

# TOTORUS

-- Build Guide --  
f.aiseur - 2025



First of all, thank you for ordering a PCB of the TOTORUS!!

It all started on a french DIY forum, Anafrog, with the great work of Damien on "cloning" the Moog Taurus. I use "" because it IS NOT a clone. It is a stripped-down version of the original, without presets, just the analog sound part. On the other hand, we added some extras, that ARE NOT on the original : V/oct conversion and VCF CV input.

It was designed to be put in a specific case, the Hammond 1455L2201 / 1455L2202 (the difference is on the side ends : 01 has aluminium, 02 has plastic), but of course you can use the pcb to the casing of your choice.

Another point to mention : the Beat Rate and Foot Loudness trimpots. They were originally normal potentiometers, but as I had (or actually thought I had) space issues, I had to make some choices. As it seemed the two less useful pots, I decided to make them trimpots. It turns out the beat rate is quite nice to tweak, so one could easily make a hole on the panel to be able to turn the trim with a small screwdriver. The Foot Loudness isn't so critical but still needs to be calibrated to taste. Once it is done, I think it's not really necessary to tweak again, but there too, a small hole on your panel would let you do that. Actually on my prototype I ended up using a fixed resistor of 4k7 to replace the trimpot.

Its a compact build, so, space will be your enemy here! Keep your leads short, think about your choice of electrolytic capacitors (not only the height but also the diameter, if you will solder them horizontally on the solder side) and try to be....compact.

Except this, we will have to match some transistors. The first time I needed to do this, it was for a MME build, and I ordered a PCB for the Ian Fritz matcher. Worked great but now, I mostly use a cheap component tester from Aliexpress. So it's up to you how you'd do that. A whole bunch of informations/techniques are available online so I won't go into this detail here.

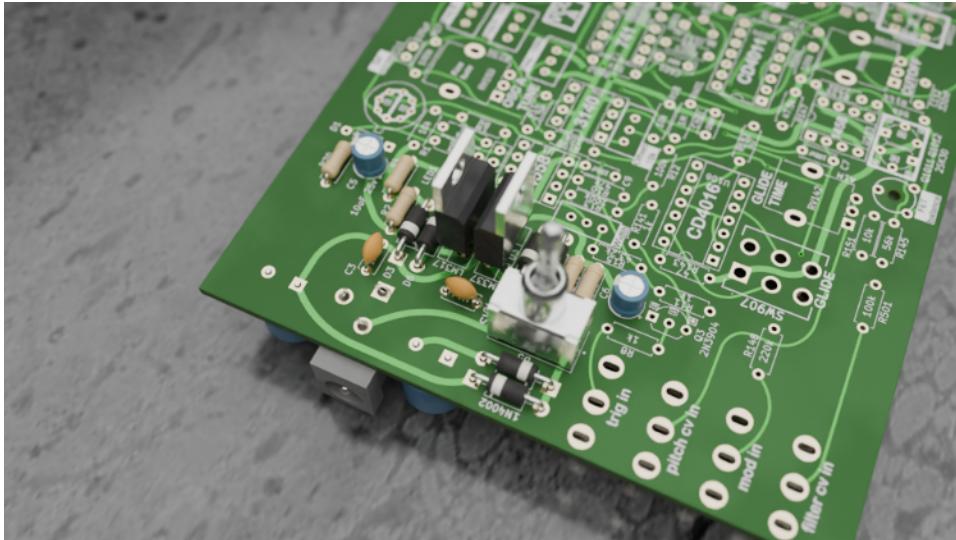
Be careful with the pinout of each transistors, it might not be the same as what is on the silkscreen!!

Now let's start building your future favorite bass synth!

1/ check for shorts between power rails (+15, -15, GND) - just in case!

2/ First, we build the PSU :

So you should start by :



- diodes 1N4002 (D2, D3, D4, D5, D6, D7)

- resistors (R1, R2, R3, R20, R21). Measure the resistors before you solder them, as they are responsible for getting correct voltages!! You should also fit now R4, the 1k LED resistor, because soon, you will cover it with the bent LM317.

- ceramic capacitors 100nf (C3, C4)

- small electrolytic capacitors : C5 and C6 are 10uf - depending on their height you can solder them on top.

Now flip the board and add :

- power connector, solder side of the pcb, as it will sit between the two.....

-....big electrolytic capacitors : C1 and C2 should be soldered horizontally

Flip the board again

- then its time for the two regulators (U2, U3). It should be pretty straightforward. I recommend to test fit them first, bend their leads at 90degrees, and solder them only after they are bent, to avoid additional stress to the solder joints. The main goal being, as you expect, to set them relatively low, but without ANY contact (between them, or with components around). It looks sketchy to some of you probably - but once they are soldered, they wont move much.

- finally, you should put the power switch. But to do that properly, it's important you already have a case (with holes prepared!) so you can align nicely the switch in his position. If you went the Hammond 1455 way, please check the paragraph about it later in this build guide.

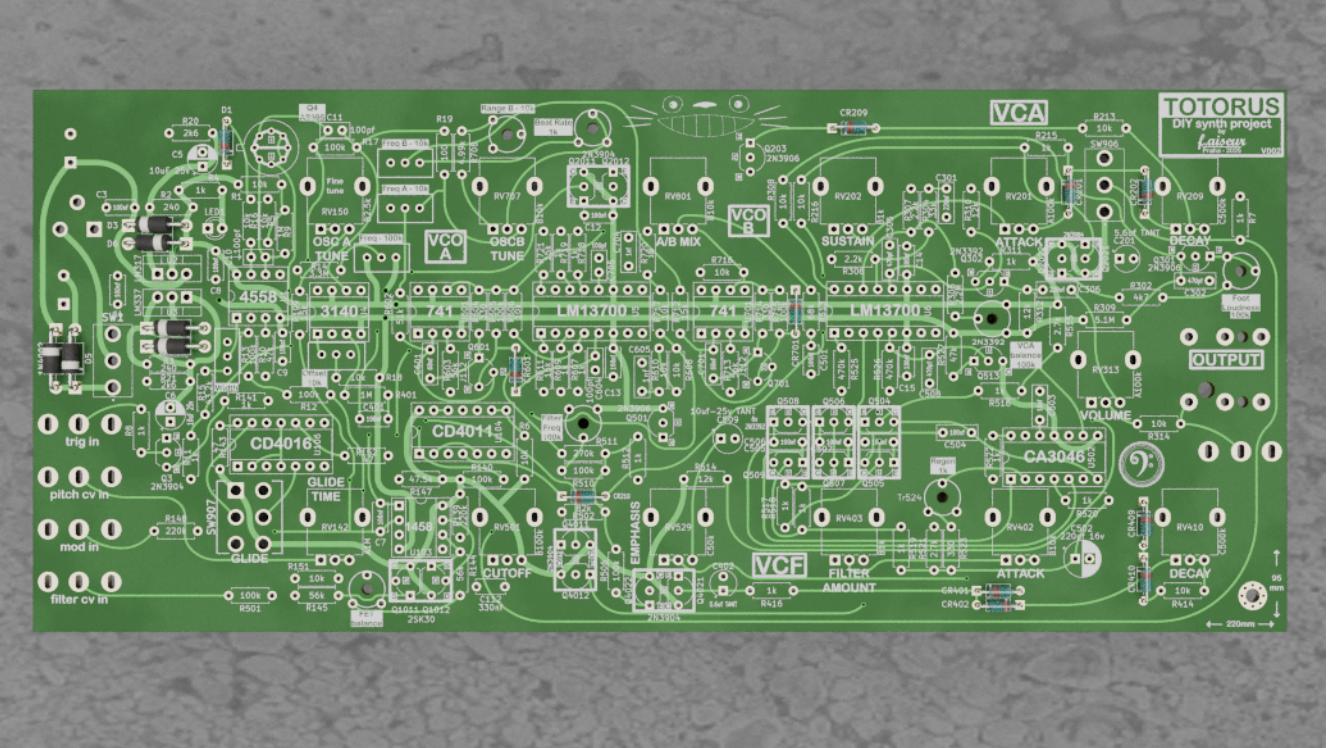
But either way, you need to make contact and test your voltages (you can solder a wire and bypass the switch of course...).

There are many spots where you can check your voltages. The mounting hole on the lower right side, where the dimensions are written, is connected to ground, so you can just clip your DMM here and forget about it. For + and - 15v, you can check at the power pins of the ICs (for example, you should have +15v at U1 pin5, and -15v at U1 pin4, the 4558 just after the PSU)

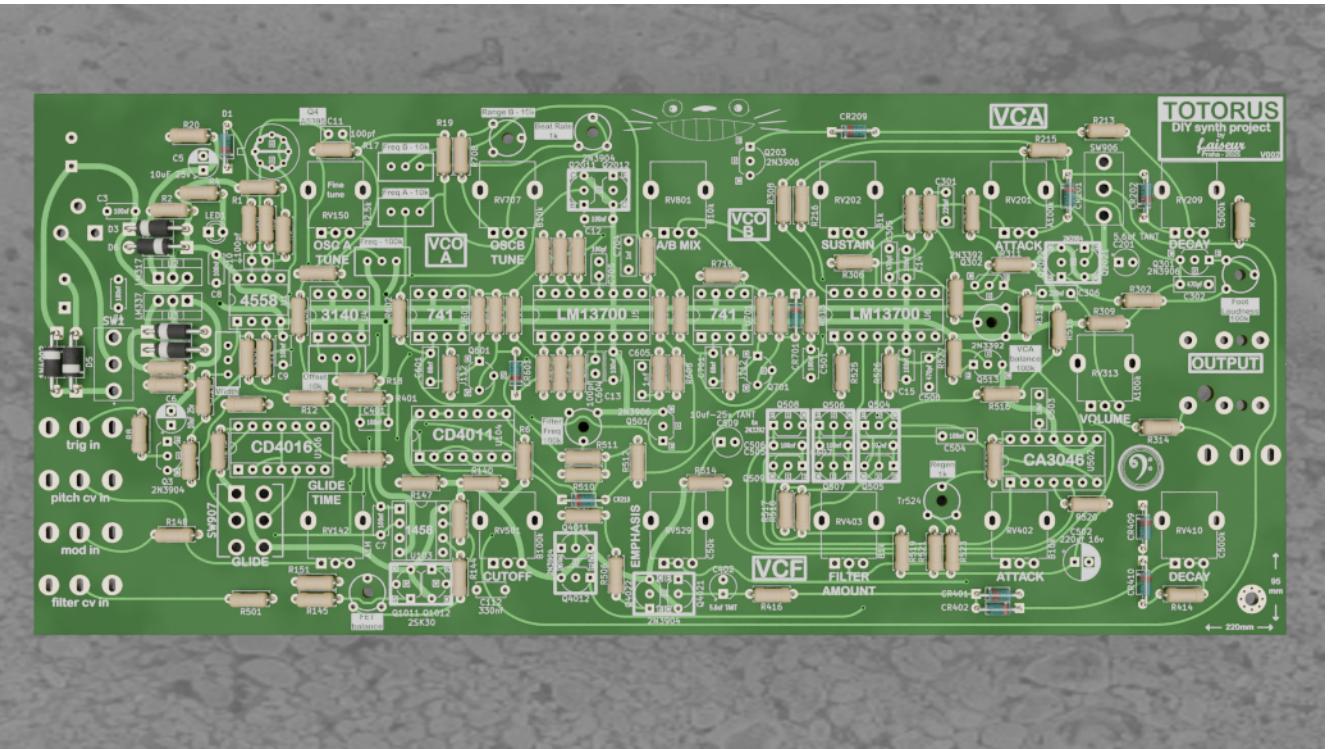
IF you have those voltages, continue. If not, you should find where is the fault now, and get them right!

Now, you can start building the whole synth, as usual starting with smaller components. It is always a good idea to check every single component before soldering. It takes time but will save you later.  
 ( I made the 3d renders without PSU parts populated so please forgive me for being lazy and not do all the rendering again...)

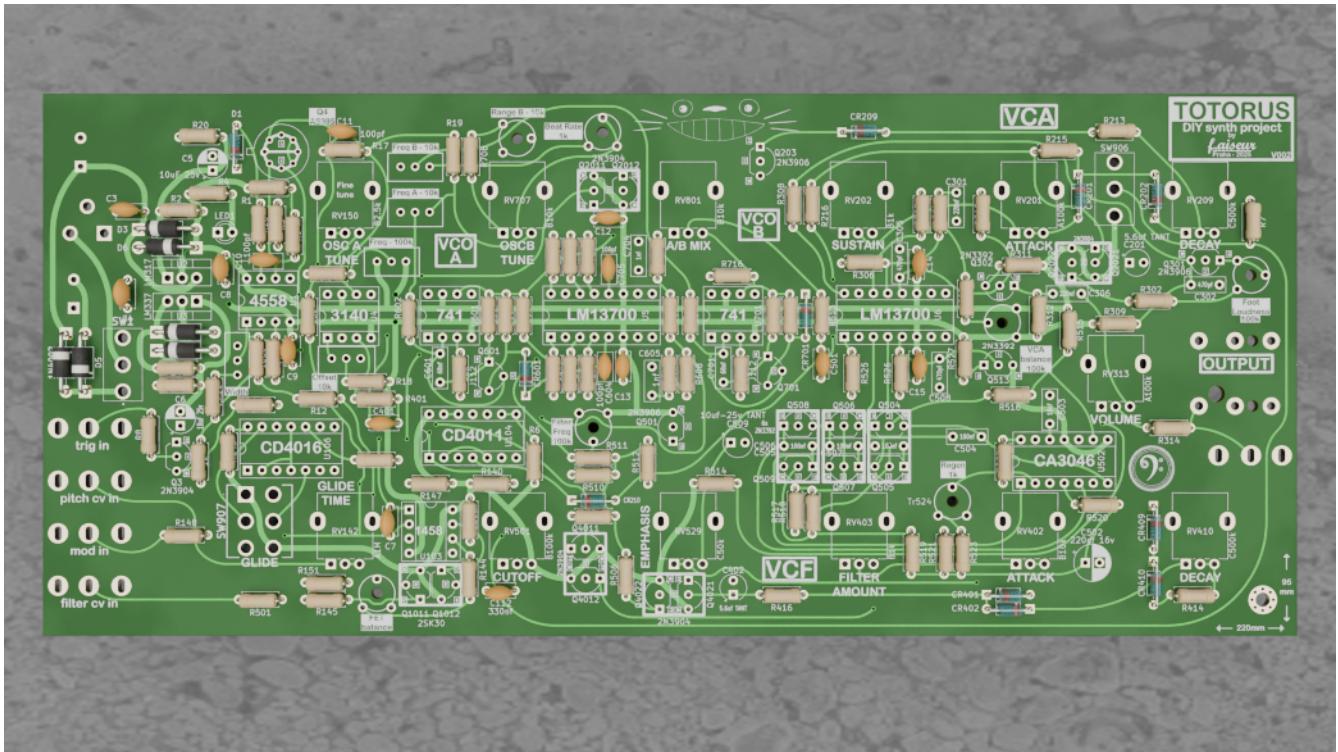
First, put the diodes 1N4148.



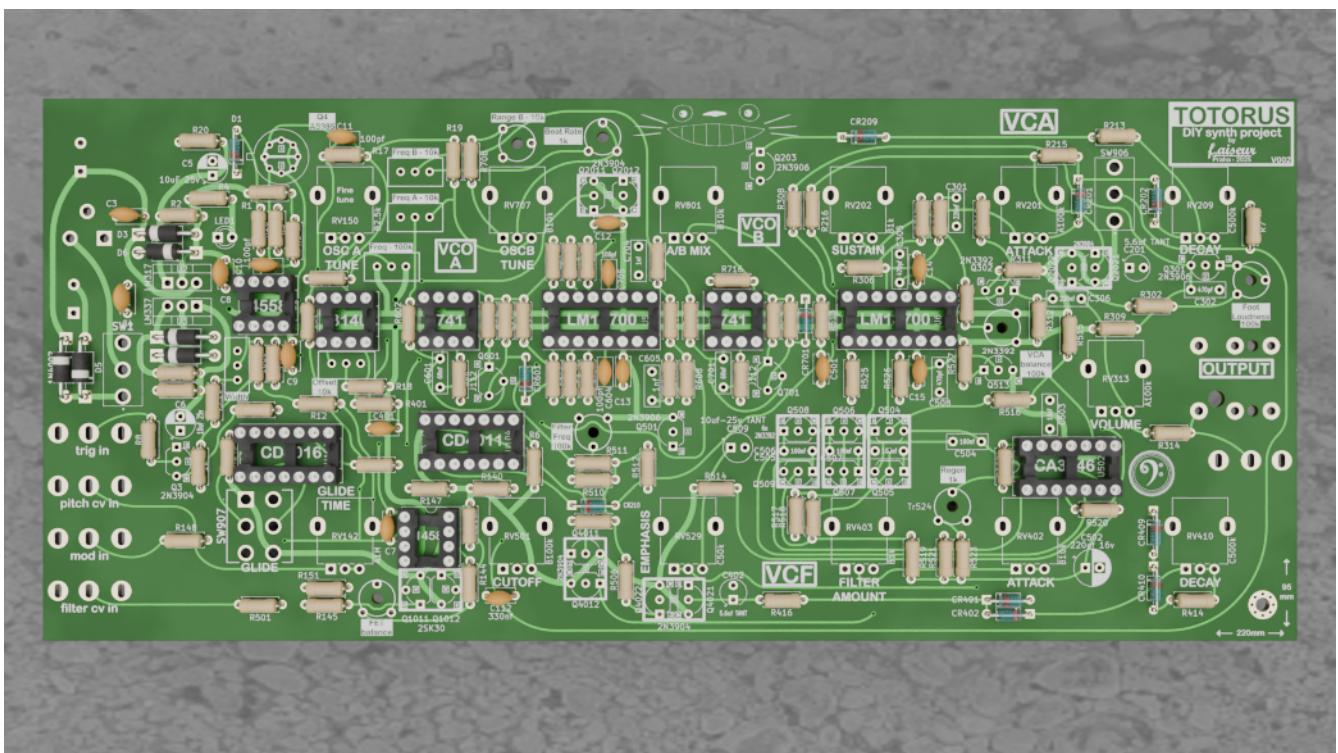
Then all resistors.



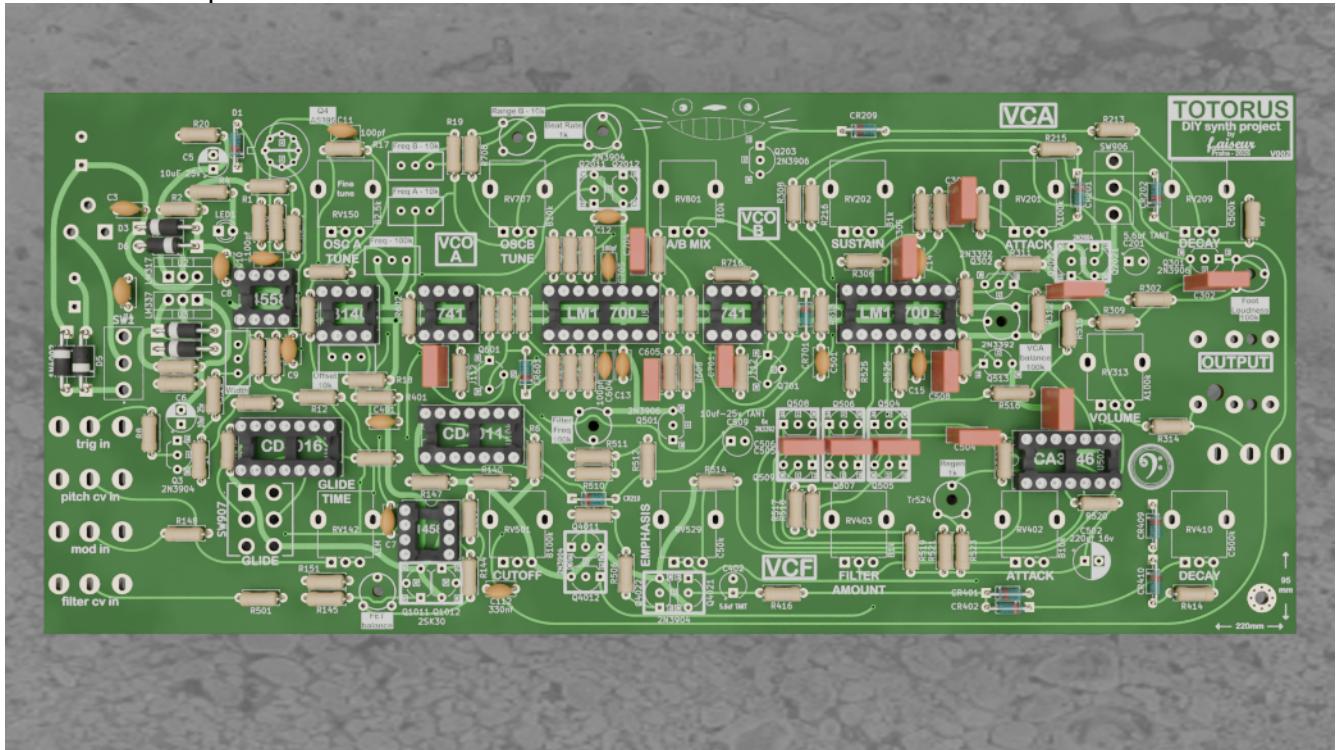
Ceramic capacitors.



If you use IC sockets, you can solder them now.

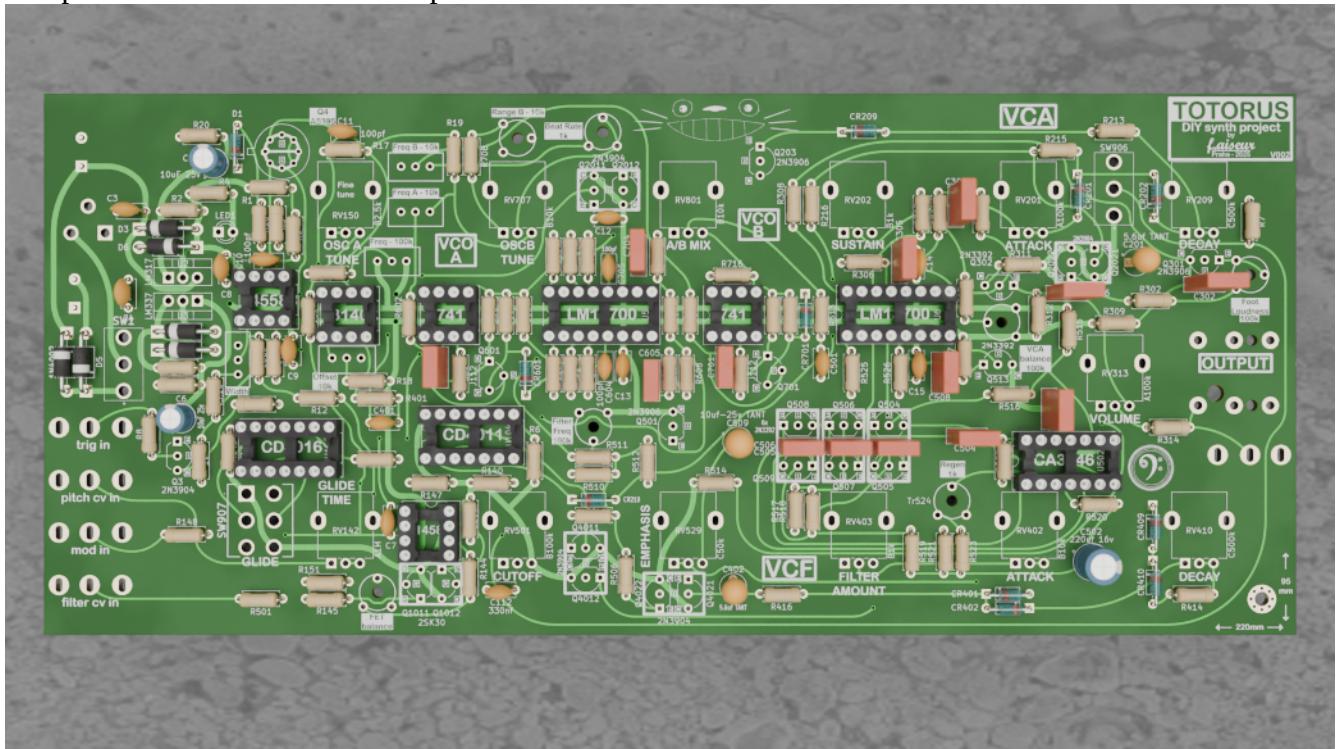


Time for film capacitors.



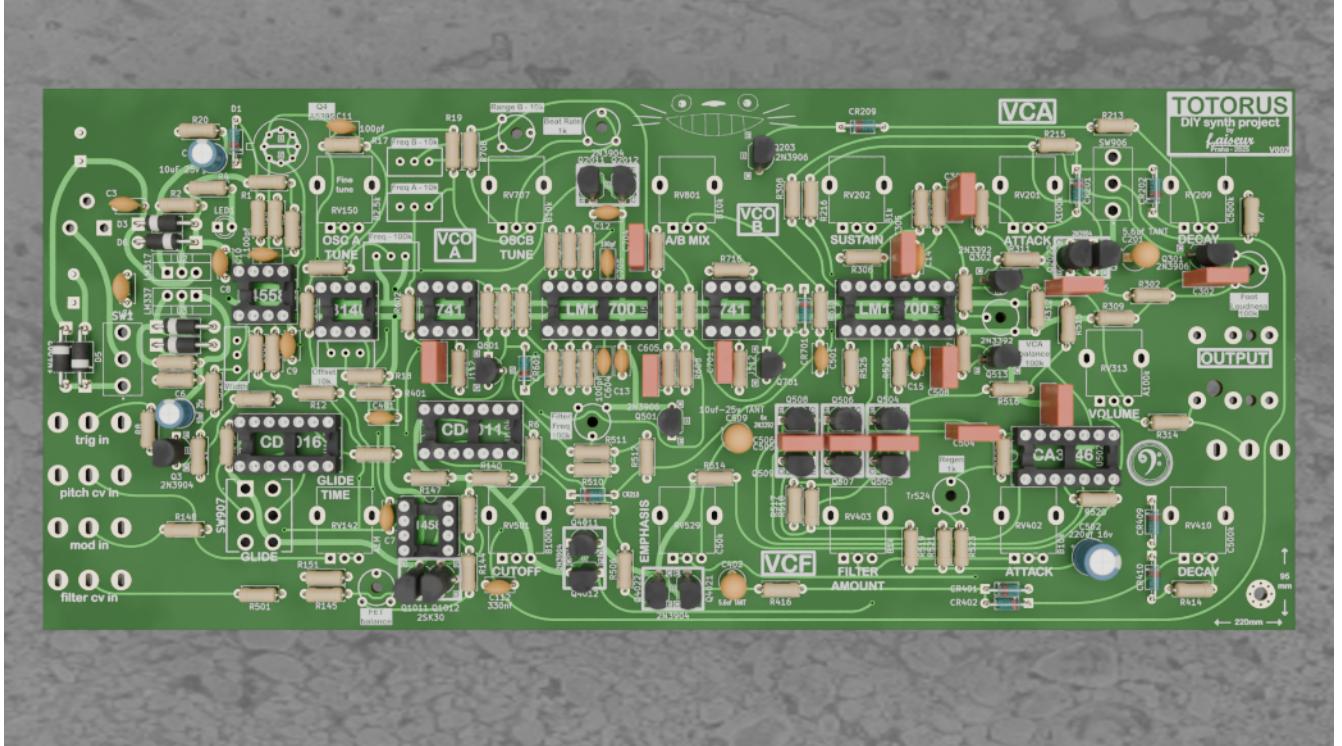
NOTE : two 68nF (C601 and C701) were originally POLYSTYRENE capacitors. On my build, i used regular film caps. They are supposed to deal with the tuning of each VCO so you might want some quality capacitor here!

Next, electro and tantalum capacitors. You can -maybe- find a small 220uf that would fit on top, but he has space on the other side of the pcb.

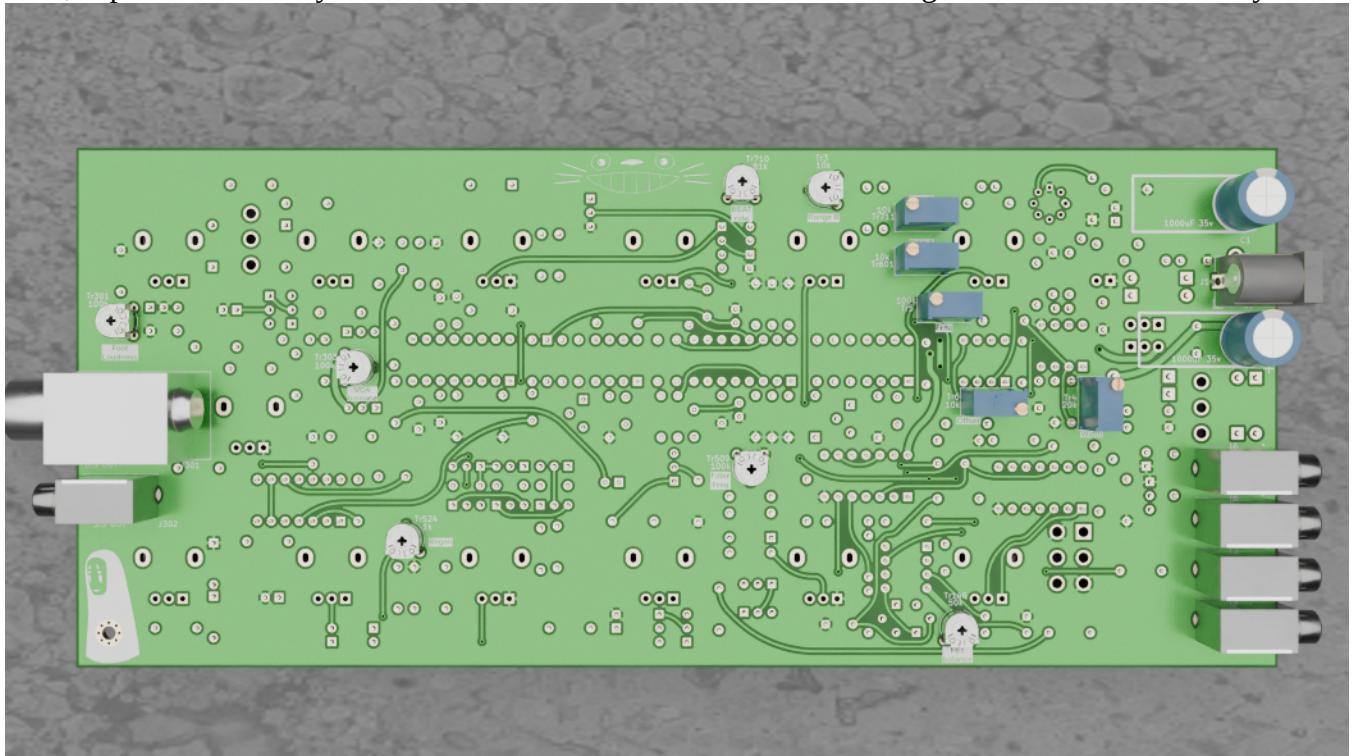


Now, the important part of transistors. Many of them in this build, and some matched pairs needed. Be careful with the pinout!!!! I learned when debugging my prototype that MPS3392 (the ones that i bought from mouser) and 2N3392 have different pinout, so you really have to triple-check against what you plan to install!

NOTE : the V001 had pinout AND silkscreen correct for 2N3392. The V002 has correct pinout for MPS3392, even tho silkscreen still says "2N3392". I will eventually make a next revision that would be less confusing...

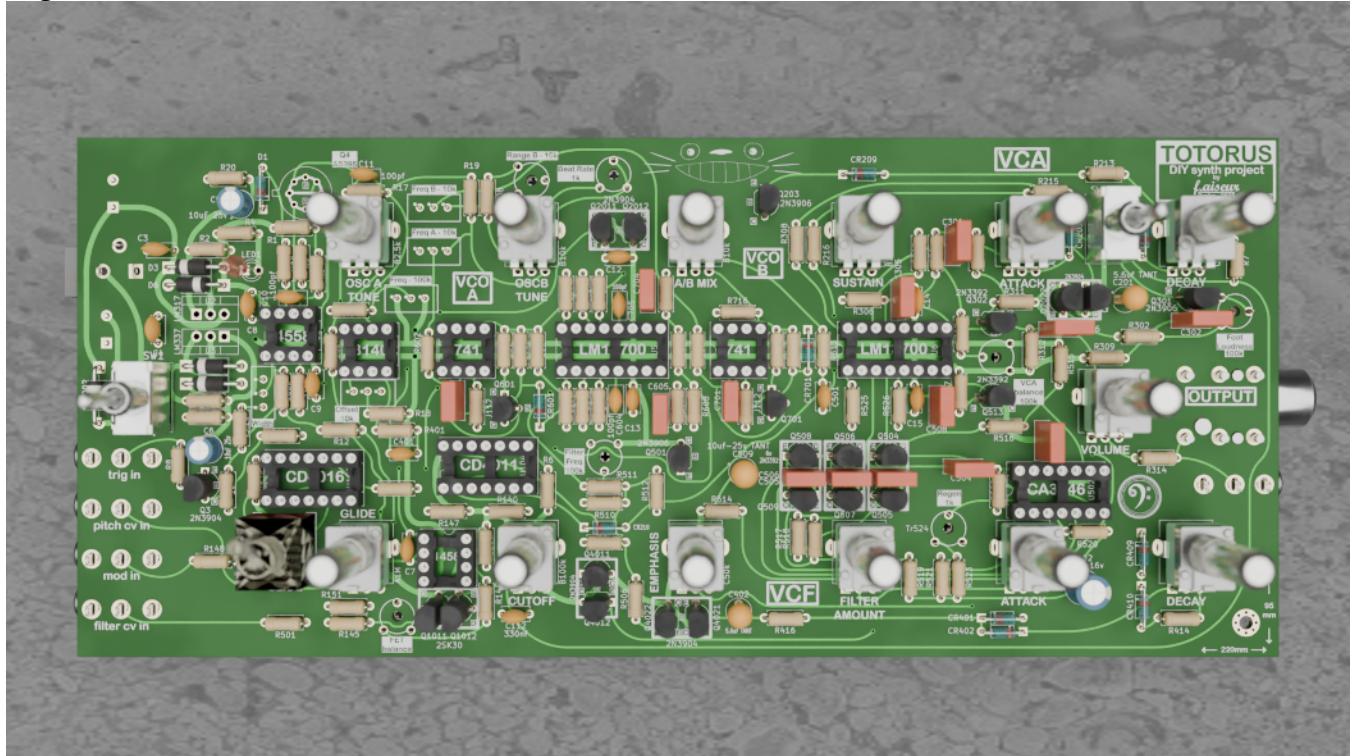


Now, flip the board and you can solder all trimmers and connectors. Align the connectors carefully.



NOTE : For the output connectors, I added an extra 3.5mm jack if you prefer to use those over the 6.35mm. You can install the two of course but I think you'd be able to use only one at once, because there is only one resistor on the output. One could tweak it, add another resistor to get different levels of output. I didn't try it so I can't comment.

Then, with the panel of your choice, you can install all the potentiometers and switches, carefully aligned.



Last, solder the Q4 transistor pair (LM394/AS394/AS194, etc etc), and install all ICs in their sockets. For Q4, you could try to use AS194 which is apparently an improved version of AS394, or a handmatched pair of NPN transistors, or MAT02, or..... I had an AS394 at hand so I didn't try the other options, but there are plenty!

**DONE!!!**

**CALIBRATION** : i wont go into details here, as you need to basically follow the instructions of the official Moog service manual.

Just the V/Oct part isnt covered there, obviously.

For this, i used a kind-of standard procedure to calibrate :

**SET-UP** : A/B mix fully CCW (only VCOA is playing). Filter fully open - CW. Sustain fully CW, attack fully CCW, decay switch ON and decay pot fully CW. Glide OFF.

- 1/ Place your oscilloscope - or frequency counter, or simple tuner - on the output.
- 2/ Play a note and adjust the VCO1 pitch potentiometer for a certain value. Note this value.
- 3/ Now play the same note one octave above, note the reading. Adjust the WIDTH trimpot (Tr4) so you'd get exactly double the frequency of the first reading.
- 4/ Play again the lower note, and eventually adjust the OFFSET trimpot (Tr6) to go back to the first reading.
- 5/ You might need to do 3/ and 4/ a few times before it's ok.
- 6/ you should have now a calibrated Log to Lin converter section, that should let you play in tune over a few octaves.

## CASE

I designed this pcb to be fitted in the Hammond 1455L2201 / 1455L2202.

So here are a few observations/pictures.

I used a standard sticking print, cut to size, and applied carefully on the top panel.

I never had such luck doing this before, but this time it got (twice) aligned pretty well.

The Hammond case has tight tolerances, but I recommend you to leave a bit of margin on the top and bottom of your printed panel, so you can slip the print between the top panel and body case, which will give you a nice feel/look, like it was here, on my first attempt :



The left and right sides will be covered by the plastic/alu ends so its ok if it doesn't look perfect there. But be careful when you slide the top panel in and out of the case, otherwise you might damage your print, and end up having not-so-nice look.

Probably some of you in this community are more skilled and imaginative than me, so if you have some suggestions please get in touch, but I will write down here what I did for my build.

So first, I printed the design. I should have used a better quality paper, but I wanted to keep things cheap for my prototype. Also, if you want to "slip" your print between the panel and the body, it might be complicated if your paper is thicker and more rigid.

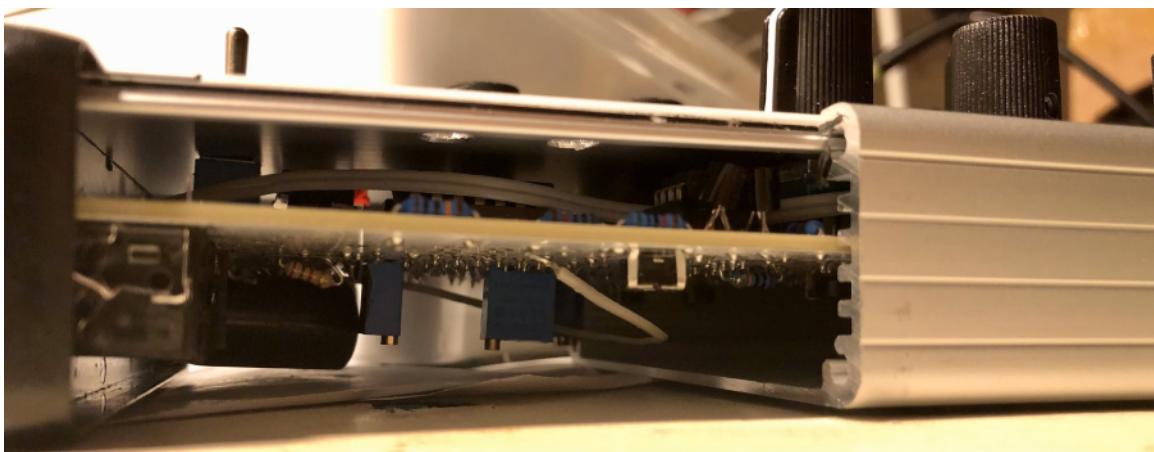
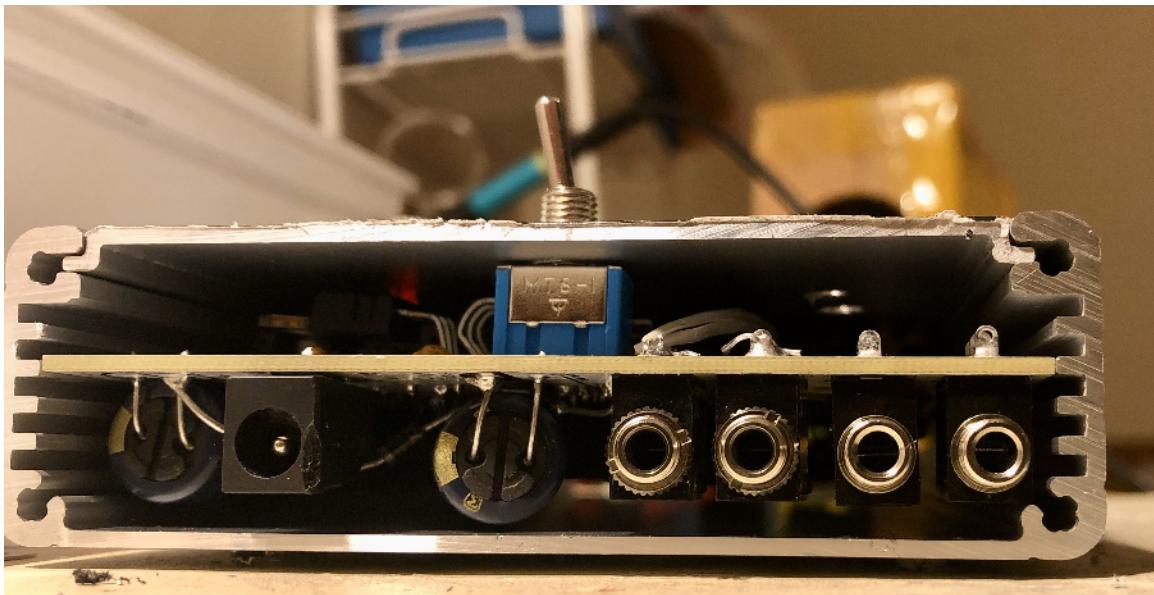
Second, you should make sure it's well aligned and that the PCB will be well centered. You can now cut all the holes and fit the top panel+print+PCB in the case.



For the sides, I also had a pretty amateur technique.

I put a bit of white paint on the (already soldered) connectors, to mark their position on the inner side of the end-sides. Then I could cut holes where they should be. Not perfect but good enough for me.





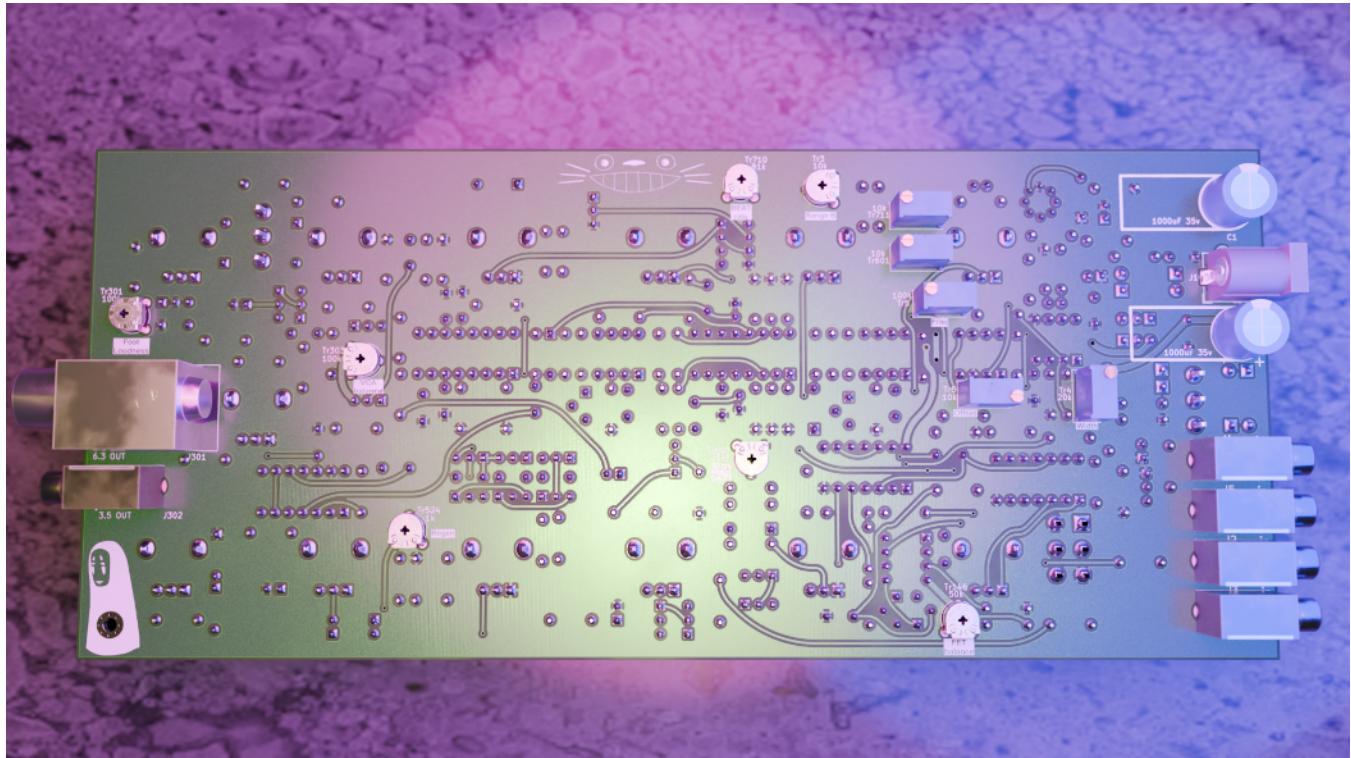
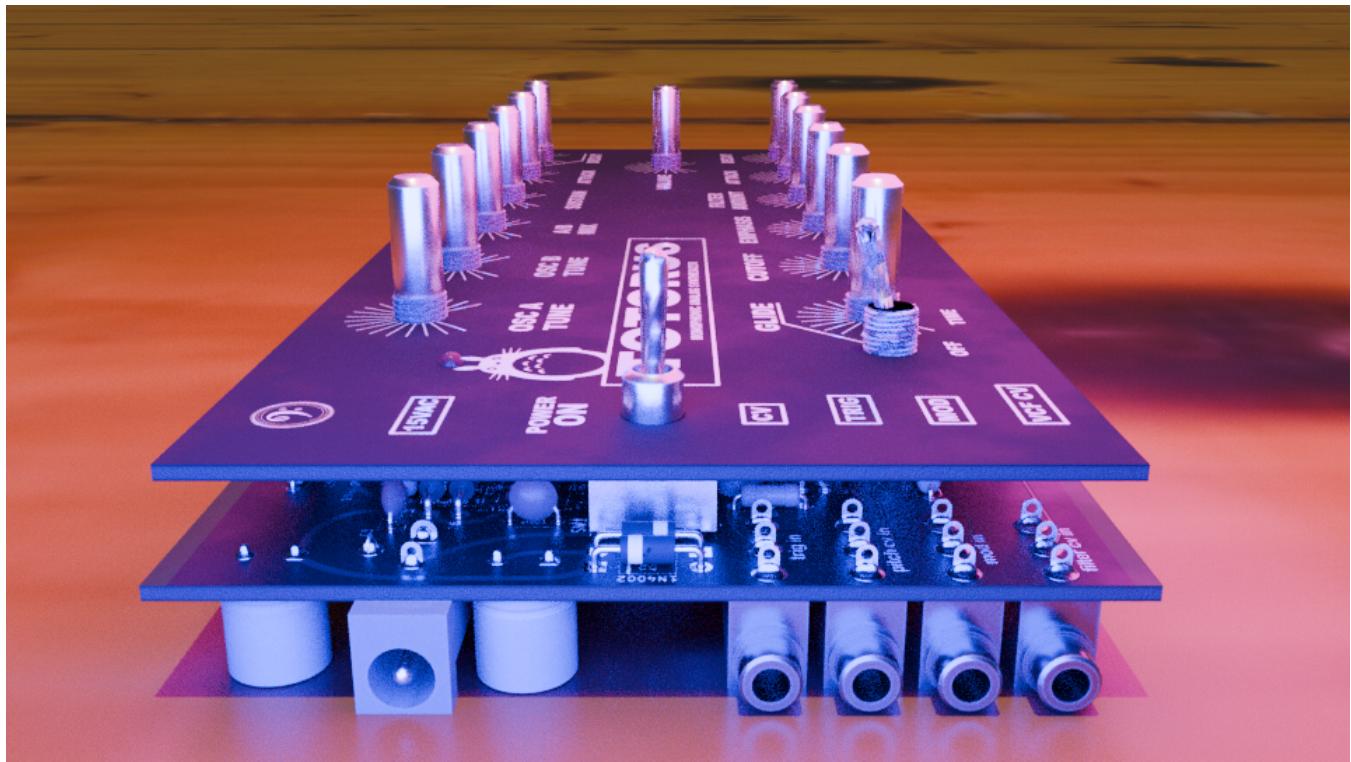


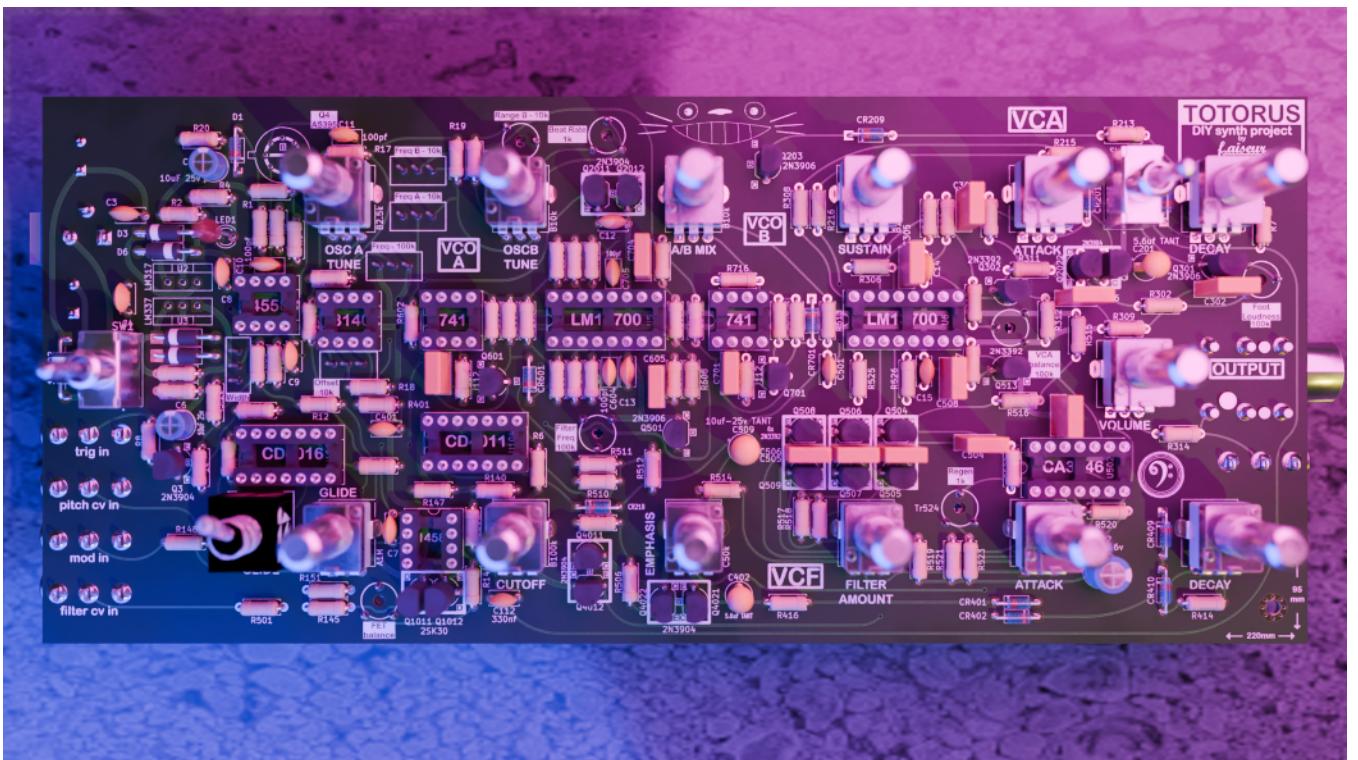
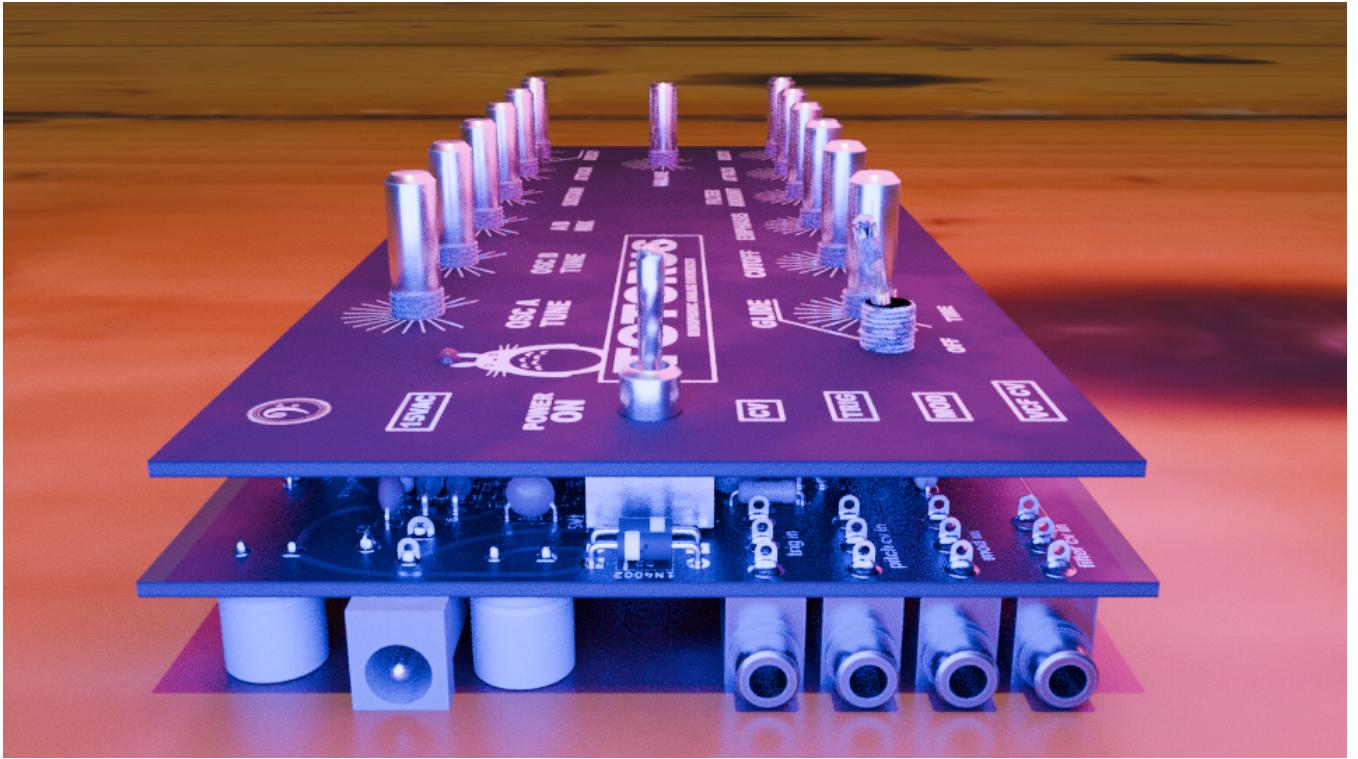
As you see, my first build was with white panel on natural alu, but I tried to have an ALL BLACK machine, so I spray painted the case and made another print panel in black.

I wouldnt recommend that method – or if you know how to paint! This 1455 case comes in a few different colors, so if you want some color I'd suggest you buy the case already factory-painted to the color of your choice. With the spray I used, the paint goes off, it got a bit thick and “ugly” in some spots – although on the next picture I tried to NOT show this, but you can imagine...



Enjoy your TOTORUS!!!





Links :

- original thread – in french :

<http://forum.anafrog.com/phpBB/viewtopic.php?f=36&t=15521>

- modwiggler thread :

<https://www.modwiggler.com/forum/viewtopic.php?t=291993>

- original documents :

<https://synthfool.com/docs/Moog/taurus1/>

For any feedback, comment, please send me an email to f.aiseur.tronics@gmail.com!