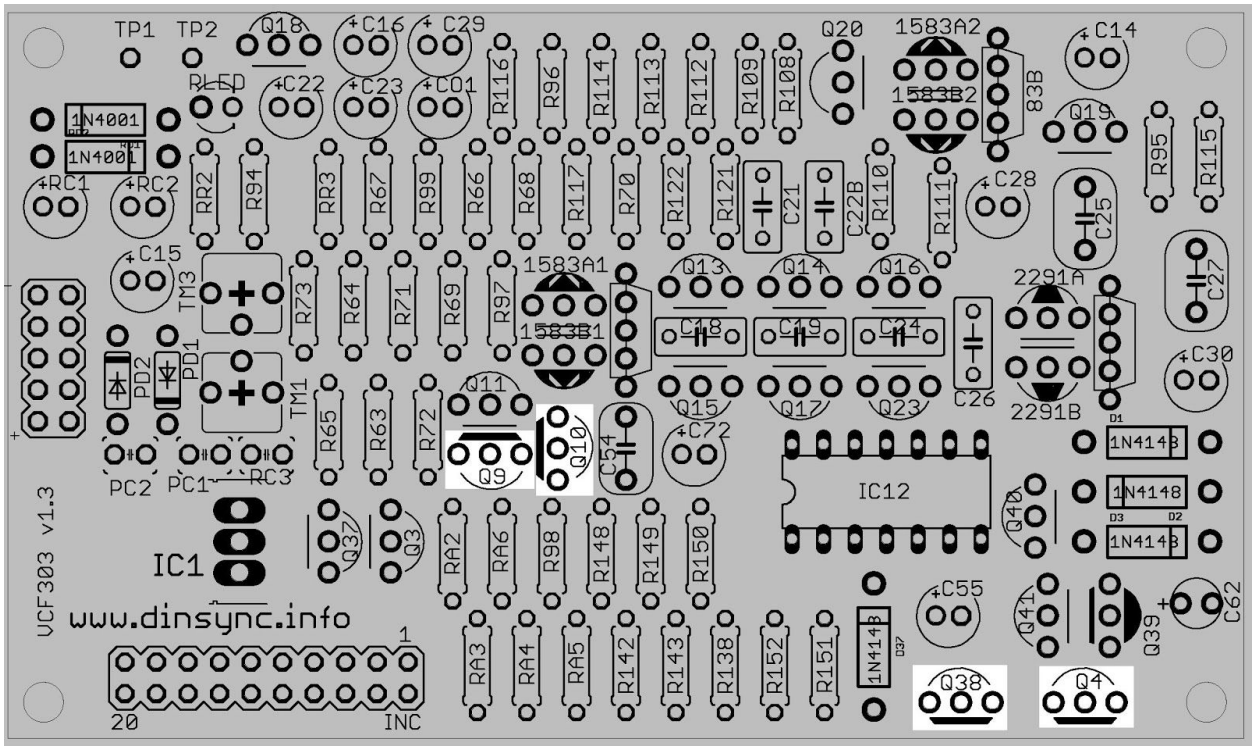

place and solder one 47nF poly capacitor.



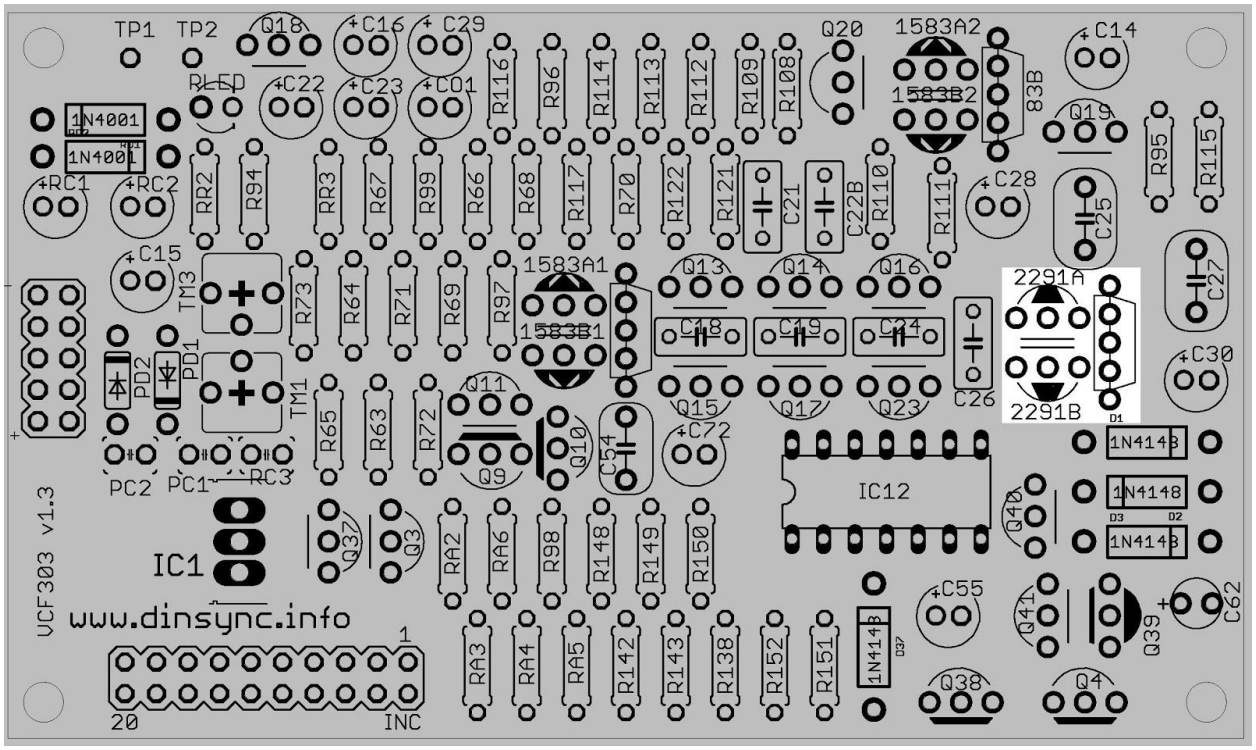

place and solder three 0.1uF ceramic capacitors.




place and solder the 2SK30A-Y transistor that came with the kit.

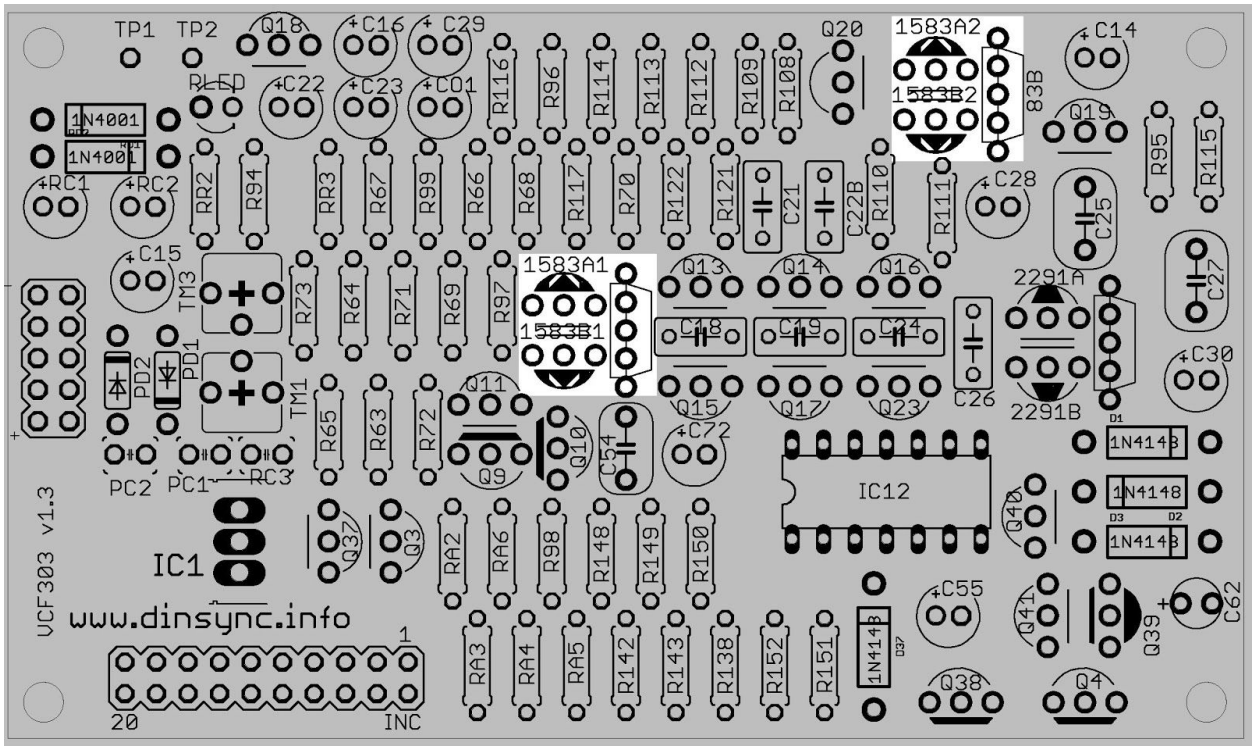



place and solder the four 2SA733 transistors that came with the kit.



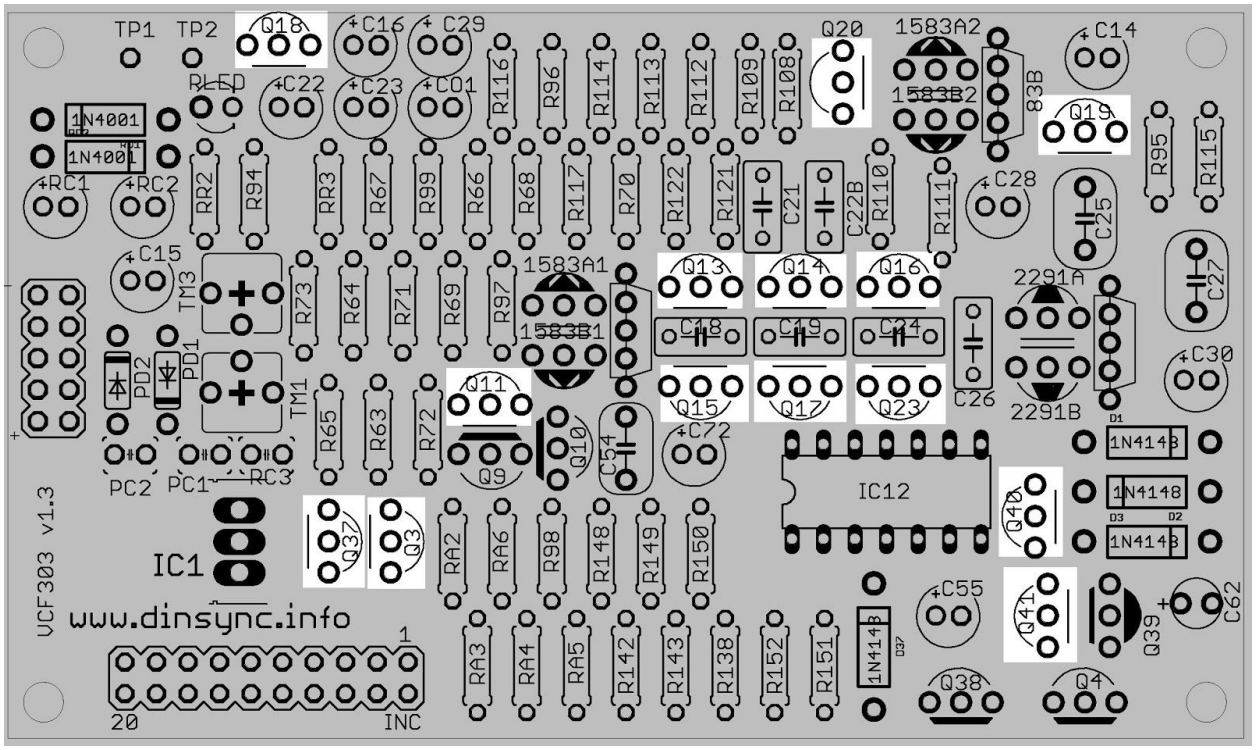

Place and solder the TRANSISTOR 2291A/B discrete pair (red top) provided with the kit.

Alternatively if you have an original 2SC2291 place this in the 5 pin adjacent slot on the right and do not solder this part.

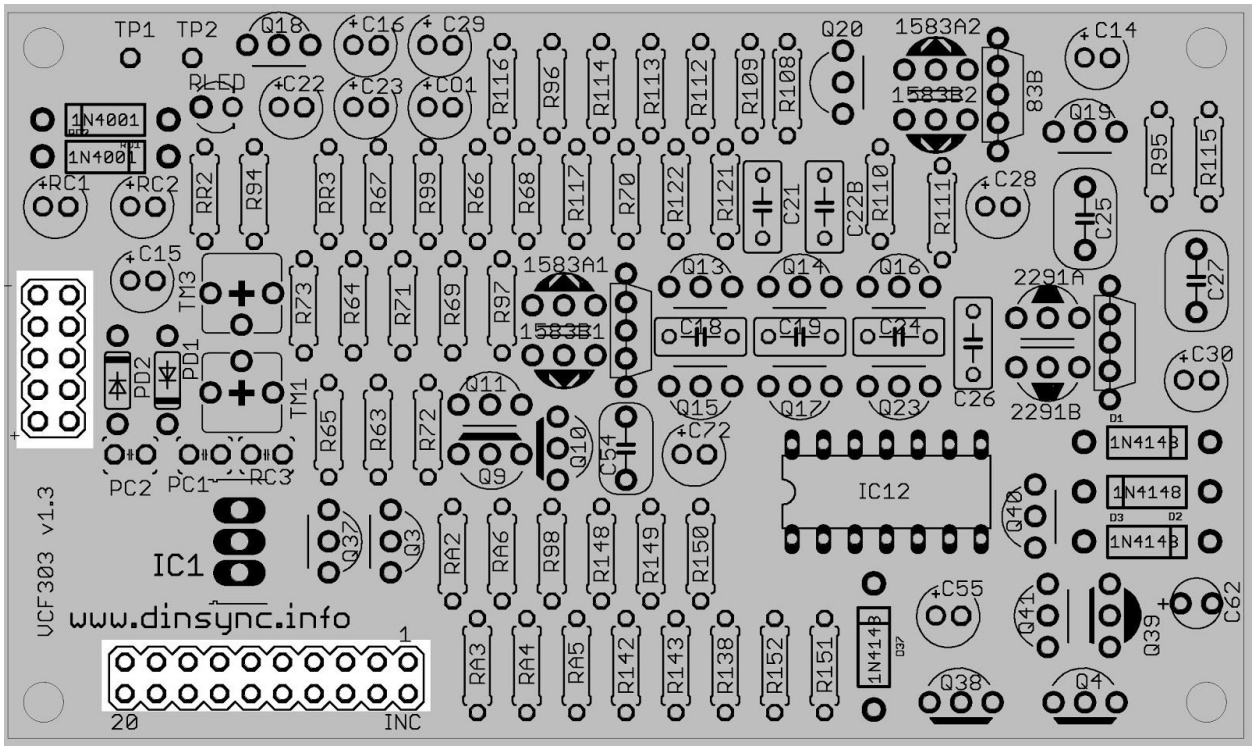



Place and solder the TRANSISTOR 1583A1/B1 discrete pairs (blue top) provided with the kit.

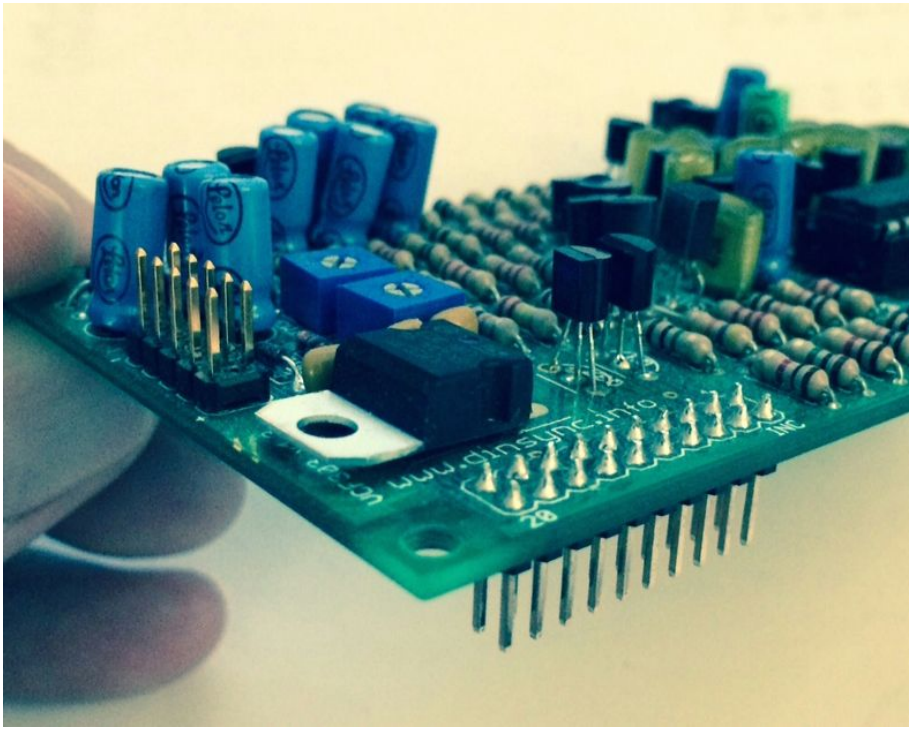
Alternatively if you have a pair of original 2SC1583 place these in the adjacent 5pin slots on the right and do not solder these parts.

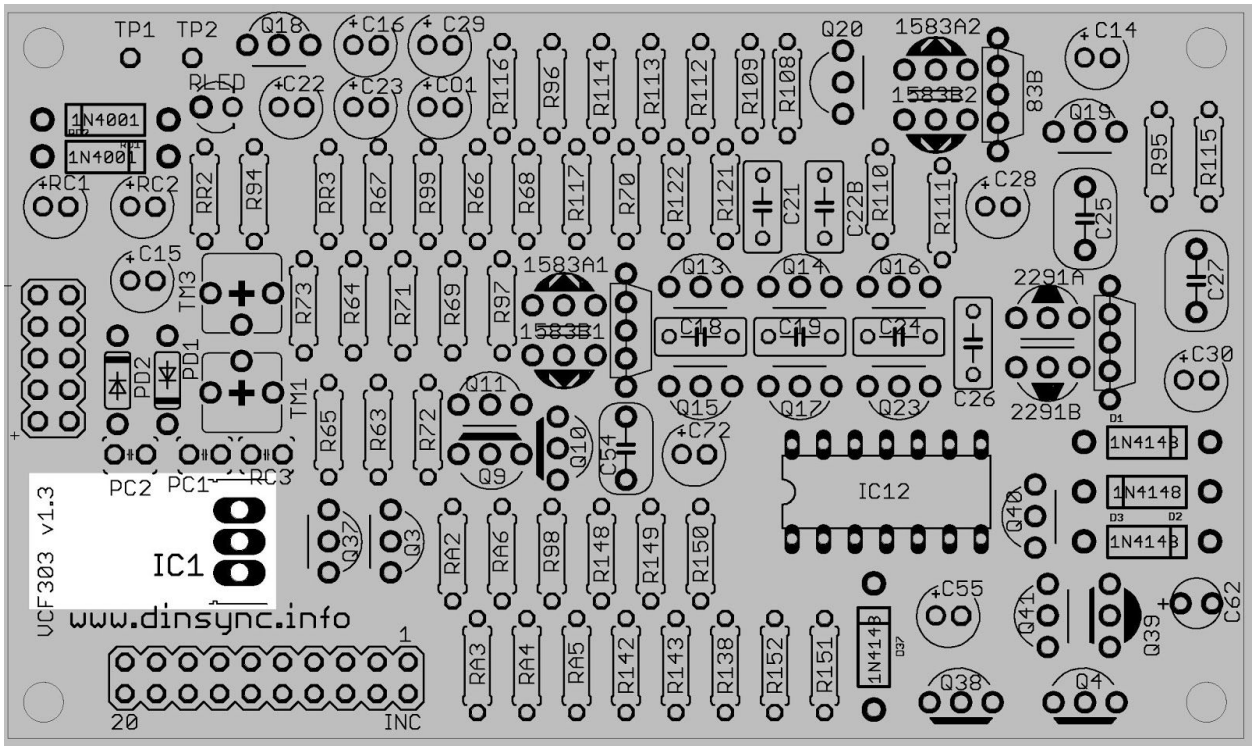



place and solder the fourteen 2SC945P transistors that came with the kit.




place and solder the 20 and 10 pin male headers. Check and double check the reference picture below before committing to soldering as removal later will be difficult without a desolder station.






finally place and solder the LM317. Gently bend it over gently but **not** so far that it touches the circuit board and that it looks like the reference picture below.

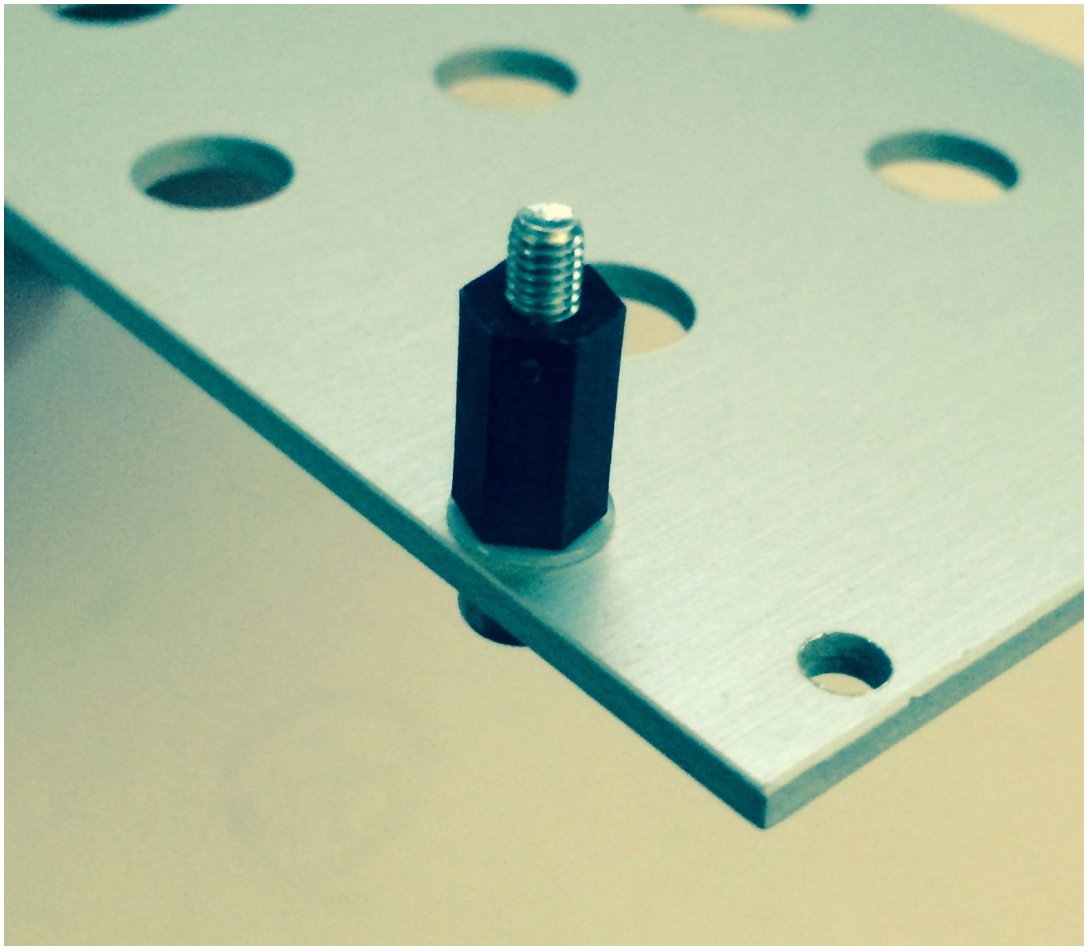
that’s it for this part of the build, time for a break, go drink a cup of dog and take the tea for a walk.

When you are rested let’s get on with the final assembly…

# Final assembly

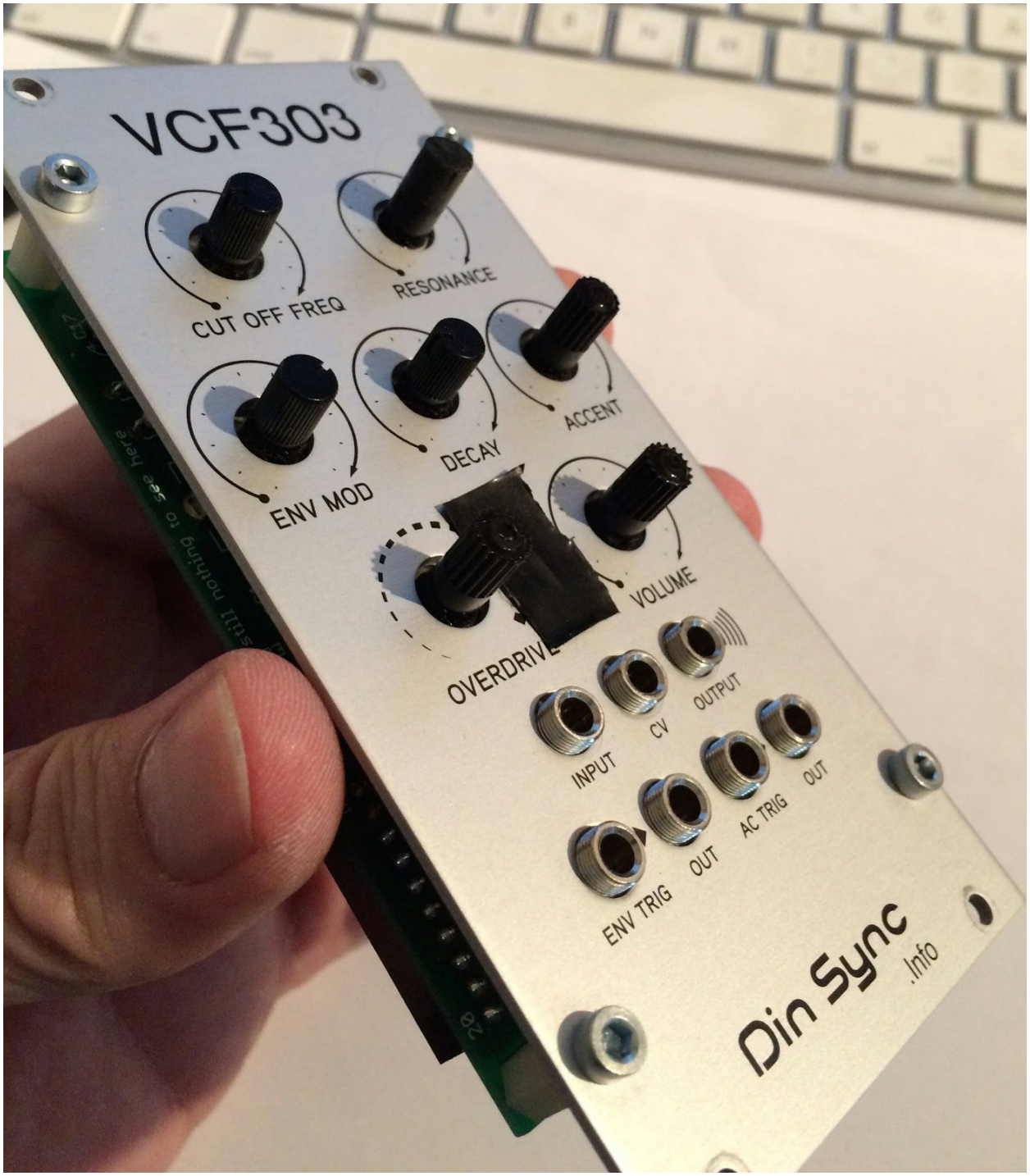
ok so now you should have completed your pcbs, all that remains is to assemble the panel hardware and solder the LED before doing the calibration. Also if you fitted IC sockets you should go ahead and place the IC's in them. Be sure to align the notch on the socket to the notch or pin1 marker on the IC.

Now grab your front panel, the washers, the hexbolts and standoffs.



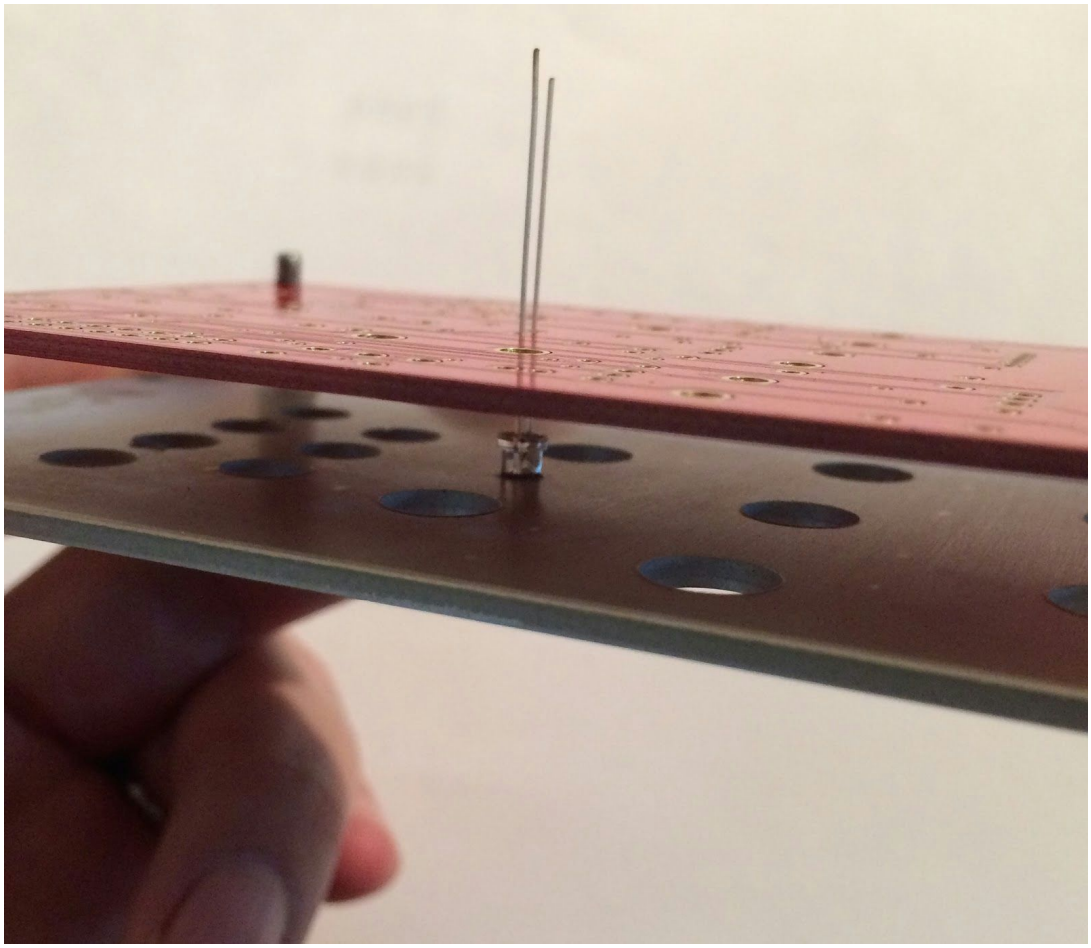
Place a hex head bolt through each of the four mounting holes, then place a single washer and standoff on. Finger tighten (don't clamp them down fully yet)

Now we shall fit the LED. Place the LED on the TOP board, the short leg should match the flat side of the silkscreen marking, **do not solder just yet**.




Grab your panel and place a small piece of electrical tape over the LED hole as shown here and place the panel on the TOP panel.

Now flip it over and let the LED drop down into the hole so that the tape can hold it in place.

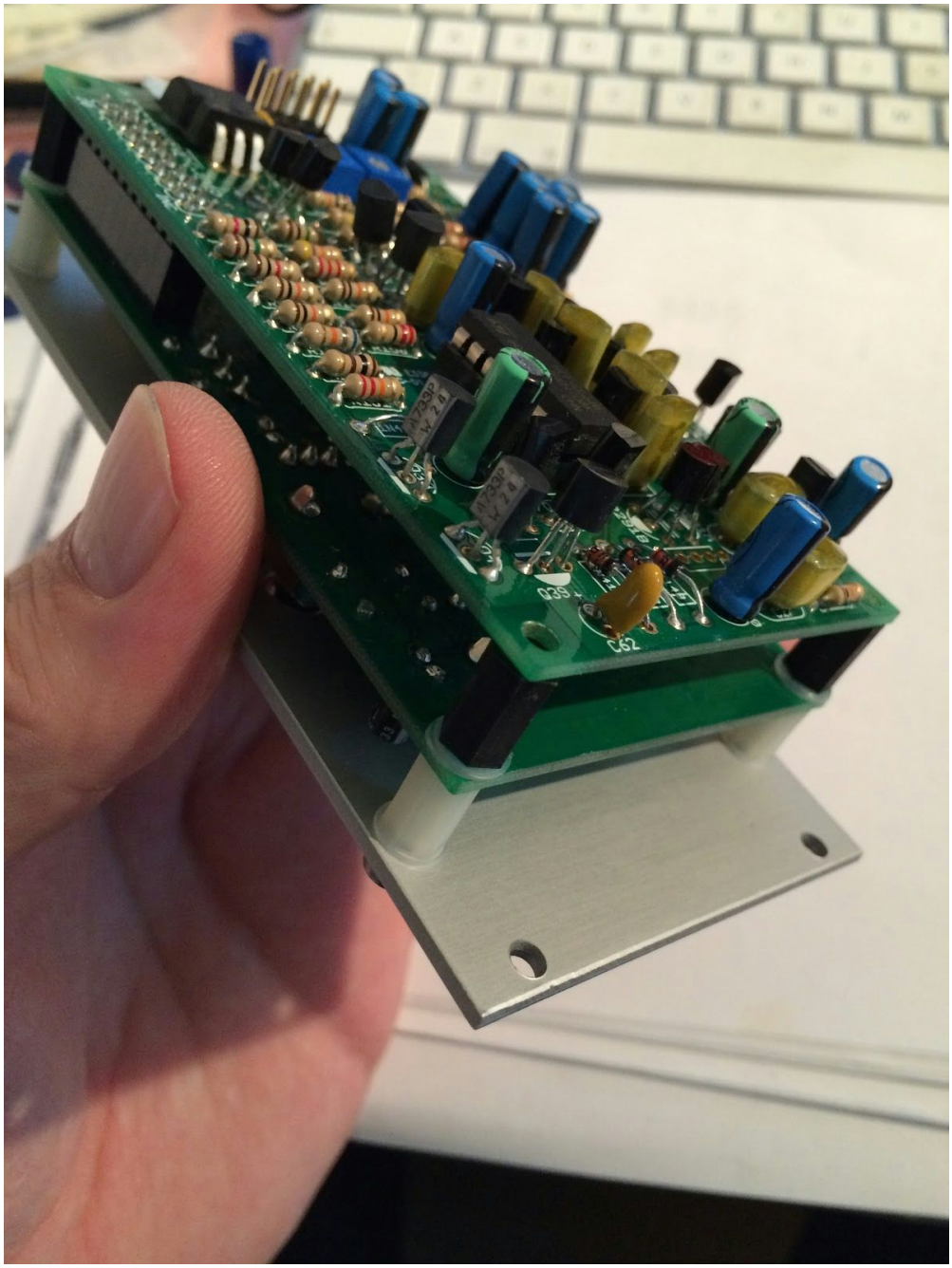


Here’s a picture without the controls soldered on (for clarity) of how the led should sit before soldering.

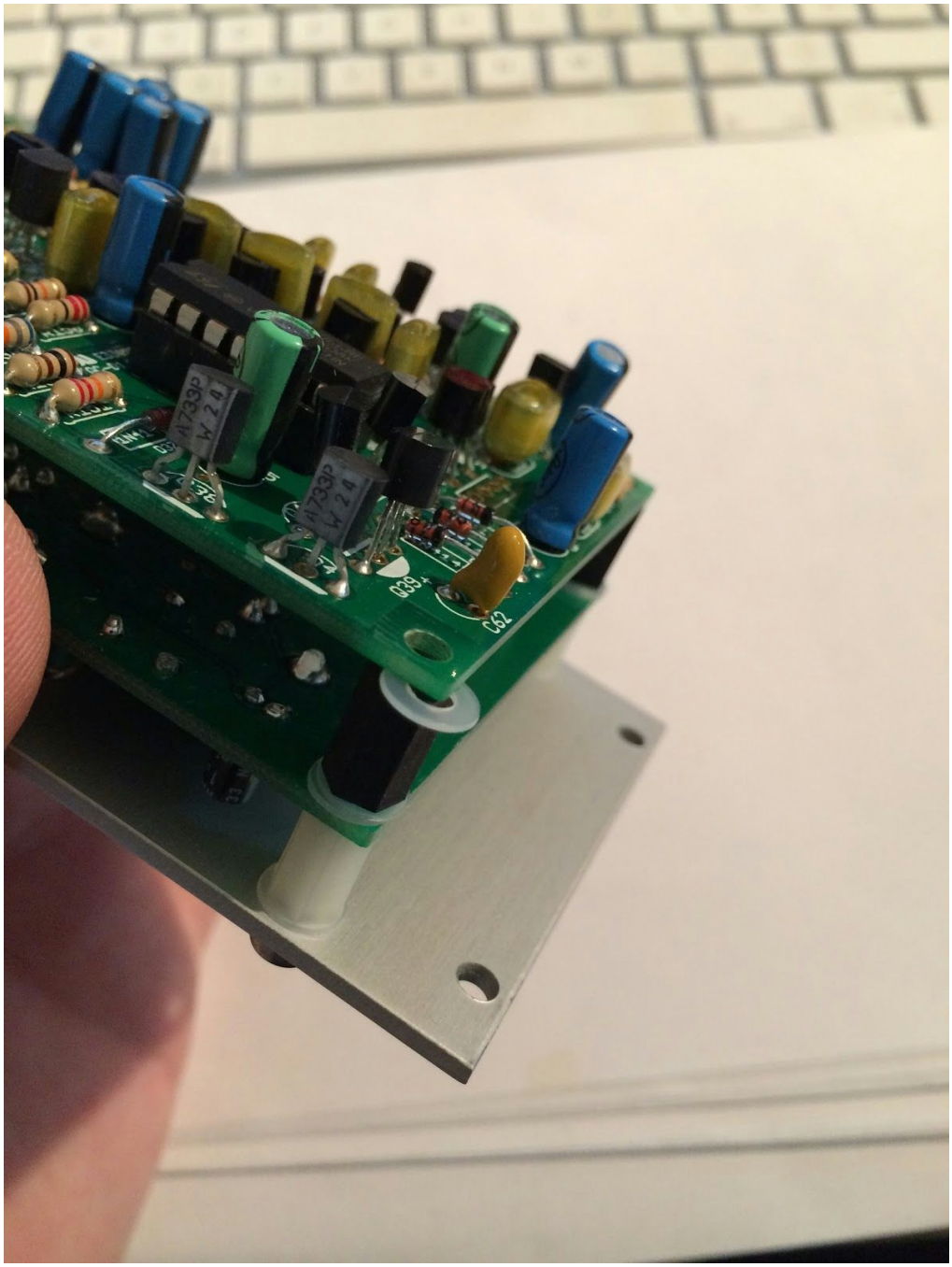
Don’t push down too hard on the LED as we don’t want it to protrude on the front, just let gravity and the tape hold it in place and **solder just one leg**.


Now peel off the tape and if the LED is sitting correctly solder in the other leg.

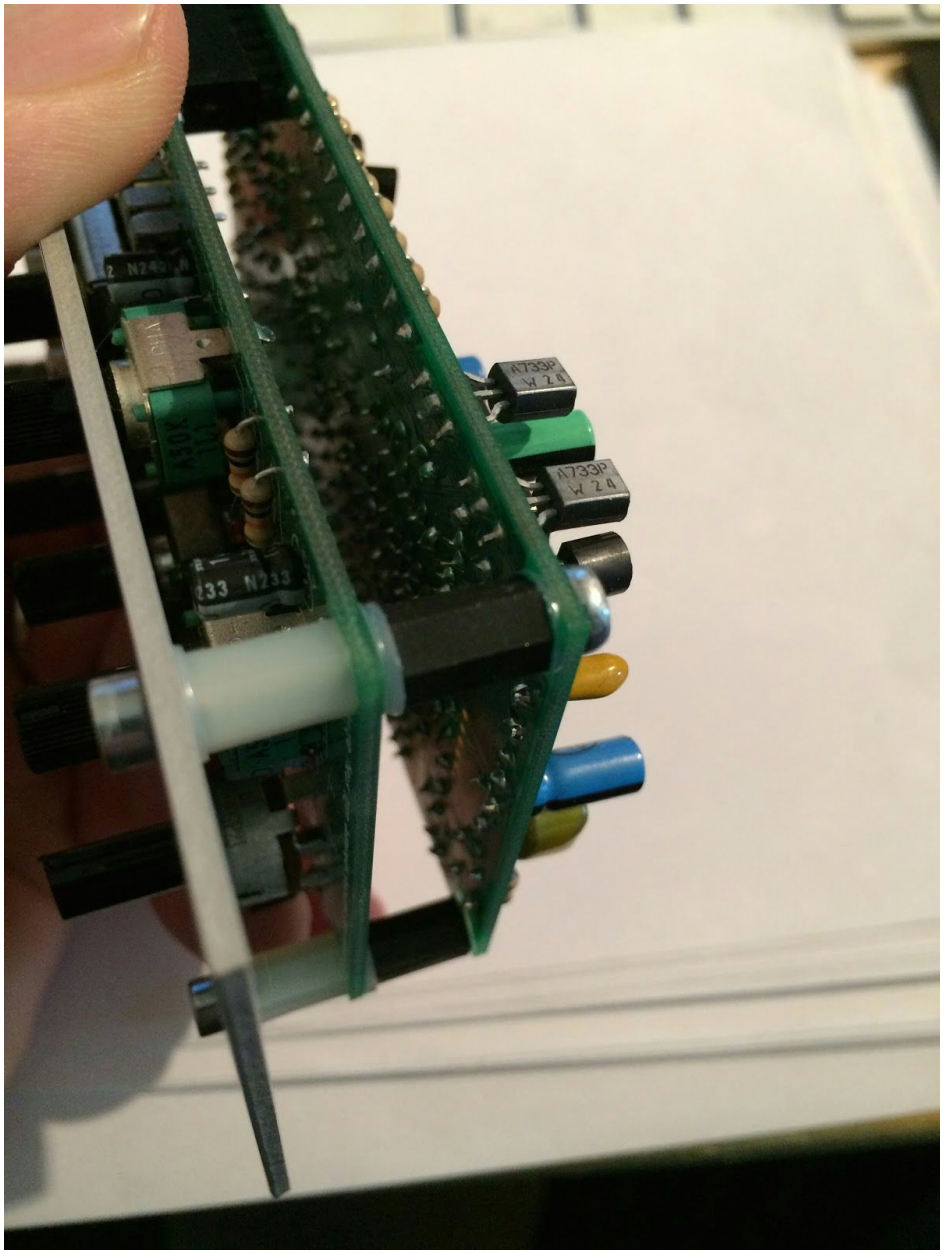
You can now put the knurled nuts on, finger tight for the moment.



place two washers (important) then a single standoff on each post and add the BASE board. Make sure the 20 pin header pins line up correctly.




before adding the screws place a single washer between the standoff and the pcb, this stops the pcb from bending when tightening the screws.



here’s a reference picture for how the standoffs should look. Don’t mind that one of them is white.

from the panel you should have one washer, one standoff, the top pcb, then 2 washers, one standoff, then the base pcb.

Once everything is aligned nicely, you can tighten the hexbolts a little (you don’t need to overdo this as it’s not a car wheel) then add four m3 screws to hold the pcbs in place. Finally you can tighten the knurled nuts, be careful not to scratch the front panel when doing so.

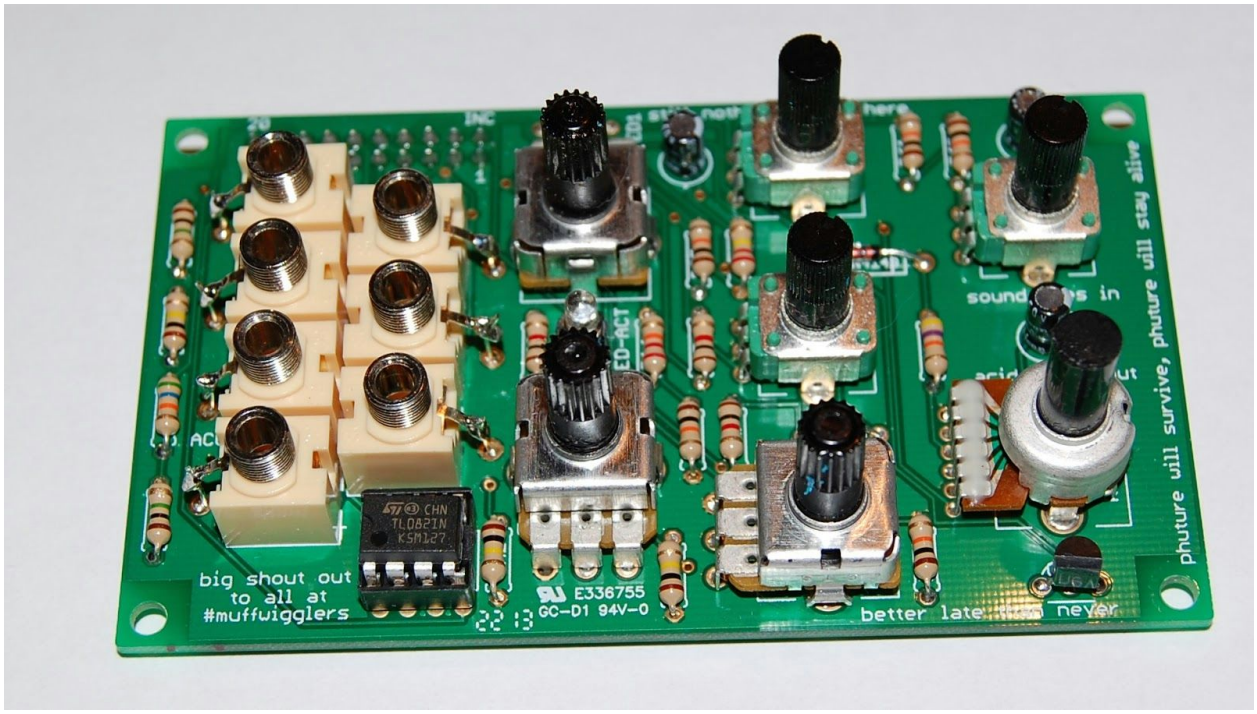

now you can add your choice of grub screw knobs, I prefer the dark blue 1900 clones from small bear but the choice is yours of course.

All that remains is to calibrate and test the module, full details of how to do this are at the end of the user manual that was supplied with this document.

I hope you had fun building the module, if you have any problems please drop me a line on info@dinsync.info and we'll figure it out.

Paul

# Reference pictures


big shout out
to all at
#muffwigglers
TL082IN
E336755
GC-D1 94V-0
phuture will survive, phuture will stay alive