GS1-E Build Guide

Sys100 Inspired Envelope

V1.2.1 October 2022

The GS1-E is based off the design of Roland's System 100M series of modular synth modules. The original design is modified for eurorack and shrunk to a single envelope in 4hp.

The partial kit includes all transistors and IC's – from NOS parts, and just the passives and control hardware are needed.

The original potentiometer values were deemed too large to get the best control out of the ranges, therefore we recommend using the adjusted values (the ones recommended in the BOM) but the "alternative" values are listed for completeness however we'd not recommend using those.

The module takes a gate input, provides two copies of the envelope output and a single inverted envelope output. A manual trigger switch is also included.

If you don't like the SH tall trimmer pots you can use normal Alpha style pots and then maybe some of Thonk's trimmer caps as normal pot caps will be too large to fit. I prefer the cleaner look of the tall trimmers but the choice is yours.

The "timing" cap for the speed of the envelope can be adjusted to taste. By default the original used a 3.3uF cap, but we quite like fast envelopes and so a 2.2uF or 1uF works nicely. Of course you could build a few and go even longer, a 10uF electrolytic works just as well here.

Contact info geosync.synth@gmail.com

!!! Important Notes !!!

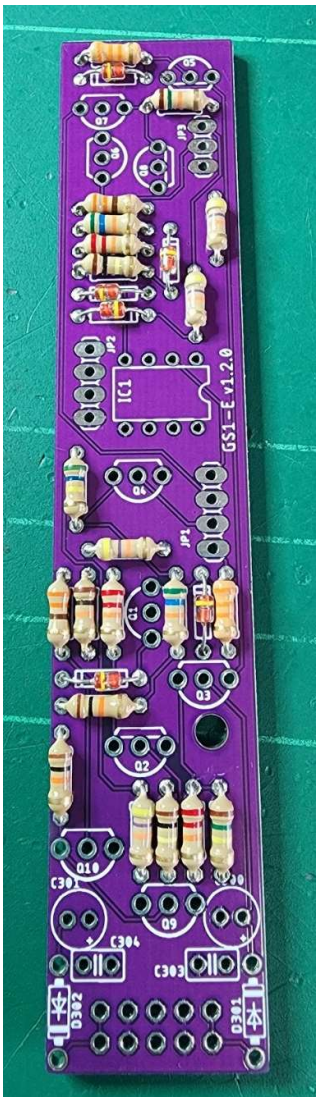
During the time Roland were making late 70's and early 80's analog synths, it seems they basically used whatever they had in stock for the 3 pin transistors. By default, in the System 100 timeframe, the 2SA1015GR and 2SA1815GR seem to have been the PNP and NPN of choice, but any ECB pinout PNP, NPN will 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.

Note on electrolytics. Due to the space constraints on this 4hp module all electrolytics must be low profile (<10mm high).

Enjoy!

Lower – Voice Board

Start with the board marked as the voice board.



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

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

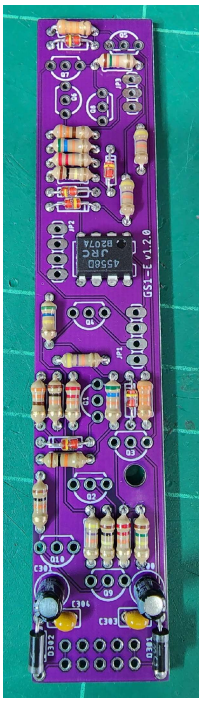
   For now, just look for and solder all the resistors that are shown with black text.

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

   a. MLCC small ceramics
   b. The electrolytics.

      Due to the stacking of the boards you must have low profile electrolytics.



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

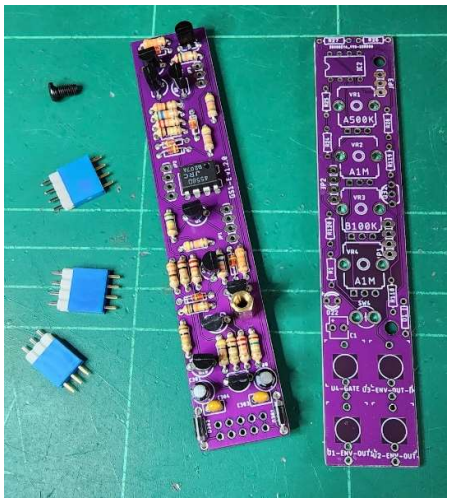
8. Add the Euro 10pin header **on the reverse side of the board.**

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 11mm standoff 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 hole 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 standoff and you should now have a nice, neat sandwich and you can simply solder all the connector pins from the top and bottom. **Make sure everything is square as there is only one standoff!**
   f. Unscrew one of the boards from the standoff and carefully pull apart the connectors. Perfectly lined up every time!

   (NB: I usually leave the standoff 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!)

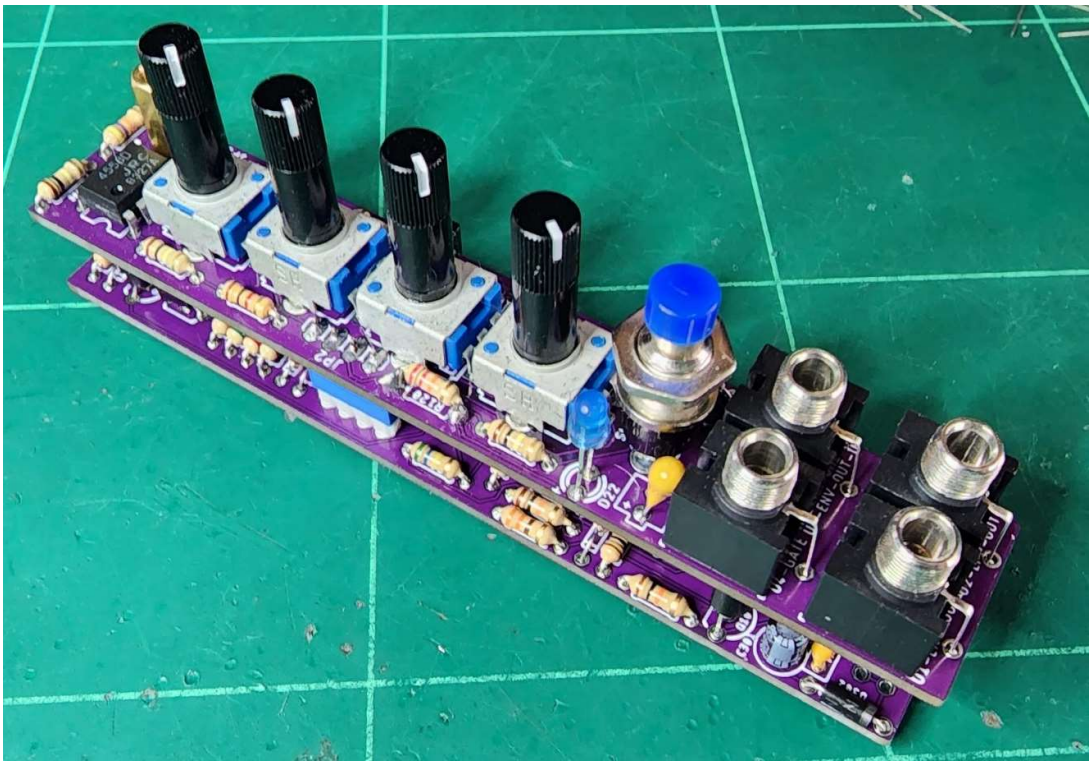


Upper - Control Board

Complete the same order of components on the control board:

Diode, Resistors, IC, capacitor and transistor.

The tantalum 3.3uF capacitor is the timing cap for the envelope. This is the speed of the envelope overall, lower values = faster, higher values = slower (you can use electrolytic here also, so if you want a slow envelope then a 10uF electrolytic for example.



Insert **but do not solder yet** the pots, 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.

Add the 11mm standoff, next to the 4558 IC as this is used to secure the top half of the panel. It should be attached on the component side of the PCB as per the brass standoff in the top left hand corner of the picture above.

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

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

Calibration

There is no calibration required with this module.

Enjoy!