GS1-F Build Guide
BA662 Inspired VCF

Vx.2.1 August 2022

The GS1-F is based off the design of Roland's System 100M series of modular synth modules. The original design uses four BA662 differential op-amps, one for each pole of the LPF.

The BA662 is no longer available, but Open Music Labs created a clone (clown) design that is pin compatible for the original. It uses 9x tiny dual transistors and a couple of resistors and is perfect as a replacement for the original. The BA6110 IC is still available if you look hard enough, and while not being pin compatible is more or less identical to the BA662 for use in these modules. For this reason, the GS1-F provides two voice boards, one for each footprint of each IC type.

To be clear, you only need to build one of the voice boards, either the BA662 version or the BA6110 version – these are clearly marked on the underside of the board. There just wasn't enough room in 8hp to have both footprints on the same PCB.

The partial kit comes with 4x BA6110. The builder is welcome to source a BA662 or OML clone from places like re303shop.com or SynthCube. However we could tell zero difference…

The module provides three audio inputs that are mixed (and gain set) via the three input jacks/pots at the top. After that the audio is sent to the Low Pass 4-pole Filter. An additional simple High Pass Filter is included that is selectable, either on or off, via the toggle switch. The LPF has the usual cutoff and resonance controls and will self resonate at full resonance. This can be tune and so is playable via V/Oct input to the CV modulation input marked "key".

The Filter cutoff can be controlled by up to three CV attenuatable inputs, these are mixed before being applied to the filter as one CV.

Contact info geosync.synth@gmail.com

!!! Important Notes !!!

During the time Roland were making late 70's and early 80's analog synths, it seems they basically used whatever they had in stock for the 3 pin transistors. By default, in the System 100 timeframe, the 2SA1015GR and 2SA1815GR seem to have been the PNP and NPN of choice, but any ECB pinout PNP, NPN will work just fine. For example 733/945 or 608/536 or even 115/603 etc These aren't particularly important to the sound, I've experimented with all of these and NOS/new parts, including cheapo Aliexpress variants of the 945P (marked P331) and all of them work.

DON'T POPULATE BOTH THE BA662 AND BA6110 voice boards, pick one and build that only.

Note on electrolytics. I've tried to make it possible to use normal 11mm capacitors, so you will often 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

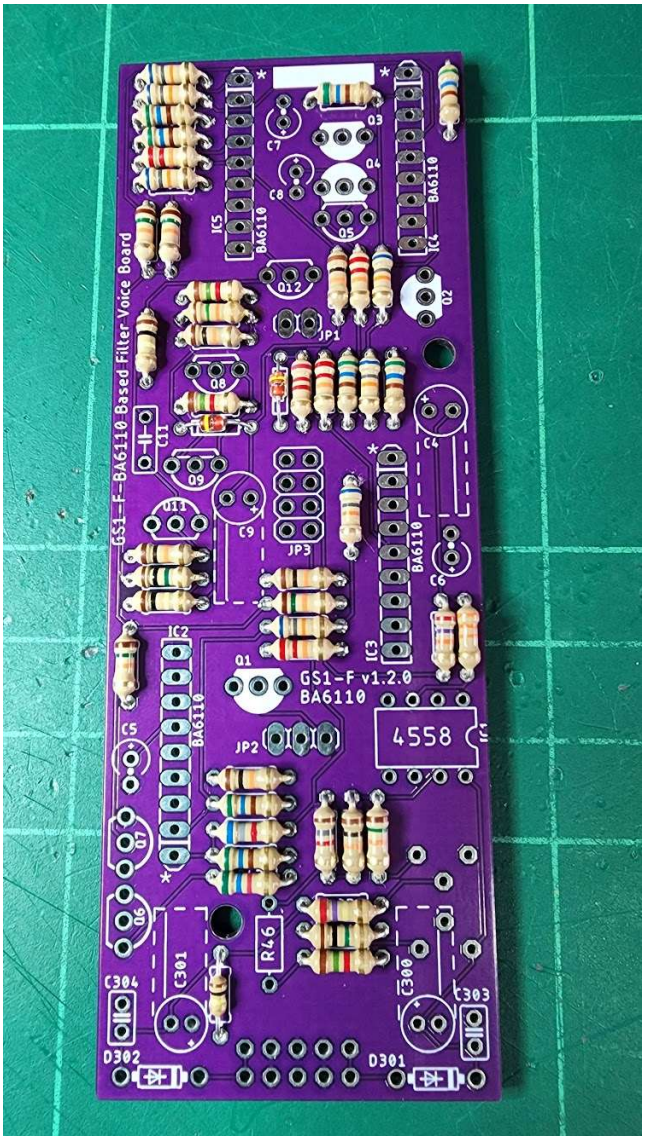
Start with the board marked as the voice board.

1. Solder the two 1SS133M diodes.
2. Solder all the resistors.

   Parts on the upper control board are marked in blue text in the BOM.

   For now, just look for and solder all the resistors that are shown with black text.

   The board should look something like this; some pictures are of prototype boards and may differ slightly. All build pictures show the BA6110 version, the BA662 board differs slightly in component positioning.



3. Solder the two 1N5817 diodes.
4. I like to add the IC sockets next, or solder the ICs in place if you are not using sockets.

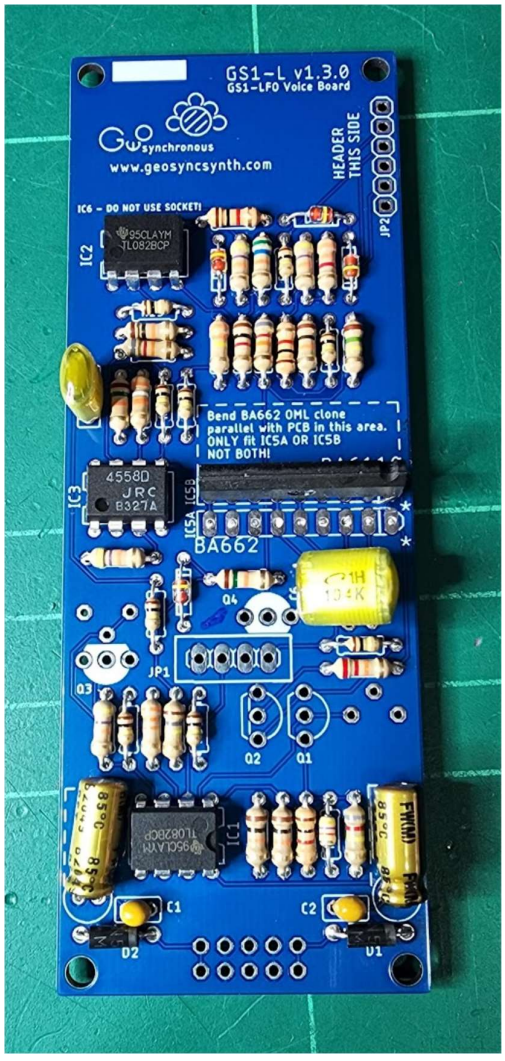
   DO NOT USE A SOCKET FOR BAxxx IC's there isn't enough room for the IC when installed in a socket..

5. Next comes the capacitors, again my routine is usually;

   a. MLCC small ceramics
   b. Polyester/polypropylene yellow/greenies and the poly box ones
   c. The electrolytics.

      Due to the stacking of the boards you cannot stand up the electrolytics if they are standard 11mm. If you have low profile electrolytics, all good, but otherwise insert the caps and bend them over to sit parallel to the board as shown. The white line in the silkscreen should match the -ve marking on the caps.

   d. Leave the polystyrene till after you have fitted the ICs.

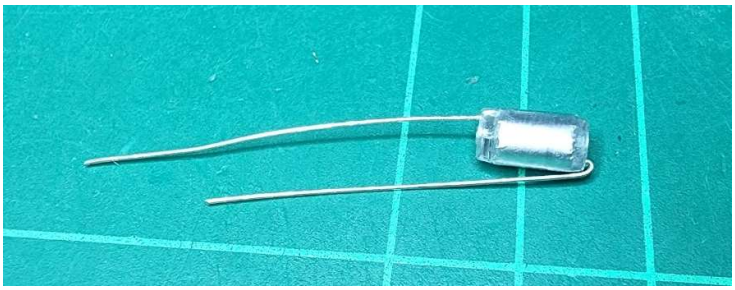


6. Now we can add the transistors.
7. Add the 4558 IC into the socket if you used one and haven't already.

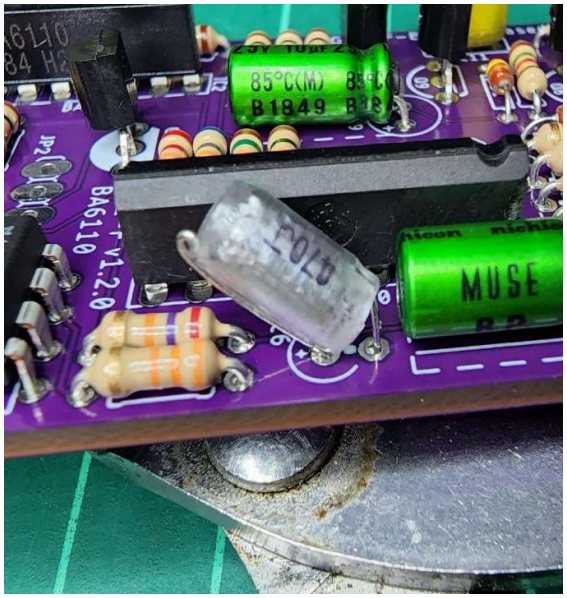
8. We are just left with a few parts, the four 470pF polystyrene caps and the NTC.

   For the styrene caps, the silkscreen shows a footprint, these caps are not polarised and so you can insert them either way.

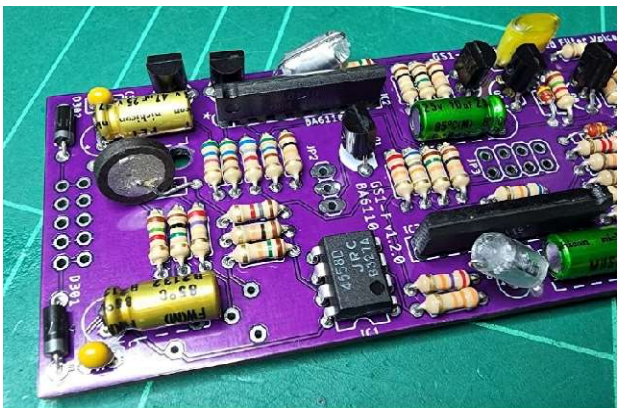
   To add the caps, bend them as shown and you may need to install them at a slight angle due to the clearance between the boards, as shown.



   Note around C6 that the space is tight and be sure not to have the styrene leg short against the end of the electrolytic. Here I usually insert opposite to the silkscreen.

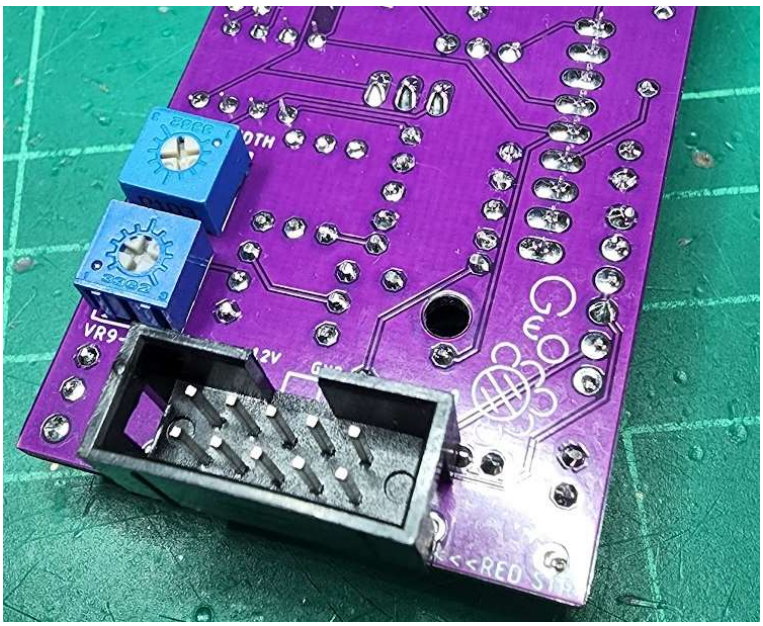


9. The last part on the top here is the NTC, bend the legs gently apart so that it will go into the holes easily and sit more or les flush with the PCB.



10. Add the Euro 10pin header and the 3pin trimmers **on the reverse side of the board.**

    You will need to move C300 out of the way while you solder the trimmers from the top. Push it back flush after snipping the wires.

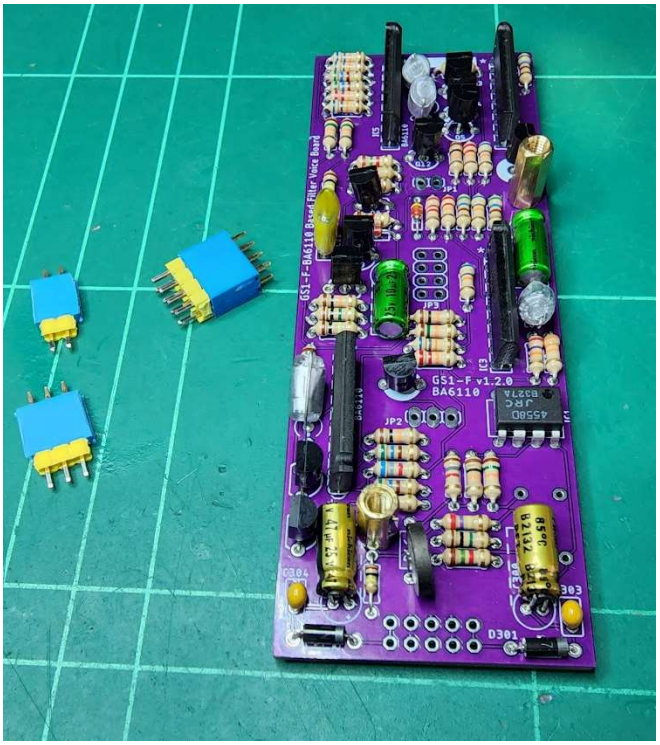


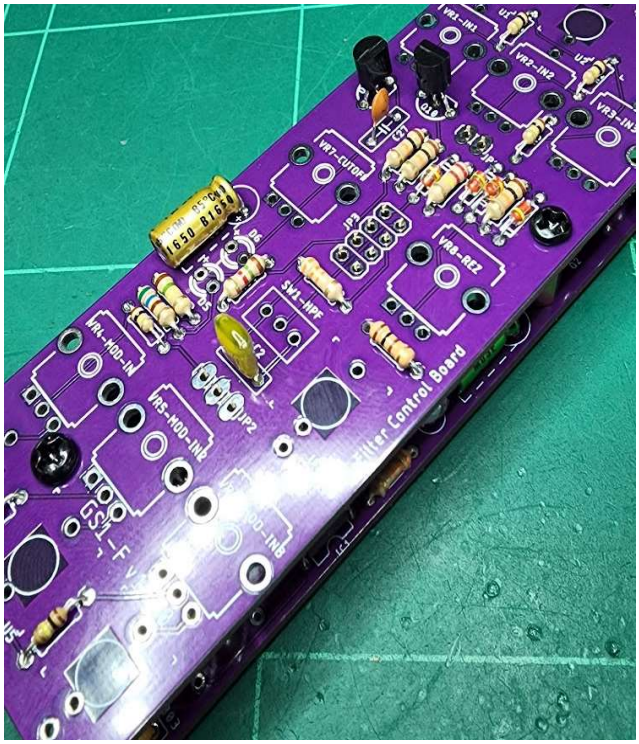
11. Finally, the headers to join the two boards.

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

    a. Screw the 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 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 simply 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!)





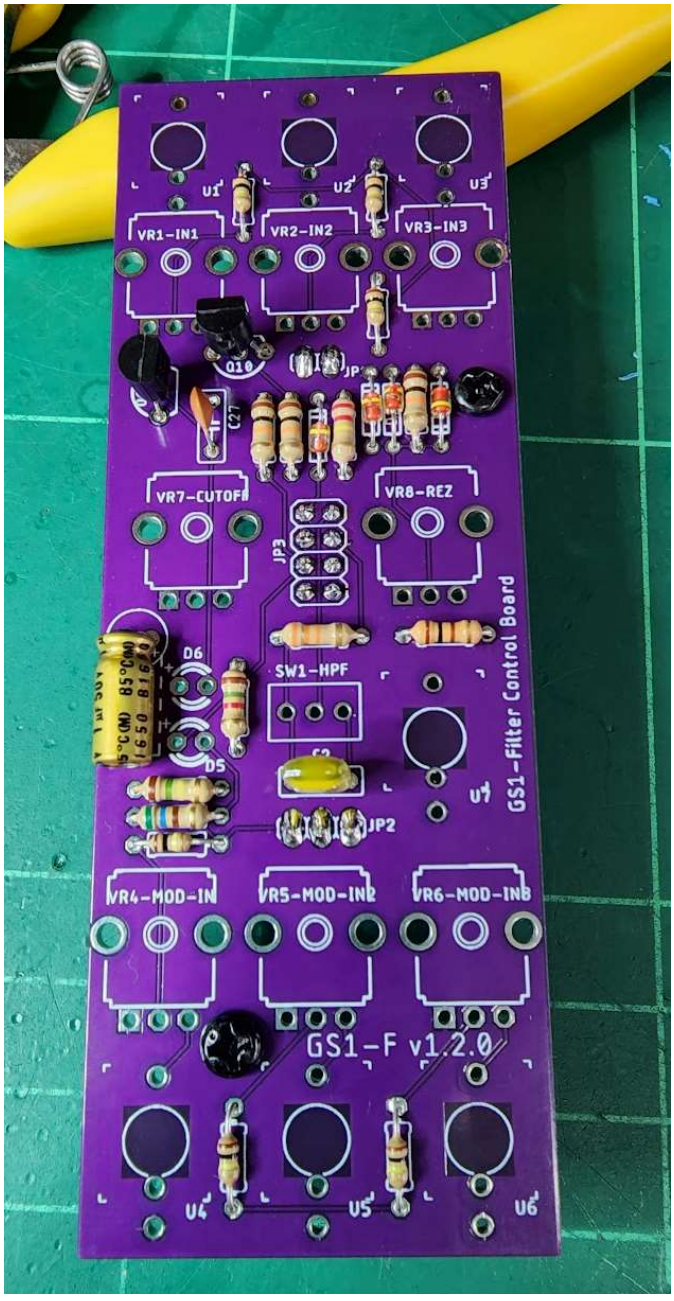
Upper - Control Board

Complete the same order of components on the control board:

Diodes, Resistors, Capacitors and Transistors.

Note C27 can be any capacitor, its just the buffer for the LED, so a cheapy ceramic is fine.

C2 is the HPF capacitor, 2.2nF is middle ground, for less filtering a 4.7nF is good. You can choose.



Insert **but do not solder yet** the pots, toggle switch and the jack sockets. Add the LED and place the front panel over them so you can ensure they line up with the holes.

Here, as I don't want to have to take all the nuts off again I make sure I have the standoffs attached to the upper control board, fit all the jack nuts, washers and nuts to the pots.

For the toggle switches, the easiest is to leave one nut on the switch body, screwed all the way down. This sets the correct height. Then use the second nut to fix to the panel.

Once you are happy everything fits and is lined up, solder them all from the underside.

Check that none of the pots or jacks foul against the components on the lower board, you may need to trim the solder joints a little, for example around Audio In Jack2 above the styrene caps.

Connect the two boards, check everything over and time to power on.

Calibration

The calibration is pretty simple but you will need a scope.

The only thing we need to calibrate is the self resonating sine wave that the filter produces.

With nothing plugged into the inputs, set the cutoff and resonance to maximum. Scope the output and use the frequency trimmer to set 20Hz (50us wavelength) – this is from the original Roland service manual… however…

Once you've done that, you can attach a CV V/Oct source to the "key" CV modulation input and turn it up a little. Set resonance to maximum so it self-resonates and set the cutoff to minimum.

Play C1 and C2 and adjust the width trimmer so that C2 is twice the frequency of C1.

One you have done this, you can tune the sine wave, the lowest I could get it, is around 130.8 Hz for C0, so its going to be a couple of octaves higher than normal, but at least then you can send it your VCO's V/Oct and have it play in tune, all be it a few octaves higher.

Enjoy!