TB-EFA Build Guide

TB-303 Inspired Filter & Envelope & Amp

V1.6.2 September 2022

The TB-EFA shares the same 3 and a bit pole filter circuit as the Roland TB-303. It also provides a complete version of the envelope used by the 303 with decay control. Rounding off the module is the BA662 based VCA circuit which is also controlled by the same envelope as the filter.

Like the original it has pots to control the Cutoff Frequency, Resonance and Envelope Modulation and Decay. These are true to the original in their operation.

The envelope is controlled by a CV gate input, triggering the envelope and amplifier. If no gate is input you will not hear anything as the VCA will not be triggered.

In the filter, the TB-EFA adds an additional “chirp” control that can set the upper levels that the resonance feedback can achieve. R97A sets the highest level – by default a 4.7K resistor. You could go lower, but it gets almost too shrill!

The original TB-303 has an internal trimmer, TM3, that sets the filter frequency maximum. The TB-EFA externalises this with its own pot control so you can adjust to your taste on the fly.

A drive control is also added which provides an inbuilt overdrive effect at extreme, and again builders can try different levels. A 33k resistor at R62 combined with the A100K pot lets you drive it hard but not too much (original uses a fixed 220k) You can reduce the 33k down even as low as just a few k for extreme distortion. For example, a 3.3K works well! This drives the initial input audio signal to the filter – so pre filter overdrive.

The TB-EFA includes the VCA from the original 303. This uses the Open Music Labs (OML) BA662 “clown” clone circuit to stay true to the original, or the alternative BA6110. This VCA is controlled by the same envelope as the filter and only the decay portion can be adjusted.

The final control is accent. Here, a trigger CV input is provided as well as a pot to set the accent level. Note that the accent controls the envelope shape and level, so it needs the envelope to be triggered. Thus, both gate and accent need to be “high” for it to affect the sound – or at least the envelope needs to be running for accent to increase its effect. The accent also adjusts the bias on the BA662 circuit and increases the loudness of the note, so without a trigger you can increase the general accent tone of all notes with just the pot.

The TB-EFA is available as a panel and pcb set, a panel and pcb set with partial kit (transistors and BA662 clown) or in very limited numbers as a pre-built module.

The build is simple if you’ve built other Eurorack modules and you know what you are doing, read the important notes below and then proceed in your preferred build order.

Contact info geosync.synth@gmail.com

!!! Important Notes !!!

During the time Roland were making 303's they basically used whatever they had in stock for the 3 pin transistors, real machines have any of 2SC536F, 2SC945P and 2SC1815G. The most common is the 536, but any ECB pinout NPN will work just fine. These aren't particularly important to the sound, I've experimented with all of these and NOS/new parts, including modern variants of the 945P (marked P331) and all of them work.

The PNP is usually a 2SA733P, which are not really made any more either, you can find some NOS ones and Q variants are fine too. The equivalent matching PNP to the NPN you are using is fine also, so 2SA608, 2SA733 or 2SA1015 all work just fine. Even things like the 2SA1115 and 2SC2603 used in other Roland products of that era will be suitable.

Because the 1980's dual transistor pairs are getting harder to find, or are often fakes, the 2SC1583F/G and 2SC2291 can be replaced with a pair of matched 2SC945P (or equivalent). Again you can match these or just take a gamble and pair some up.

To make it easier I have included both a 5pin footprint for each (Q12,Q21,Q22) AND a pair of 3 pin footprints (Q12A,Q12B etc)where you can substitute for two matched 3pin NPNs – whatever you are using for the other NPNs will be fine. If you bought a partial kit, it will be the A/B pairs you install.

ONLY INSTALL COMPONENTS IN ONE FOOTPRINT, i.e. either Q12, OR Q12A+Q12B

The 1583 is common emitter, so you end up with BCECB as the 5 pins, or BCEECB for the pair of 3 pins, with the middle E's connected via the board.

The 2291 is common base, so you end up with ECBCE as the 5 pins, or ECBBCE for the pair of 3 pins with the middle B's connected via the board.

You can of course make a 5 pin also, and I've included some pictures of how I find it easiest to do this in the appendix.

From rev 1.6.2 of the PCB, dual footprints are also added for IC1. This means you can use EITHER a BA662 or a BA6110 – these two are almost identical but have different pin out so two footprints are needed. Your partial kit will include one of these, make sure you use the correct footprint for your IC. BA662 = IC 1A, BA6110 = IC 1B. Boards earlier than 1.6.3 only have the BA662 footprint.

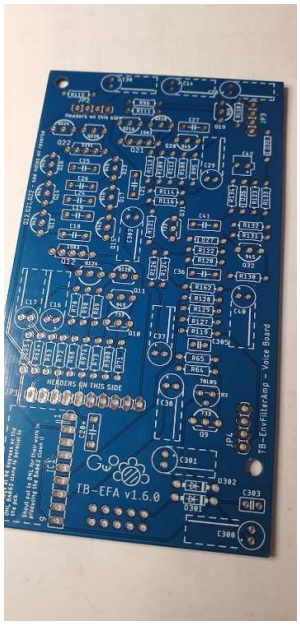
ONLY INSTALL THE IC IN ONE FOOTPRINT, i.e either IC1A or IC1B – follow the BOM.

Note on electrolytics. I've tried to make it possible to use normal 11mm capacitors, so you will see a marking on the silkscreen showing where you can lay the capacitor on its side. The solid white line designates the side where the -ve stripe should end up. You can of course use <11mm high caps and stand them up, but if you are like me I have loads of 11mm normally.

Enjoy!

Lower – Voice Board

Start with the board marked as the voice board.



1. Solder the 1N4148 diodes.
2. Solder all the resistors.

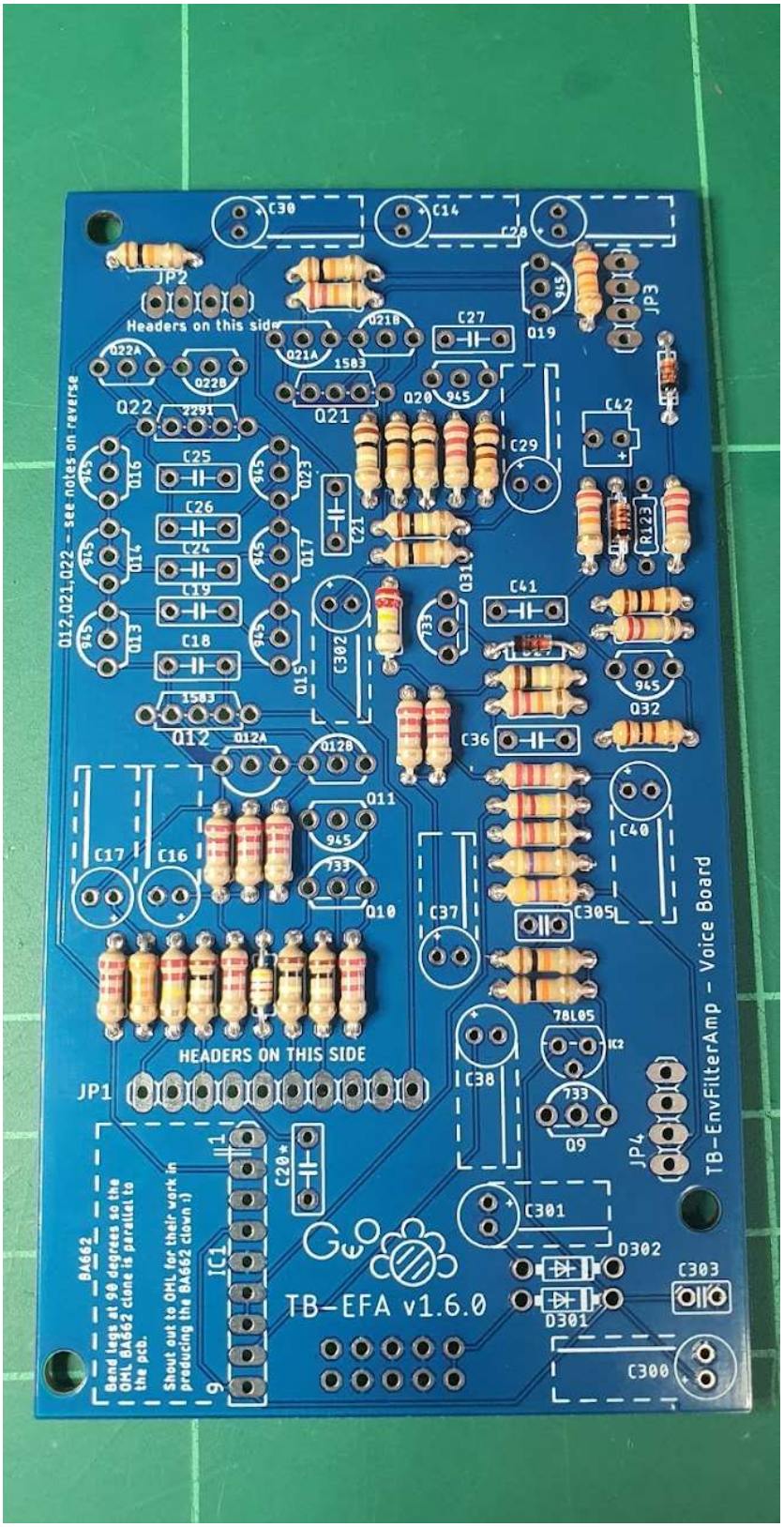
   Keep some of the resistor legs, we will need them for the BA662 clown.

   Note that there is some method in the madness that is the component numbering. Where the analogue voice parts are directly corresponding with the TB-303 circuit, they maintain the original machine part numbers, typically diodes, resistors, capacitors and transistors.

   Where the parts are specific to this module, on the lower board they are numbered starting at 300 and 400 for the upper board.

   In the BOM the original parts that live on the upper Control Board are in blue text.

   The board should look something like this; some pictures are of prototype boards and may differ slightly.


C30
C14
C28
JP2
Headers on this side
Q21B
Q21A
1583
C27
Q19
945
JP3
Q22A
Q22B
Q22
2291
Q21
Q20
945
C42
C29
C25
C26
C24
C19
C18
Q16
Q14
Q13
Q23
Q17
Q15
C21
C302
Q31
733
C41
R123
Q12,Q21,Q22 - see notes on reverse
Q12
Q12A
Q12B
C36
Q32
945
C40
Q11
945
C17
C16
733
Q10
C37
C305
78L05
IC2
C38
733
Q9
HEADERS ON THIS SIDE
JP1
JP4
TB-EnvFilterAmp - Voice Board
C20*
C301
D302
C303
D301
C300
TB-EFA v1.6.0
IC1
BA662
Bend legs at 90 degrees so the OHL BA662 clone is parallel to the pcb.
Shout out to OHL for their work in producing the BA662 clone :)

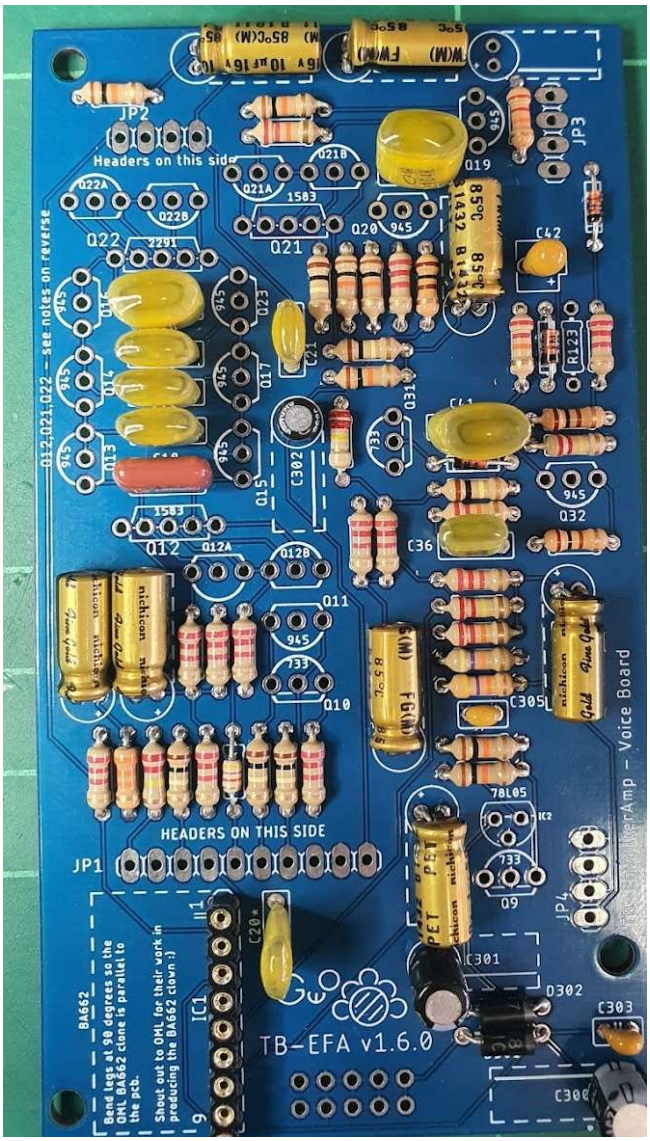
3. Solder the two 1N5817 diodes.
4. Next comes the capacitors, again my routine is usually;

    a. MLCC small ceramics and tantalum.
    b. Polyester/polypropylene yellow/greenies
    c. Finally, the electrolytics.

    Due to the stacking of the boards and the normal 11mm clearance between them, you cannot stand up the electrolytics if they are standard 11mm themselves. If you have low profile electrolytics, all good, but otherwise insert the caps and bend them over to sit parallel to the board in the space marked in the silkscreen.

    The solid white line marks the side where the -ve marking should end up.

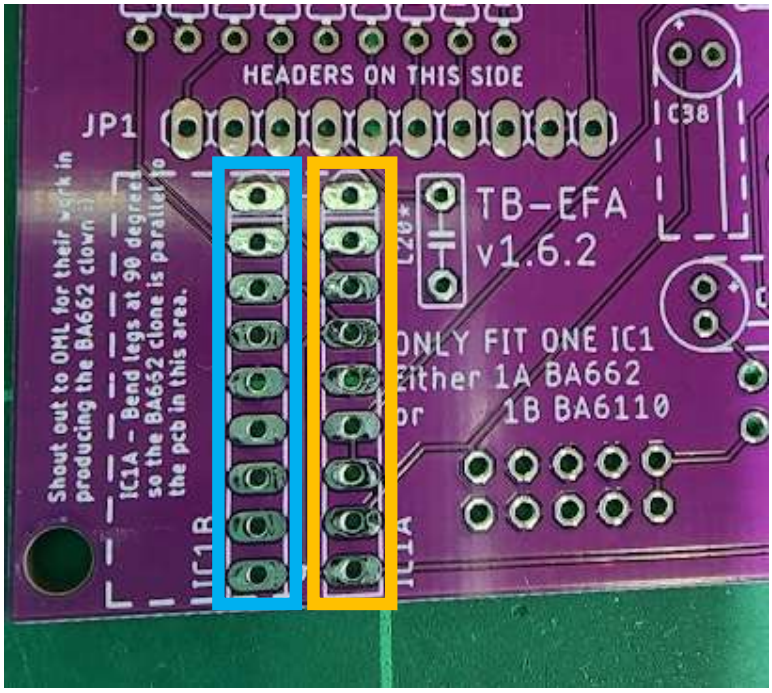
    [Don't solder the 9 pin SIP socket for the BA662 yet, we may need it to add legs to the clown]



5. If your kit comes with the BA662, it will be connected to a 9pin sip socket. You can carefully remove the BA662 from the socket and solder in location IC 1A (yellow box below) on the PCB.

   DO NOT FIT THE BA662 into the socket yet. Just solder the socket to the board.

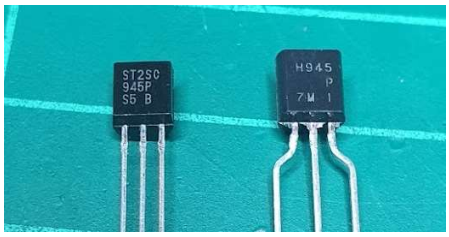
   If your kit came with a BA6110, use location IC 1B (blue box below) and solder the BA6110 directly to the board in this location now. Pin1 (notch) it at the top of this picture.



6. Now we can add the transistors.

   Q12,Q21,Q22, – here you will either have a 5pin 2SC1583 & 2SC2291 – in which case install in the 5 pin spaces (orientation does not matter) – or if you are using 2x 3pin NPN such as 2SC945P then install two in QxxA and QxxB

   *** If you purchased the partial kit, you may have 21x NPN all the same, in which case you can use any of them in these dual positions. You may have 15x of the same NPN style, and possibly 6 different 945 then use the 6 different ones in these dual positions.



These maybe marked either marked “ST 2SC945P” or “H945P” these are pre-matched and have the hfe levels best suited to the dual transistors. So check which ones you have just 6 of, the others should be used in the other 945 slots. i.e. the ones you have 15x (945,1815,536 etc)

Batches of kits vary depending on what stock of transistors we have at the time – at the end of the day it doesn’t make much of a difference ***

DO NOT POPULATE ALL 3, if Q26 is used Q26A+Q26B should be empty, and vice versa.

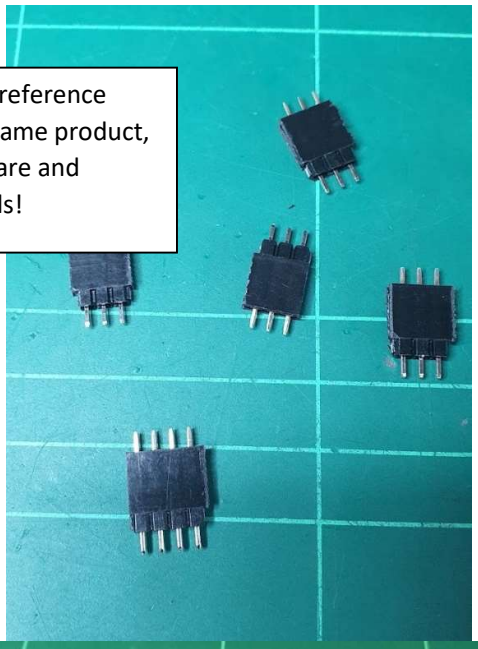
7. Add the voltage regulator IC2.
8. Add the Euro 10pin header and the 3pin trimmer **on the reverse side of the board.**
9. Finally, the headers to join the two boards.

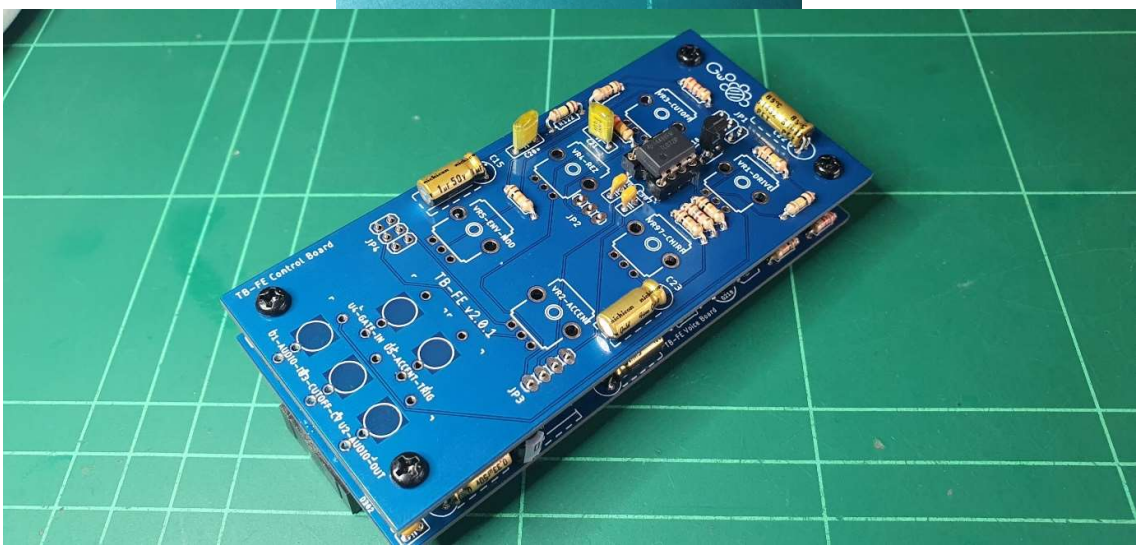
Use your usual technique for this but I find the following makes it easy.

a. Screw the three 11mm standoffs to the voice board. The standoff should be on the same side as the components, so screw from the underside.
b. Cut / prepare both the male and female connectors and join them together.
c. Place all 4 (JP1 to JP4) into the voice board sticking up the same way as the standoffs. **Don’t solder yet.**
d. Bring the control board to the stack and place it carefully so the standoffs line up with the holes and most importantly so that all the connector pins go through the corresponding holes in the control board.
e. Screw the control board to the standoffs and you should now have a nice, neat sandwich and you can simple solder all the connector pins from the top and bottom.
f. Unscrew one of the boards from the standoffs and carefully pull apart the connectors. Perfectly lined up every time!

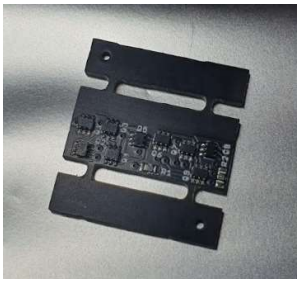
(NB: I usually leave the standoffs attached to the upper board, so unscrew the bottom board. This is because I am lazy and if they are already attached to the upper board, you can fit the panel later and not have to worry about access to the screw holes!)

These pictures are for reference only they are not the same product, but show how to prepare and connect the two boards!





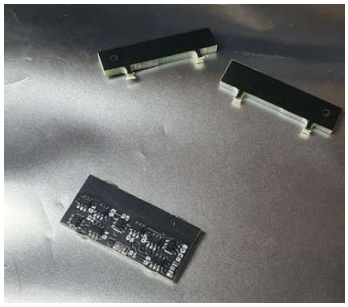
BA662 Clown



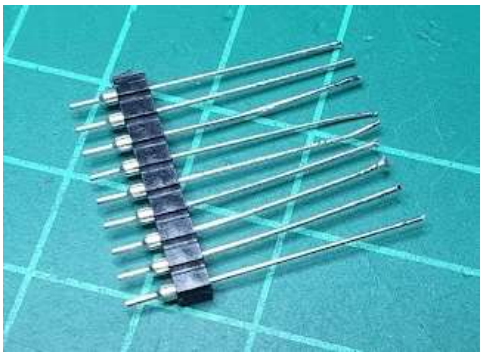
The BA662 clown may already have 9 legs soldered to it – some are pre-built to test a random sample from the fab.

But if yours looks like this we need to do a few things.

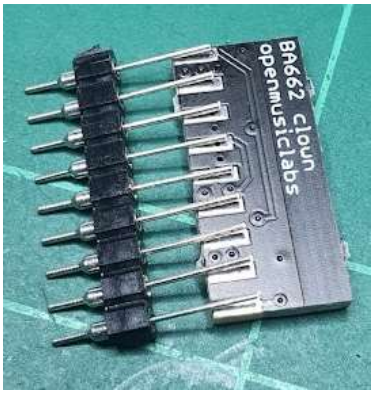
1. Cut the fab support bars off as close to the circuit board, but don't damage the main board. You can file the rough edges if you want but it's generally not necessary.

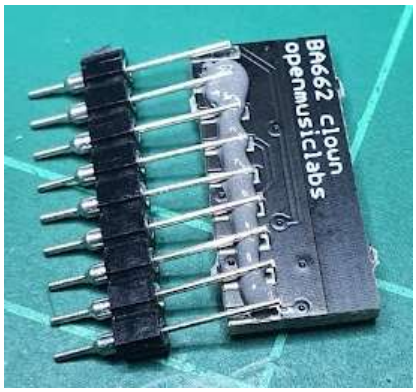


2. Take the 9 pin SIP socket and insert a resistor leg into each pin, make sure they are fully inserted and trim them to about 1cm.

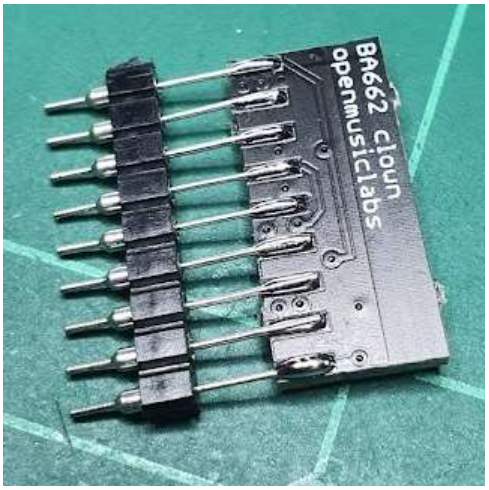


3. We are basically using the SIP socket to help line up the legs we are going to solder to the BA662 clone. I like to use solder paste here, it helps hold the legs in place while you solder, but normal solder will be fine, just take care to keep the legs as neatly aligned as you can, and careful not to overheat and lift the pads on the pcb.

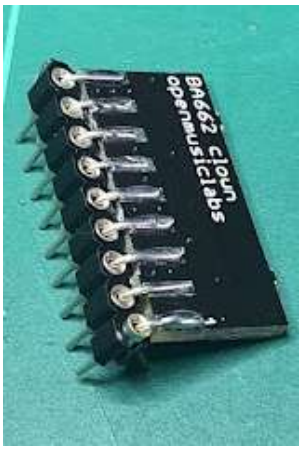


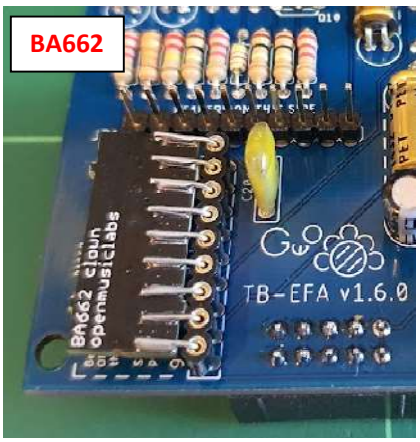


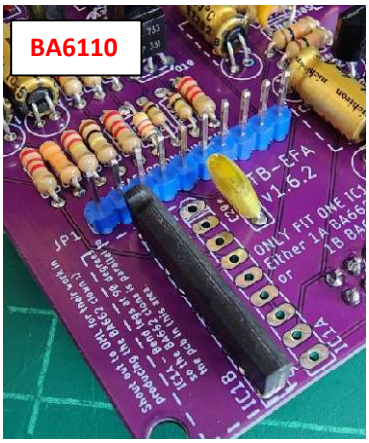
4. Clean up the flux residue and you should have something like this



5. Final step is to trim the legs to size. We want to be able to bend the legs at 90 degrees to the socket. You need enough length to insert into the SIP and a little extra so you can bend it over and not foul on the SIP itself. Something around 6-7mm is about right. It maybe easier to first solder the SIP to the lower PCB at position IC1, then insert the clone and bend over. This way you can trim a little more off if needed. Pin 1 is marked (bottom of picture here) and is designated on the lower PCB with a curved end.







REMEMBER rev 1.6.2 onwards has the footprint for either a BA662 like this, or a BA6110, be sure to use the correct footprint for the IC type included in your kit.

BA662 = IC1 or IC1A

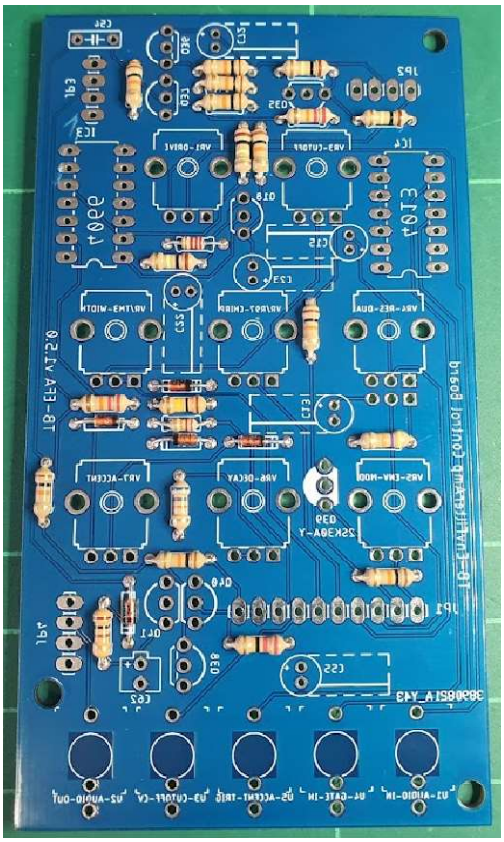
BA6110 = IC1B

For the BA6110 we recommend not use the pin strip socket and soldering the IC directly to the PCB.

Upper - Control Board

Repeat the process for building the upper control board.

1. Add the 1N4148 diodes.
2. Add the resistors.

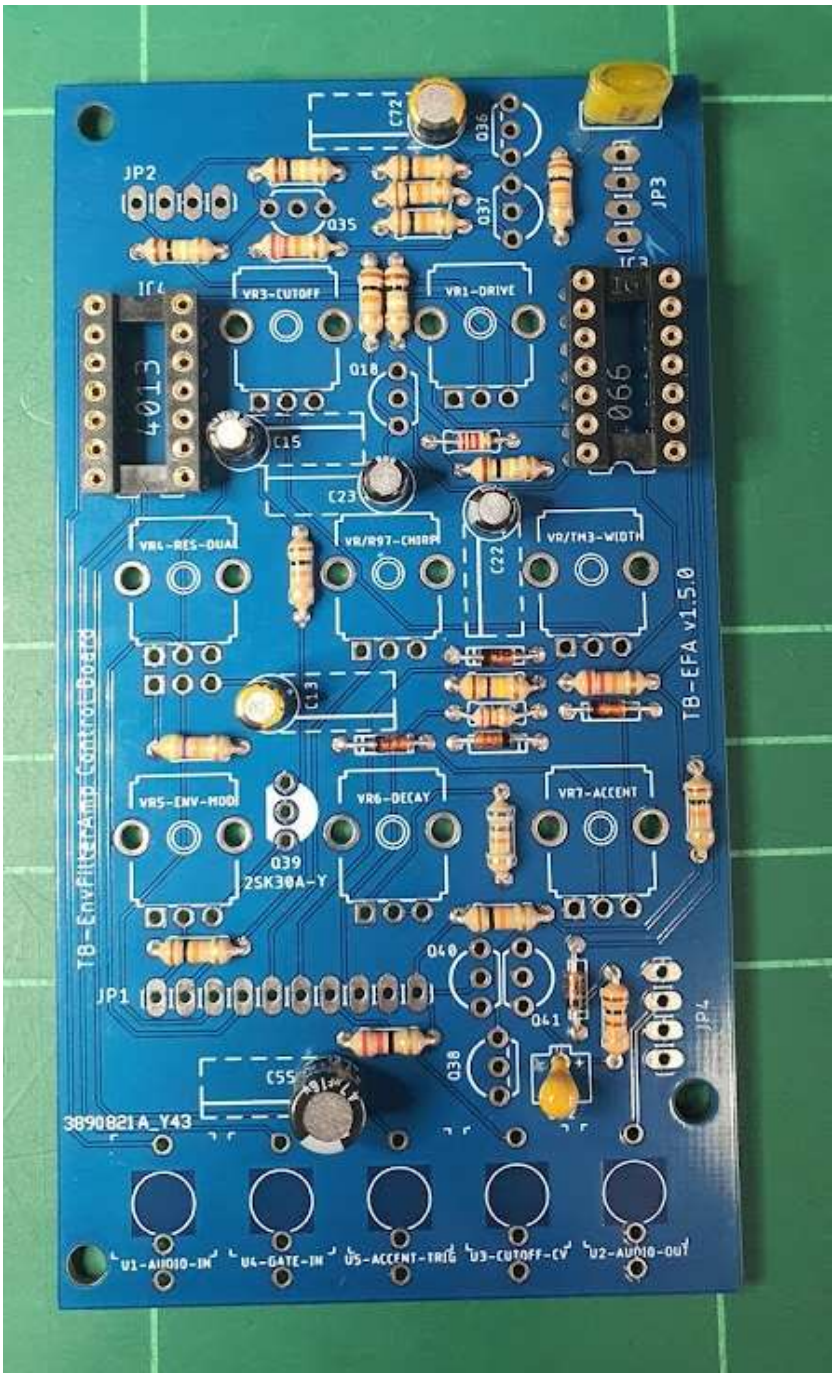


3. Add the two DIP14 IC sockets if using them.
4. Add the capacitors, again my routine is usually;

    1. MLCC small ceramics and tantalum.
    2. Polyester/polypropylene yellow/greenies
    3. Finally, the electrolytics.

    Due to the panel being above this board, with the normal 11mm clearance between them, you cannot stand up the electrolytics if they are standard 11mm themselves. If you have low profile electrolytics, all good, but otherwise insert the caps and bend them over to sit parallel to the board in the space marked in the silkscreen.

    The solid white line marks the side where the -ve marking should end up.



5. Add the transistors and insert the 4013 and 4066 into their respective sockets.

6. Finally insert the 5x thonkiconns, you may need to trim the corners of the first and last jack sockets to fit flush around your standoff screws, and the 8x 9mm pots. **Do not solder them yet**. Add the panel over the top and tighten the nuts on all the hardware. This way you get everything nicely aligned and only then should you solder the jacks and pots in place.

7. Connect the two boards together and screw the standoffs in place.

Simple Checks - optional

If you want to check everything is OK, you can check the power rails before fitting the ICs on the lower board. So disconnect the upper board and just power the lower board on its own without the BA662 clone inserted.

You should get a nice steady +12v on pin 4 and pin 9 of the BA662 socket.

You should get +5 on pin8 of JP1. You should also see +12v on pin10 of JP1 – (pin 10 is towards the middle of the board, pin1 is closest to left edge)

Connecting together again, without the panel on, with a scope you can check the third pin of the resonance pot shows a < ~500mV smooth waveform, and as you change the cutoff and resonance it adds in an oscillation to the waveform.


TB-EFA
TB-303 Inspired
VCF
ENV
VCA
Drive
Width
Gate
Accent
Cutoff
Out

Calibration

No calibration is required.

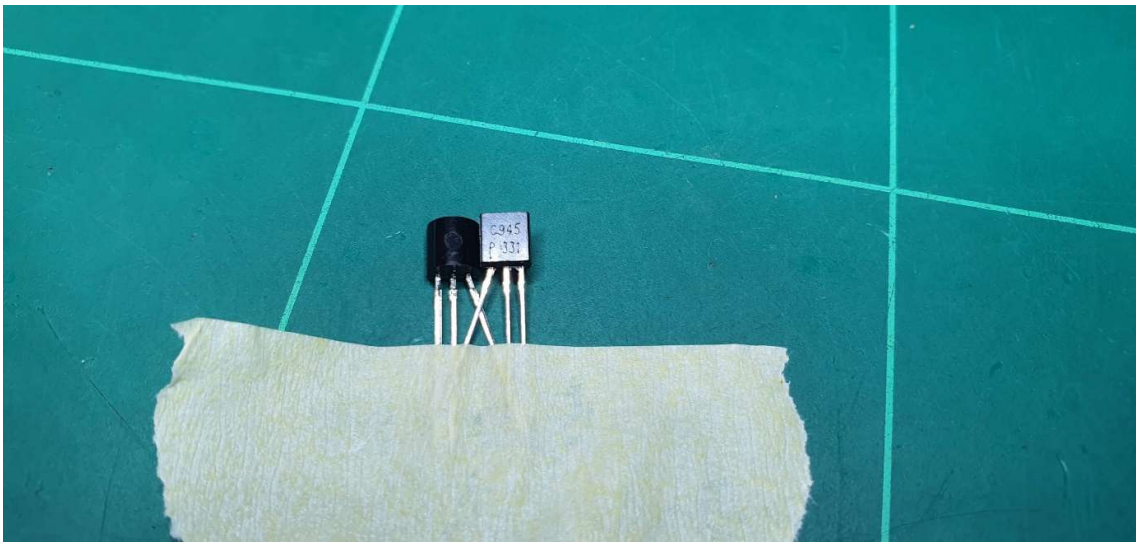
Appendix A : Making the Dual Transisors

Easy way to build a common emitter dual transistor.

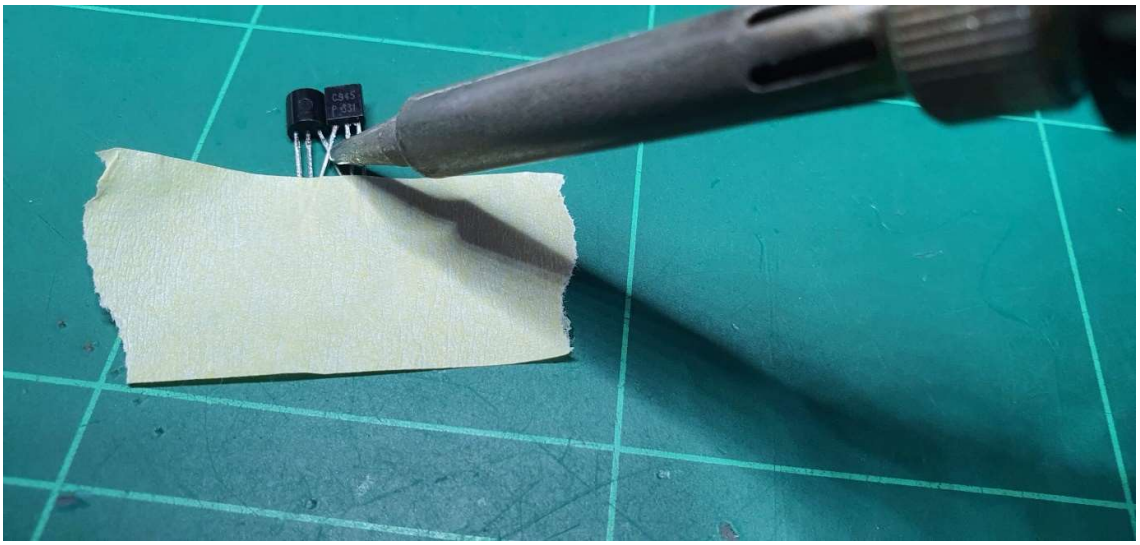
The board has footprints to install two 3 pin, so this is only needed if you want to use the 5 pin footprint – there will be no sonic difference.

Start with you two NPN transistors, left hand one facing down, right hand one facing up and gently bend the emitter legs at an angle as shown.

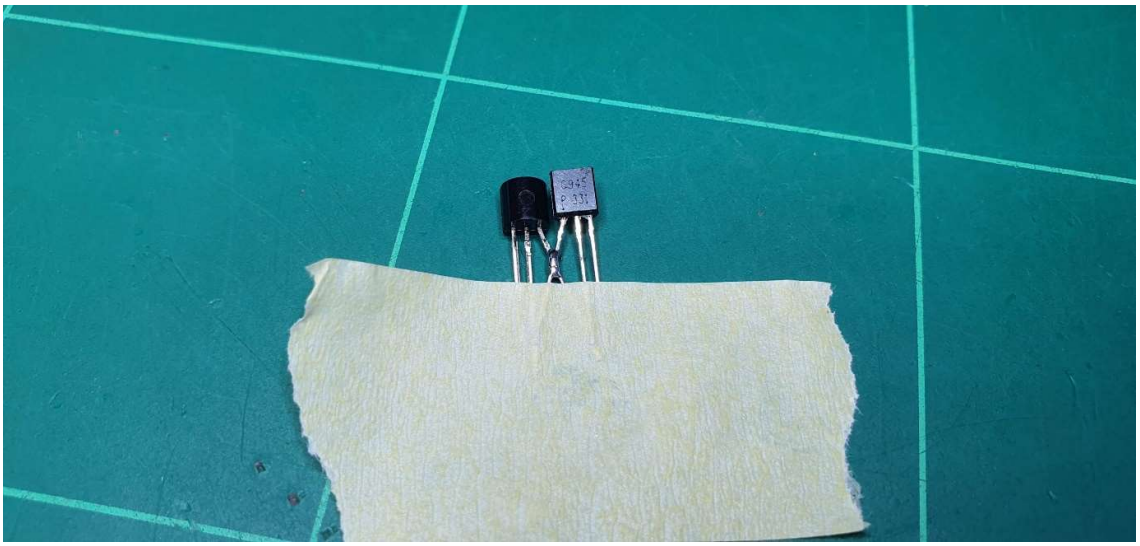
Next place the two legs across each other and tape the lower edges to hold in place while you solder.



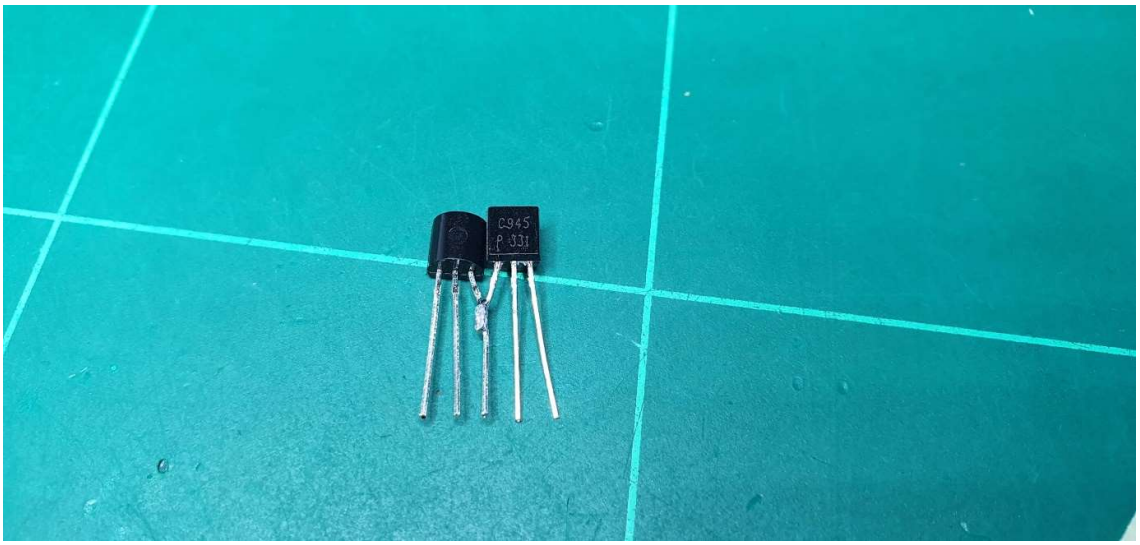
As shown there is a little ledge you can rest the soldering iron tip against to warm the legs and then dab with a small bit of solder.



The solder should flow nicely around the join and you end up with :



Finally, carefully cut off one of the legs at the join and bend the remaining leg straight.



The images show a common emitter, so a 2SC1583.

To build a common base dual transistor, such as the 2SC2291, simply reverse the two transistors before soldering, i.e. left hand faces up, right hand faces down.