GS1-O Build Guide
CEM3340 Inspired VCO

V1.2.A February 2023

The GS1-O is based off a modified reference design for the CEM3340 integrated VCO IC.

Curtis has re-released the CEM3340 Rev G IC and Alpapar have a clone AS3340 IC available, the circuit works with either IC. These are readily available from Thonk, Alpapar direct etc

The design follows a standard reference provided by Curtis, but uses the square to create a more reliable hard and soft sync circuit, the default hard and soft is a bit hit and miss from the IC pins themselves. Sync in and out is provided, so a pair of modules can be sync'd, or sync provided from or to another oscillator type.

In addition, wave shaping circuits are included to create a Ramp and Sine wave as we as the default Sawtooth, Square and Triangle. The Square has manual pulse width control as well as CV pulse width modulation with an attenable input.

The oscillator can be frequency modulated, and again provides an attenable CV modulation input.

The default ranges of the oscillator are provided with 5 steps from 32' to 2' – the tuning can also be fine tuned via the Tune pot.

Contact info geosync.synth@gmail.com

!!! Important Notes !!!

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…

This module consists of 3 PCBS, one being a mezzanine daughter card to mount the rotary switch. This should be mounted so the switch sits LOWER than the upper PCB, see building the upper control board section for details.

Enjoy!

Lower – Voice Board

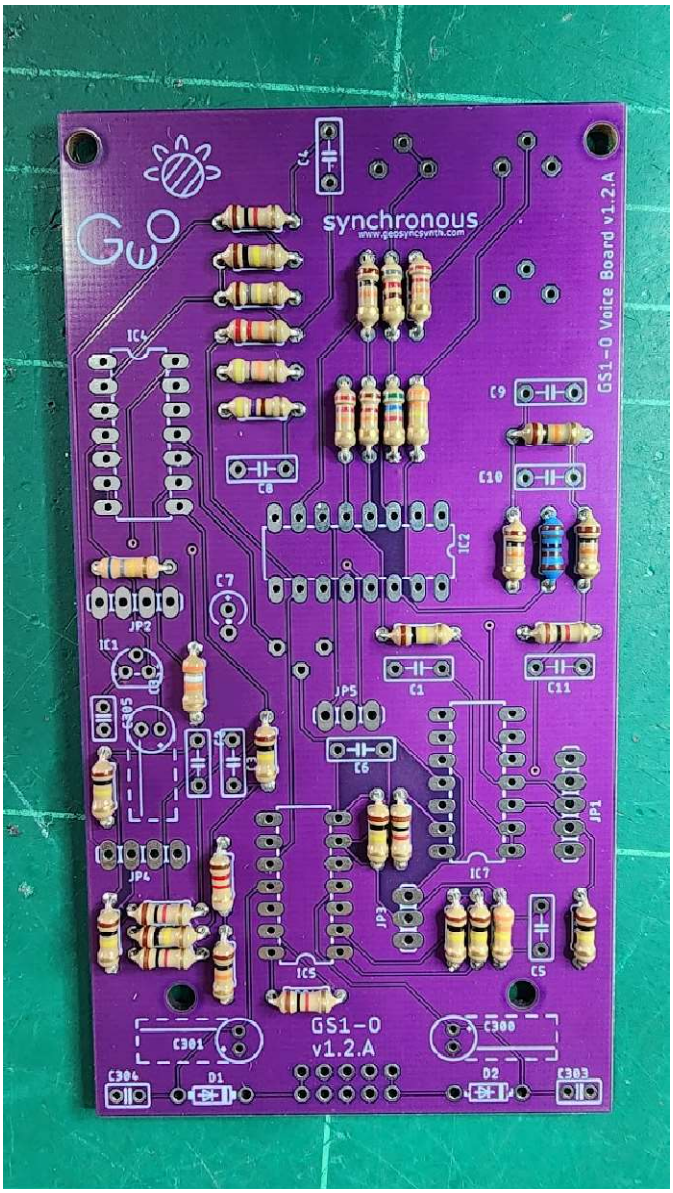
Start with the board marked as the voice board.

1. 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.

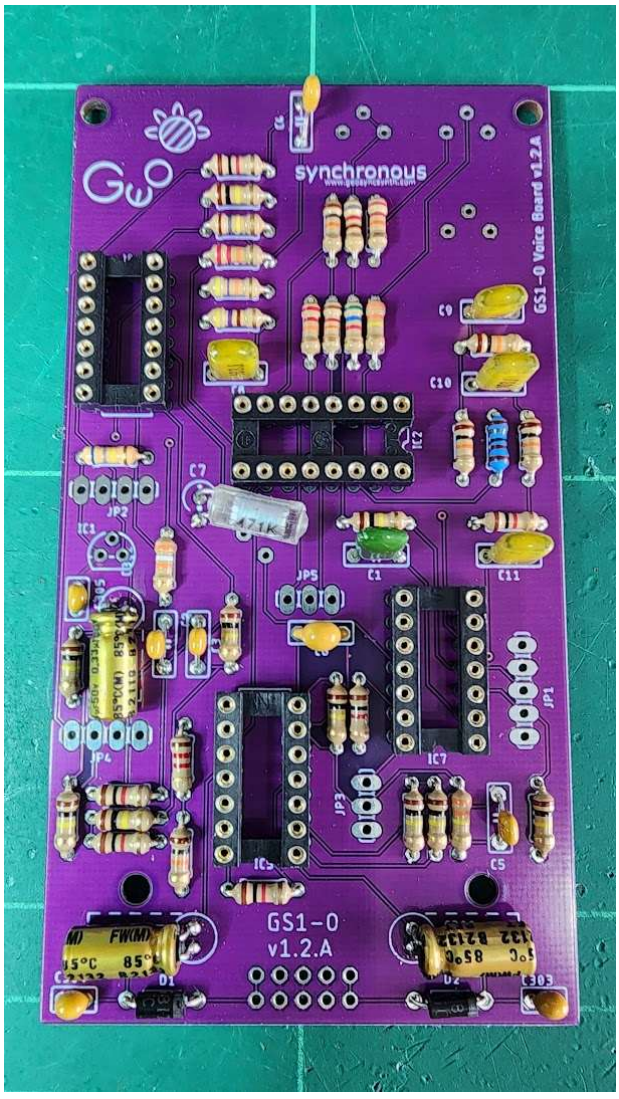
   The board should look something like this; some pictures are of prototype boards and may differ slightly.



2. Solder the two 1N5817 diodes.
3. I like to add the IC sockets next - if you are using.
4. Next comes the capacitors, again my routine is usually;

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



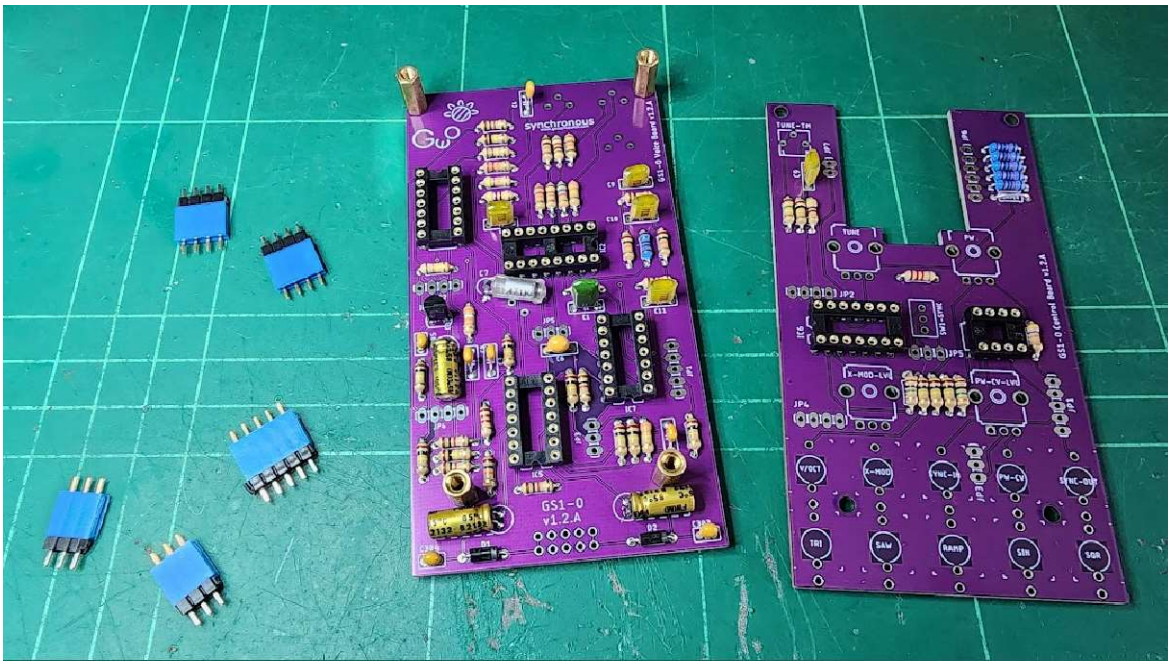
5. Now we can add the one 5v power regulator IC.

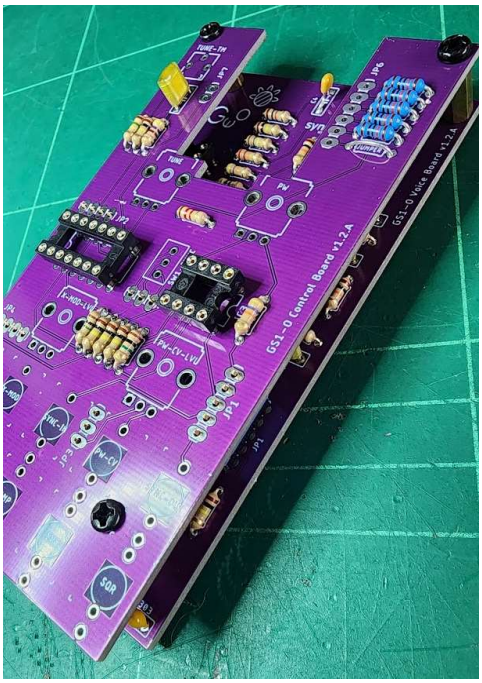
6. Add the ICs into the sockets (or solder them).
7. Add the headers to join the two boards.

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

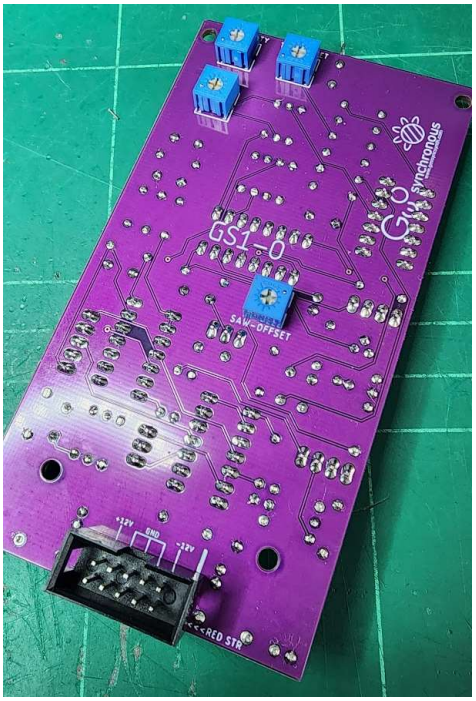
a. Screw the 11mm standoffs to the voice board. The standoff should be on the same side as the components, so screw from the underside.
b. Cut / prepare both the male and female connectors and join them together.
c. Place all into the voice board sticking up the same way as the standoffs. **Don't solder 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!)





8. Once done I like to clean the flux off the voice board at this point then finally, add the Euro 10pin header and the 3pin trimmers **on the reverse side of the board.**

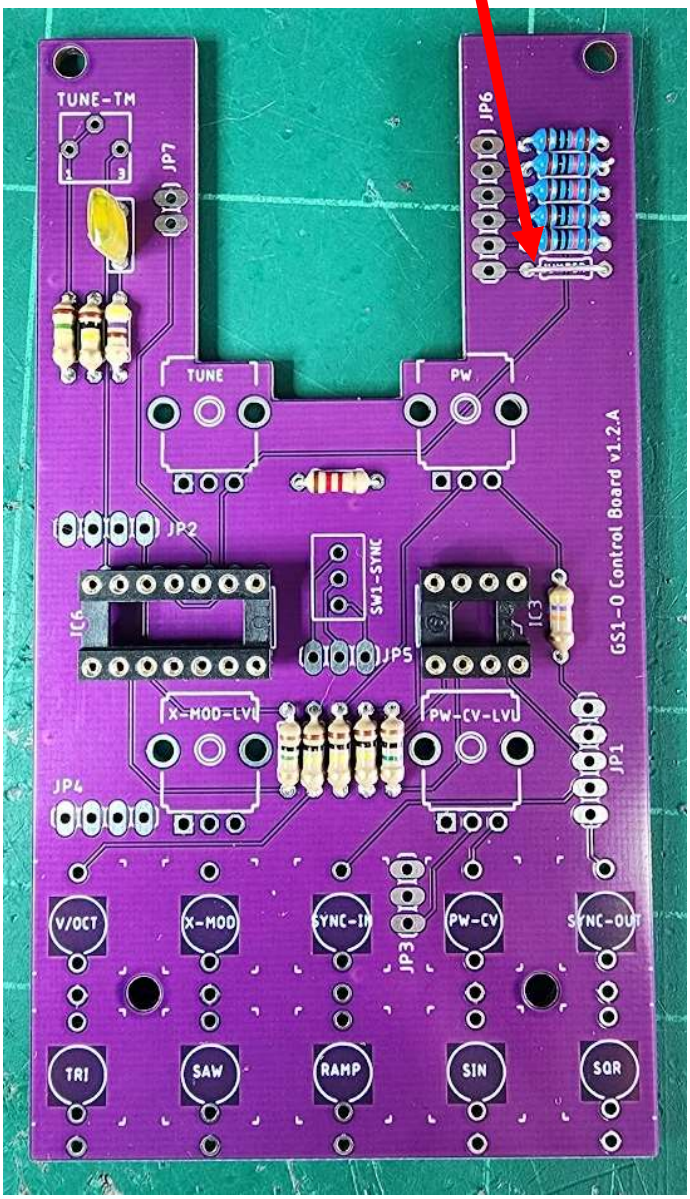


Upper - Control Board

Complete the same order of components on the control board:

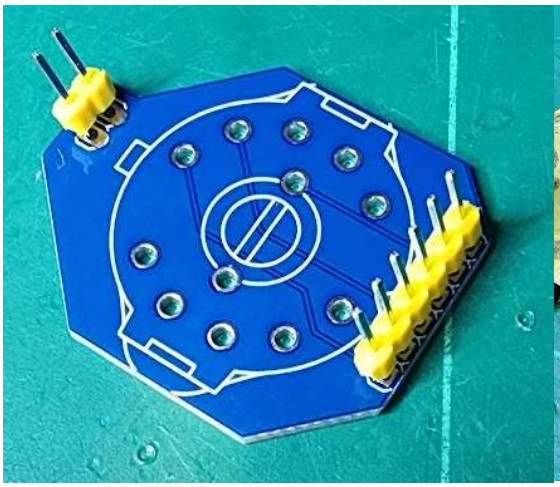
Resistors, ICs, capacitor and the trimmer on the top side this time.

Don’t forget about the jumper wire where marked JUMPER

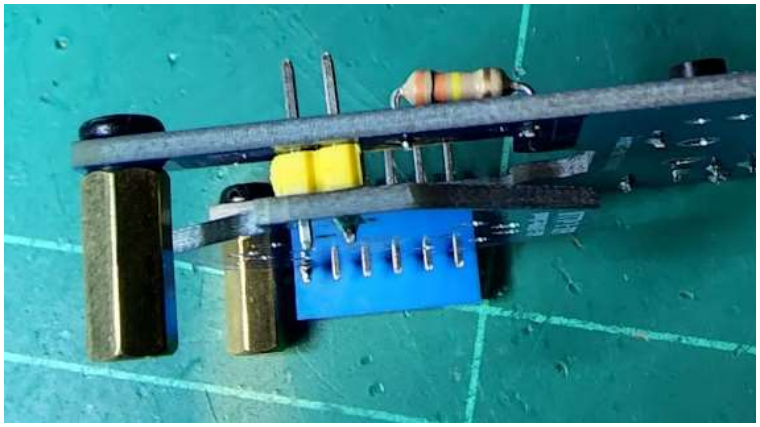


Now we need to attach the mez board for the rotary switch. Note these pictures are from the GS1-L build, but the steps and process is the same.

1. Place the 2pin and 6 pin header strip into the top of the mez board as shown. Use some masking tape if it helps keep it together, turn it over and solder JUST ONE pin on each side.

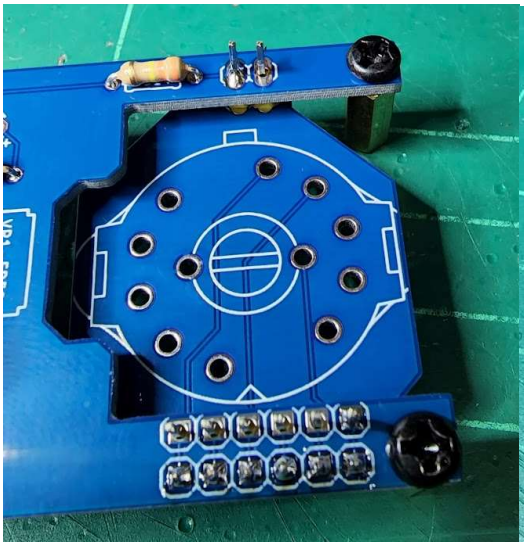


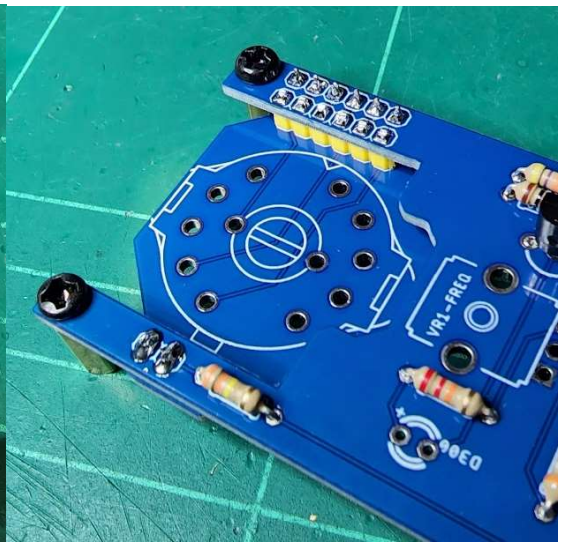
2. Insert the mez board into the upper board as shown. You may need to heat and adjust one of the pins you soldered if they are not lining up.



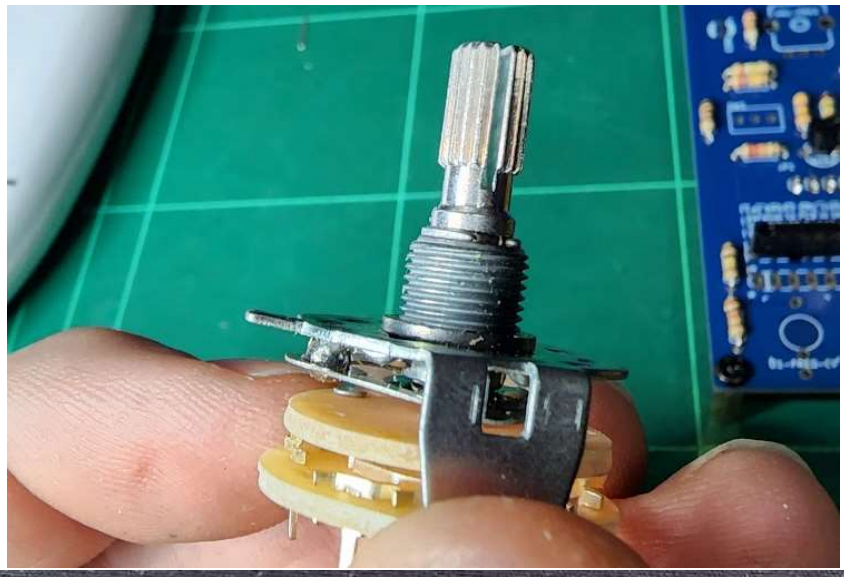
3. When you are happy that its all lining up, push the two board together as close and parallel as you can so the plastic on the pin strip is even all around then solder the rest of the pins on the underside. Push down slightly as you solder so it stays flush.

4. Now repeat the process on the top side, solder just one pin on each strip, check its all flush and then solder them all. Finally trim the excess pins.





Before we insert the rotary switch check all the pins are straight and bend the locator tab flat as shown and insert it carefully into the mez board **but do not solder yet.**



Insert **but do not solder yet** the pot, two toggle switch, potentiometers and the jack sockets.

Place the front panel over them so you can ensure they line up with the holes.

For the toggle switch, 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.

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

You may not want to attach all the nuts yet as we will need to remove the panel to access the tuning trimmer when we calibrate.

Calibration

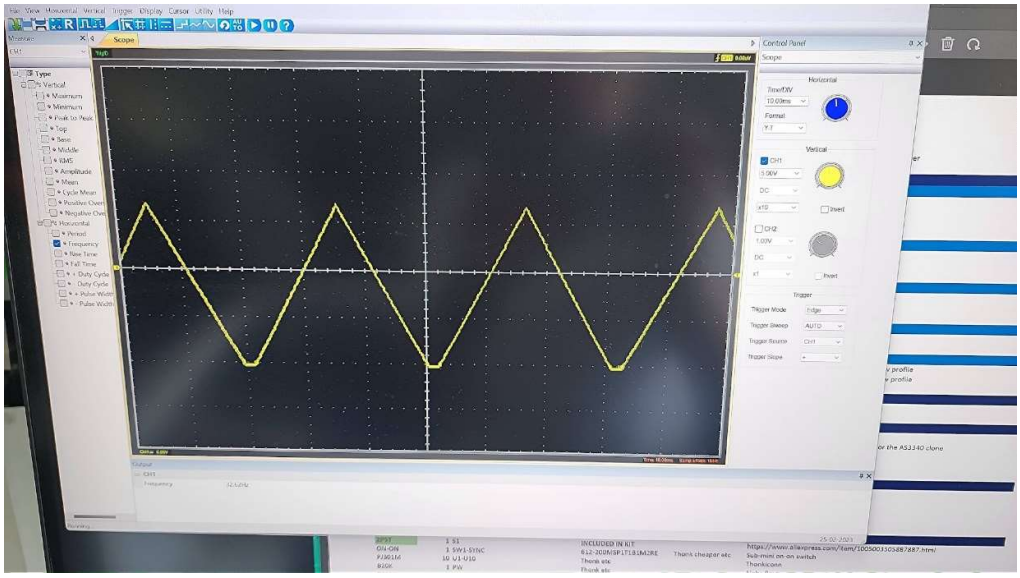
The calibration is pretty simple but you will need a scope.

Remove the panel to get access to the Tune trimmer, play C1 and set the frequency to 32.7Hz. I like to set the Tune potentiometer to roughly center and then you have some control up and down when playing the osciallator to enable tune control.

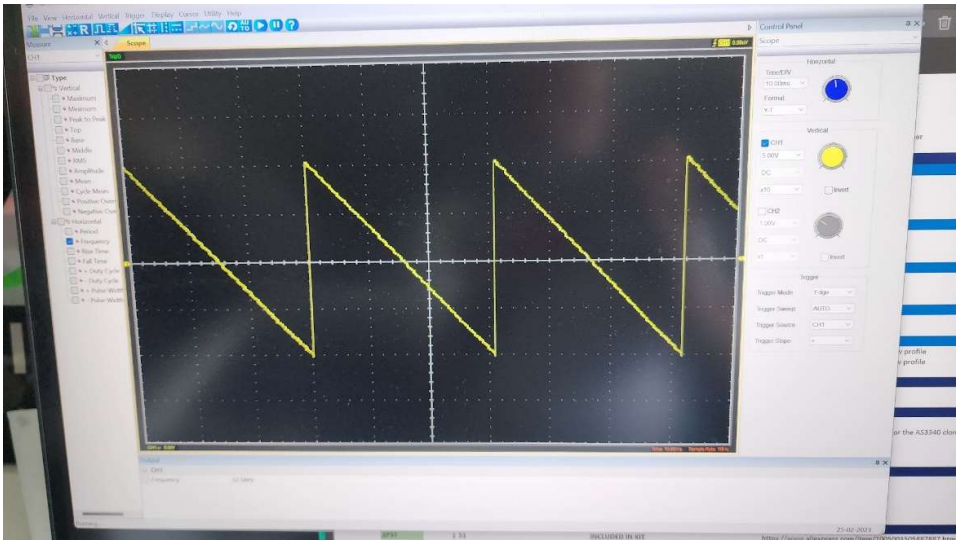
Play C2 and check you get 65.4Hz, adjust the width trimmer on the rear and then check C1. By adjusting both trimmers iteratively you should end up with a nice perfect octave scaling. As you move higher up the octaves you may find the width gets further out. You can use the linearity trimmer to being the higher frequency octave back in range. As always, with tuning calibration it can be a real pain, but time spent now can be really fruitful when you are using the module so have a break if you are getting frustrated and come back to it later.

Assuming you tested and got matching 10K resistors the five position switch should double the frequency with each notch – within some reason. Now that we have the frequency set, we can replace the panel and work on the trimmers on the rear.

On the rear, set the TRI offset to get as close to a perfect triangle as you can. It will be slightly offset from zero, if you want to be perfect you can increase the trimmer value, i.e swap the 20K for a 50K.

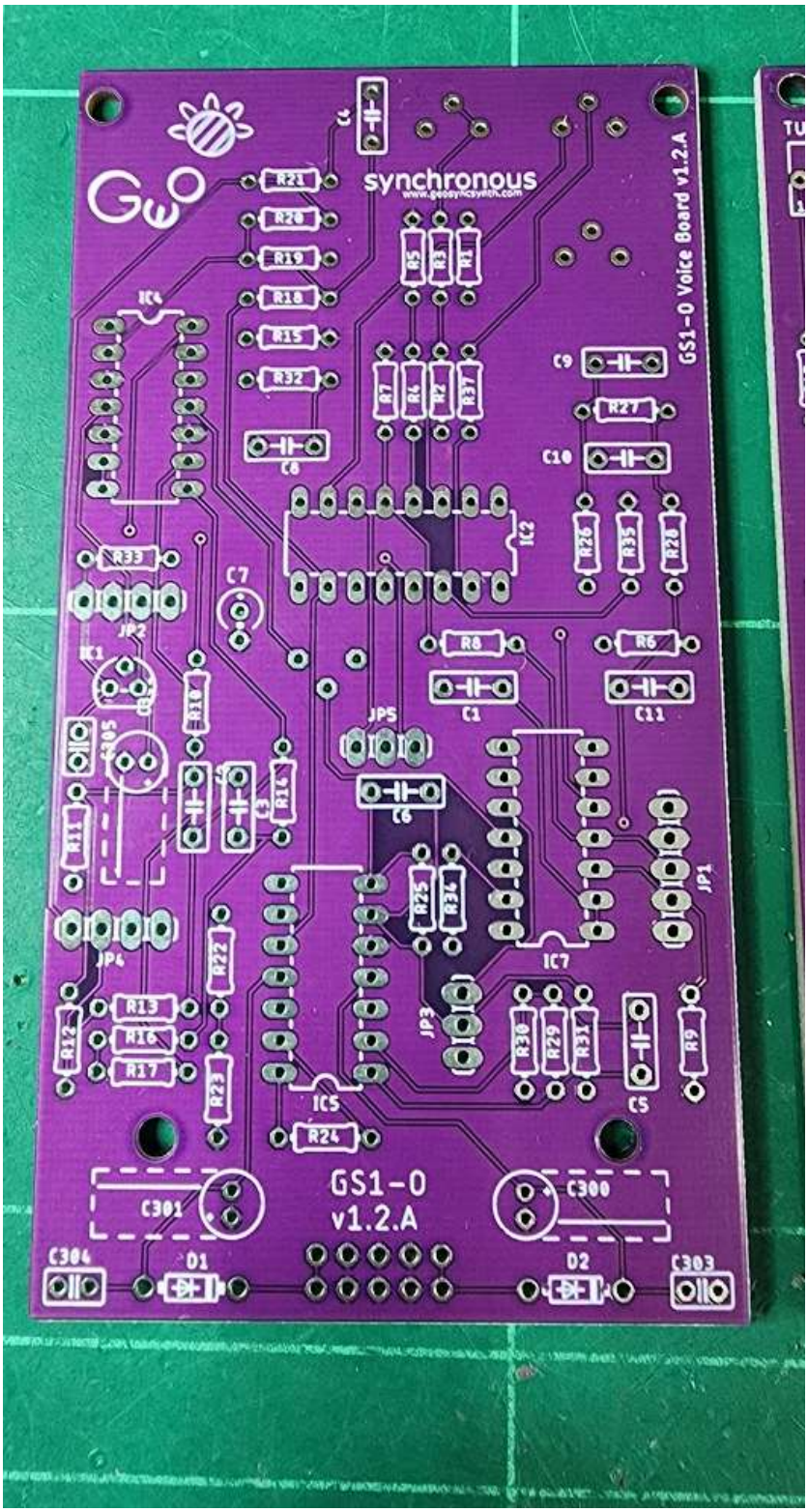


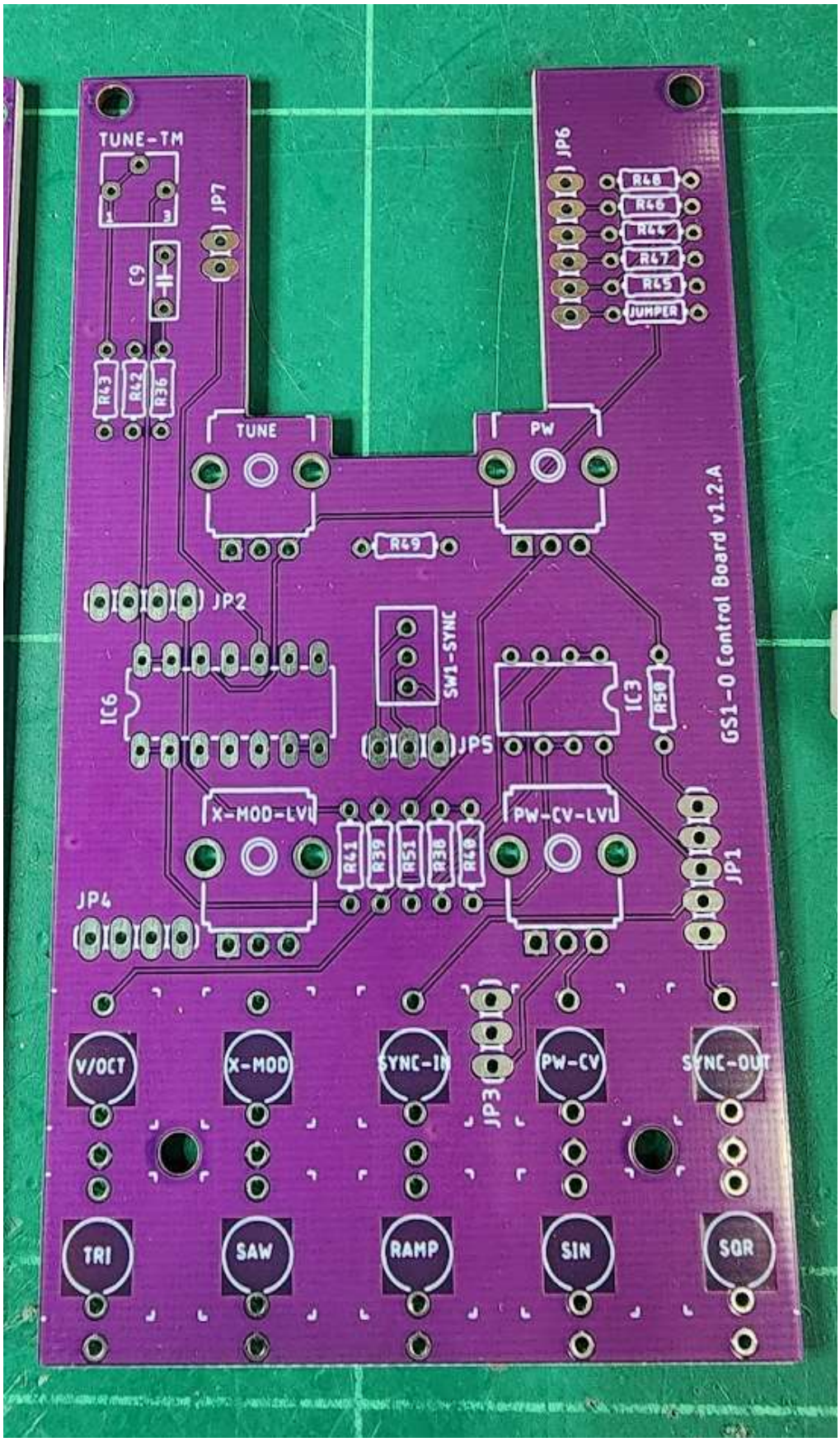
Then do the same for the SAW Offset :



PS. You may notice the Sine isn't really a Sine!...

Appendix A – PCB Layout




TUNE-TM
JP7
C9
R43
R42
R36
JP6
R48
R46
R44
R47
R45
JUMPER
TUNE
PW
R49
JP2
SW1-SYNC
IC6
IC3
R50
JP5
X-MOD-LVL
PW-CV-LVL
R41
R39
R51
R38
R40
JP1
JP4
V/OCT
X-MOD
SYNC-IN
JP3
PW-CV
SYNC-OUT
TRI
SAW
RAMP
SIN
SQR
GS1-O Control Board v1.2.A