

The VCS3 Card for 208 buildnotes

7812 and 7909 : align them well with the golden pad and avoid going over. Screw them to the PCB with the screw head on solder side and nut on components side.

"key" resistor : select according to the desired V/oct tracking 1V/oct : 100k - 1.2V/oct : 120k - 2V/oct : 200k (the BEMI / Buchla USA Easels track at 1.2V/oct).

"LED 1k" resistor : value may be adapted depending on the LED used and desired brightness.

"shape" resistors or trim pots : should be selected on test - both emplacements should remain empty if the "shape" trim pots are used - more details in the calibration schedule below.

"shape lo" resistor or trim pot : resistor should be selected on test or a trim pot can be used - more details in the calibration schedule below.

"10k lin CD pots" : a center detent pot is recommended but not essential.

"2N5172 SOT" transistor : should be selected on test for the noise - more details in the calibration schedule below.

"1k TC" resistor : a 1k 3300ppm tempco is recommended here for better pitch stability and should be glued to the metal can supermatched pair (2N2916 or other). A normal 1k resistor can be used if you don't have a tempco, if so, no need to glue it to the metal can.

100k "frequency" trim pot : be careful that there's no solder bridge between its central lead and the 4k7 resistor above.

100µF capacitor : mount it horizontally if the height exceeds 11mm as pictured below.

2N5172 transistor emplacement without central hole : snip the transistor's central lead.

AS394 supermatched pair : snip the leads 1 and 5 (lead 1 is the tab, lead 5 is the opposite one) to fit in the 6 holes of the "2N2916" spot on the PCB. The tab orientation is the same.

Tinijax socket : the solder lugs should be bent to be parallel to the panel to fit between the surrounding ICs. The sleeve (ground connection) is not needed, solder a cable **to the tip only**, prior to mounting the front panel.



Rear edge connector : use no washer or a plastic one on the left mounting screw near the +/-15V pads, it might cause a short between the traces and ground plane if the soldermask is scratched.

Calibration

Calibration should be performed with the VCS3 Card plugged in the 208's program card edge connector. An oscilloscope and a frequency meter or are needed.

Noise

On an oscilloscope monitor the noise signal at the "noise level" pot with the "colour" pot at mid-course. Test different transistors with central lead bent : insert without soldering them into the holes of the "2N5172 SOT" emplacement to the rear of the board, next to the "noise" trim pot, and adjust the trim pot to find one that gives a noise symmetrical with respect to 0V without distortion. The amplitude should be between 3 and 4Vpp. Check for a smooth "colour" sweep.

When you have a good working transistor, snip it's central lead and solder it in place horizontally.

The noise needs about 30 seconds after power up to start working. No noise or random voltage just after power up is normal. The random voltage generator needs a clock. The clock input is the 208's inverter "to prog" / "to card" socket.

The Fairchild 2N5172s (available from Tayda) work nice for the noise.

The CDIL ones (available from TME) don't work for the noise, they make no sound, although ok elsewhere.

The BC169C is a good drop-in replacement.

The BC547 too but the pinout is different : ECB for the 2N5172 / BC169C and CBE for the BC547.

Oscillator shape

The normal procedure in the EMS Service Manual is SOT resistors but on the VCS3 Card board the "shape" 500k trimpots can be used instead, if you don't have a stock of various resistors. **Do not use the trimpots and resistors at the same time.**

Monitor the triangle output on an oscilloscope with the "shape" pot at mid-course (center detent) and the range switch to "hi". You'll most probably have an asymmetrical triangle. The goal is to make it isoceses.

SOT resistors : insert a resistor in one of the emplacements and see if it makes the triangle closer to isosceles. If it does, adjust the resistor value until you get an isosceles triangle, if it doesn't, try the other emplacement. You may need to select a resistor in both emplacements for accuracy. When you have an isosceles triangle, turn the shape pot fully CW and fully CCW and check there's no "flat end", if there's one, readjust the resistors values until you get perfect ramp and inverted ramp with the pot fully CW and fully CCW. In most cards I built (using 2N5172 and BC214L), a single resistor between 27k and 36k in the spot between the 2 BC214L below the "noise level" pot is OK, as shown on the pic below.

500k trimpots : adjust the trimpots to get an isosceles triangle. When you have it, turn the "shape" pot fully CW and fully CCW and check there's no "flat end", if there's one, readjust the trimpots until you get perfect ramp and inverted ramp with the pot fully CW and fully CCW.

The triangle has a smaller amplitude than the ramps and doesn't go negative, this is normal. The square is reduced to a line when the "shape" pot is fully CW or CCW, this is normal.

Oscillator shape lo

This compensates for the waveshape change when the range is switched to "lo" and should be adjusted after the shape was calibrated in "hi" position as described above. Here again, a 500k trimpot or SOT resistor can be used.

Monitor the triangle output on an oscilloscope with the "shape" pot at mid-course (center detent) and the range switch to "lo". Feel free to increase frequency to ease monitoring.

SOT resistor : insert a resistor in the emplacement to get an isosceles triangle, the proper value should likely be between 150k and 220k, when you have it, solder it in place,

500k trimpot : adjust the trimpot to get an isosceles triangle.

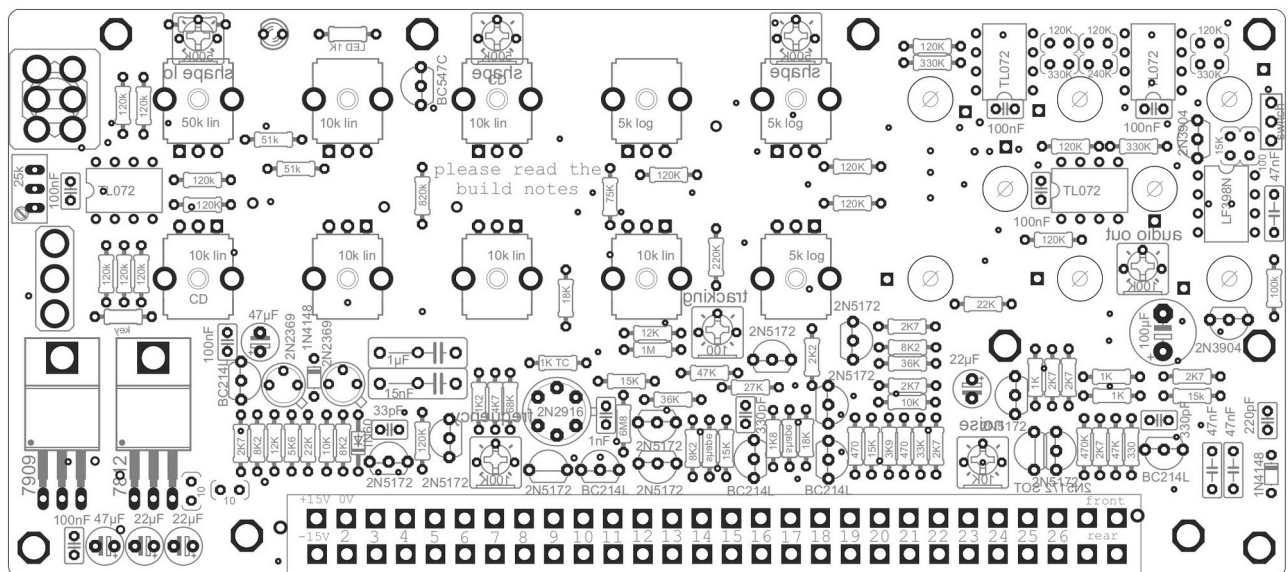
Oscillator frequency and tracking

Allow about 10 minutes warm up for better stability.

With the "frequency" and "fine" panel pots at mid-course and the range switch set to "hi", adjust the "frequency" trimpot to get about 100Hz.

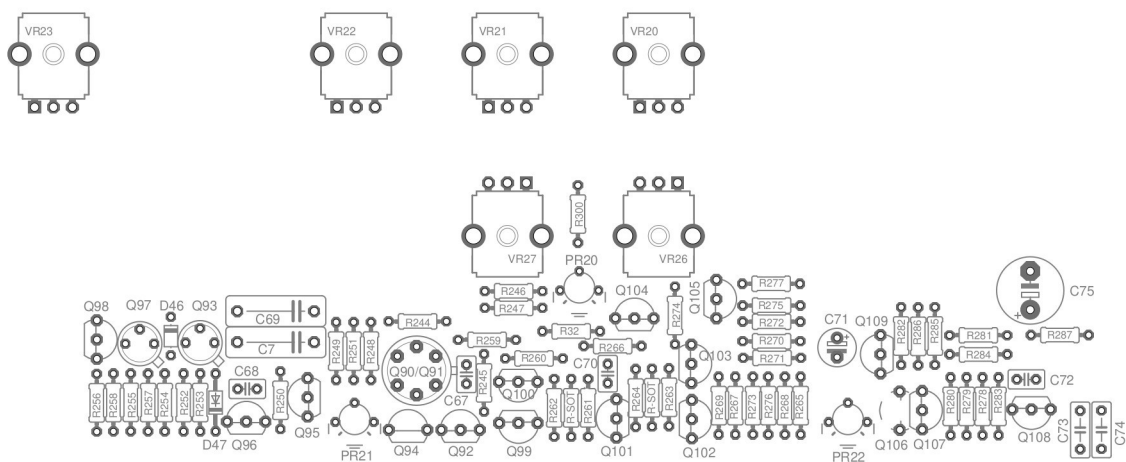
Connect a keyboard to the 208's "keyboard voltage" input and switch the VCS3 Card's "key" switch to "in".

Adjust the "key in" multiturn trimpot accessible via the small panel hole to get tempered tuning. The fine tracking can be set with the "tracking" trimpot and adjusting it will need to readjust the multiturn trimpot. With a proper setting of both trimpots, the tracking should be good over about 5 octaves.



The oscillator and noise generator schematics can be found in the Synthi A / VCS3 service manual downloadable from <https://www.synthxl.com/wp-content/uploads/2020/05/ems-synthi-aks-service-manual.pdf>
https://elektrotanya.com/emsa_synthi-aks_sm.pdf/download.html

Reference designator



Build thread for support : <https://www.muffwiggler.com/forum/viewtopic.php?p=3016934>

CO output modification (optional)

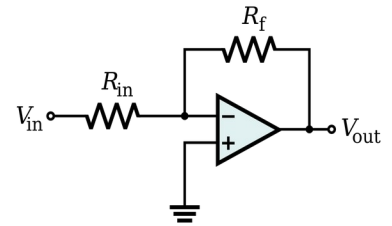
The CO is a great master for the VCS3 Card oscillator but needs a dedicated pre-LPG output. One of the 208's panel "signal output" tinijax emplacement can be used for this purpose, as is (disconnected from the PCB) with a banana/tinijax cable or replaced by a banana socket (this needs to enlarge the panel hole to 8mm).

Inverting amplification is needed. Use a TL071 or 741 with R_f 330k and R_{in} 120k.

On 208p, 208r rev2 and 208r rev2.1, an easy CO output spot is card 8 contact 4 (count starting from the top).

I'm not sure where it is on the 208e but it's likely the same, if someone can confirm, please let me know and I'll update the build notes, thank you.

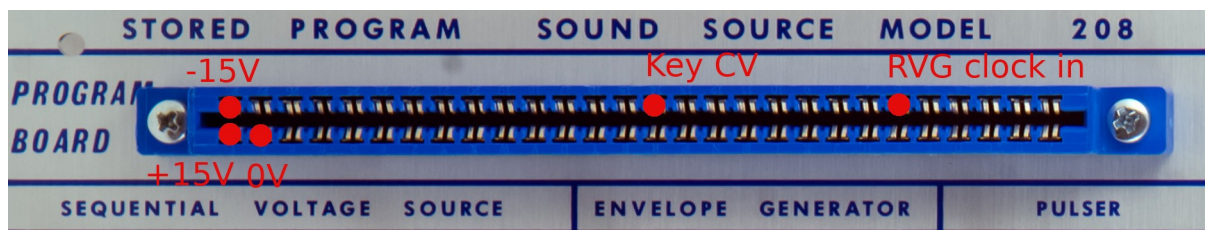
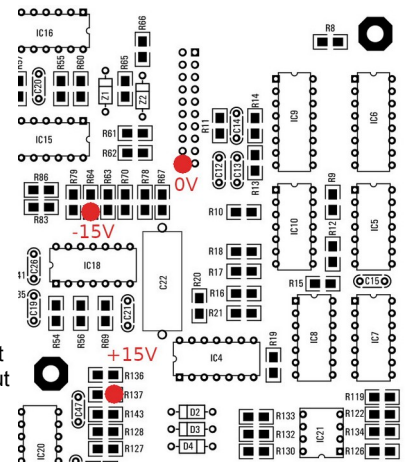
On 208r rev1 the "CO. SIGNAL" pad is an easy CO output spot.



208r rev1 modification

The VCS3 Card works with 208r rev1 if its former program card slot is rewired.

1. unscrew and remove the edge connector with transfer card from the 208's panel
2. snip all the pins soldered to the transfer card, leaving about 2mm metal
3. solder cables to contacts 1 and 2 of the bottom row and to contacts 1, 15 and 23 of the top row
4. mount the modified edge connector to the panel, it should fit above the PCB one thanks to the shortened pins
5. run the cables to destination, shorten them to the needed length and solder them
 Bottom contact 1 to +15V on R137 right lead
 Bottom contact 2 to 0V on the header
 Top contact 1 to -15V on R64 bottom lead
 Top contact 15 to "voltage" black banana socket in the 208's bottom left corner
 Top contact 23 (random voltage generator clock input) to any banana socket you like (disconnect it from the PCB of course), the 3 "to/from prog" banana jacks are CV inputs to the envelope times but the "duration" CV is not super useful, so the "inverter to prog" can be used, or another banana socket of the bottom such as a pressure purple socket or one that you don't use much
6. double check +/- 15V voltages before inserting the VCS3 Card for the first time



The VCS3 Card needs 55mA on +15V and 50mA on -15V. Because of their weak PSU, early BEMI Music Easels may not work properly with the VCS3 Card. A powered [208 Hub](#) is the easiest solution to connect cards without extra load on the Easel's PSU but is a bit overkill to connect one card only. One could ask Buchla USA ("ex"-BEMI) for a more recent upgraded PSU board but sometimes it takes long to get a reply from their customer service. Another solution is to add a dedicated PSU inside the Easel.

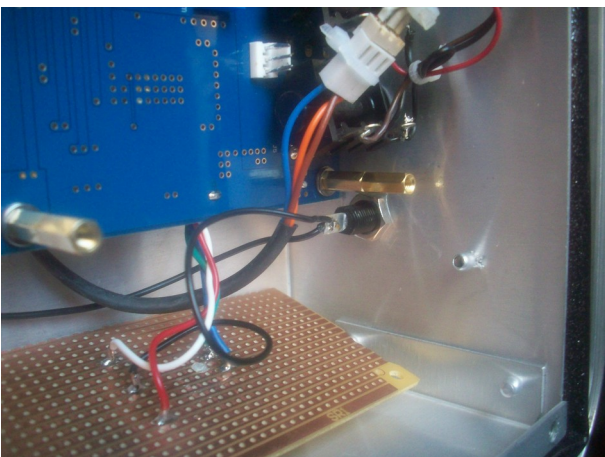
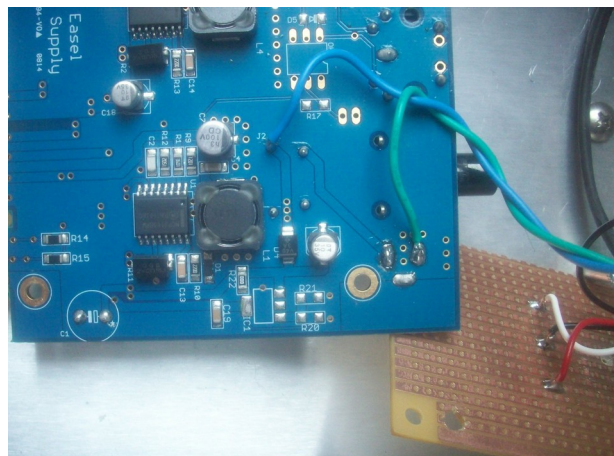
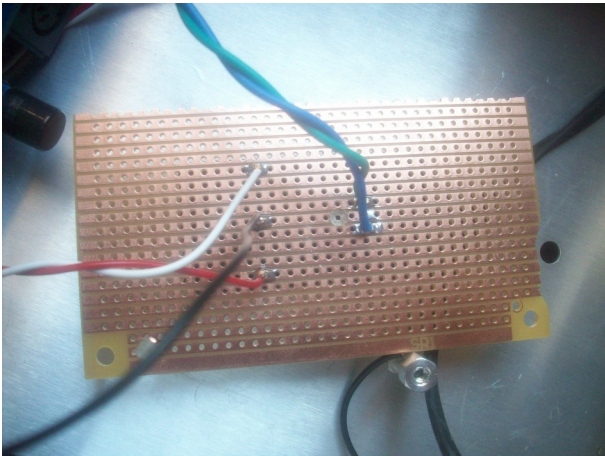
1. buy an Artesyn AEE00CC18-LS DC converter (Mouser part # 826-AEE00CC18-LS , also available from other suppliers)
2. mount the Artesyn AEE00CC18-LS to a piece of veroboard and drill the corners to match the PSU/outputs/midi board mounting screws spacing
3. unmount the 208 and 218 from the boats and unplug them from the PSU/outputs/midi board located in the 218 boat
4. unscrew the PSU/outputs/midi board
5. connect the Artesyn AEE00CC18-LS +/- Vin terminals to the PSU/outputs/midi board as shown on the pics below (green and blue cables)
6. connect the Artesyn AEE00CC18-LS COM terminal to the ground banana socket as shown on the pics below
7. connect 2 wires to the Artesyn AEE00CC18-LS +/-Vout terminals long enough to reach the 208 top left corner and run them through the boats rectangular hole
8. mount the PSU/outputs/midi board back in place with the Artesyn AEE00CC18-LS wires below and replace the 2 bottom screws with standoffs
9. mount the Artesyn AEE00CC18-LS veroboard to the standoffs
10. unmount the 208 panel edge connector by removing the screws
11. unsolder both contacts 1 pads from the transfer card
12. run the +/-Vout wires throughs the 208 panel edge connector hole
13. connect the Artesyn AEE00CC18-LS +/-Vout to the top and bottom pads 1 of the edge connector (-Vout to top, +V out to bottom) and insulate the pads with heathshrink tube
14. put the edge connector back in place
15. before plugging the 208 and 218 to the PSU/outputs/midi board, power the Easel on and check +/-15V voltages on the edge connector
16. plug the 208 and 218 to the PSU/outputs/midi board and double check all connections before powering the Easel, both PSU may not appreciate to receive +/-15V at their outputs coming from another PSU
17. double check +/-15V voltages on the edge connector again before inserting a card for the first time

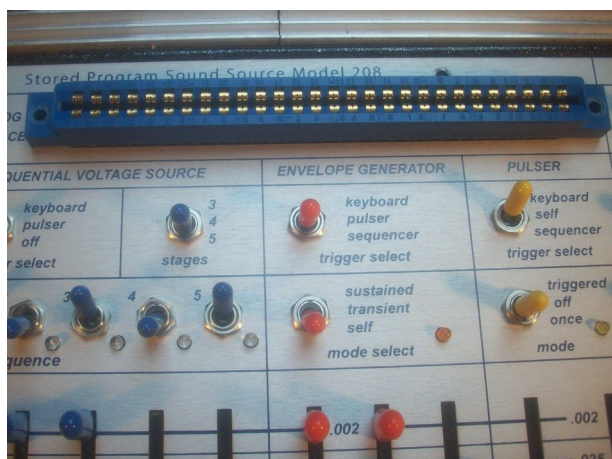
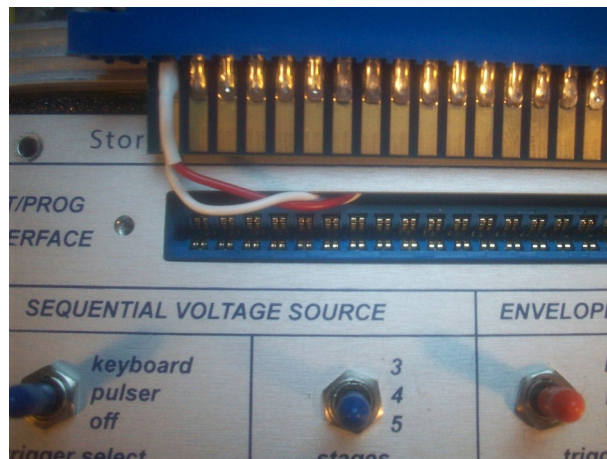
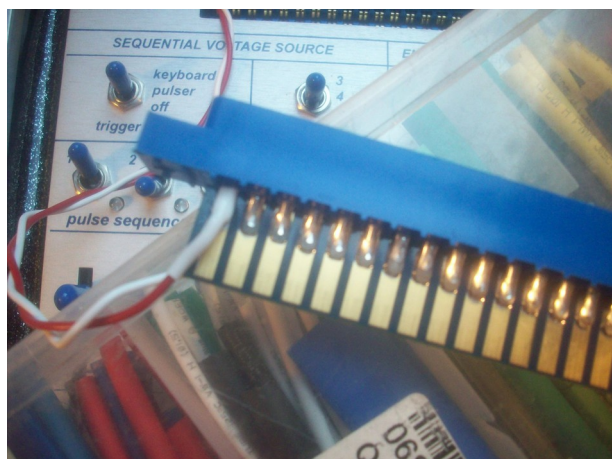
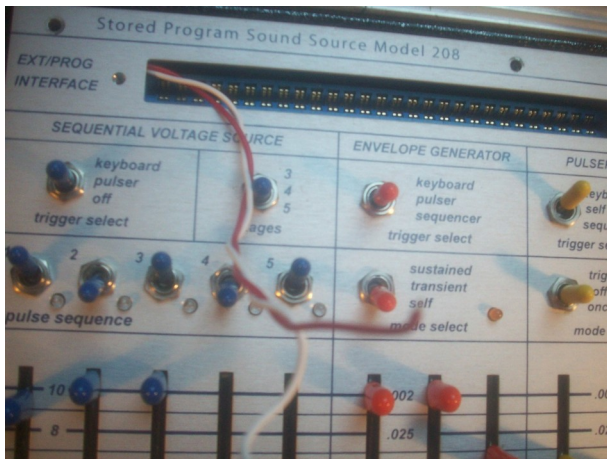
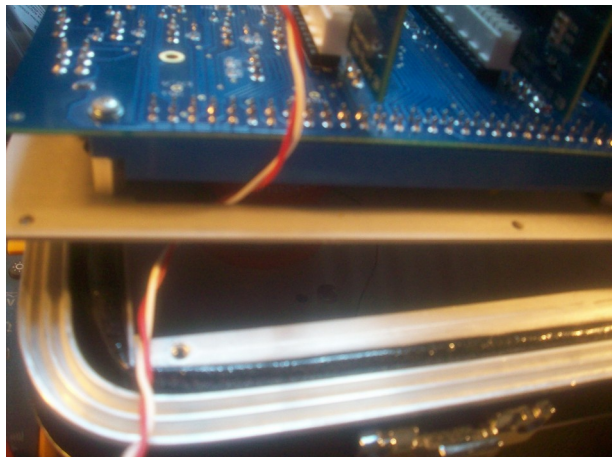
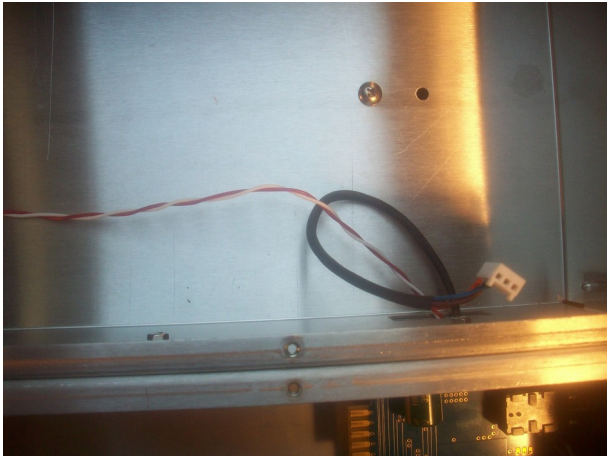
Pics

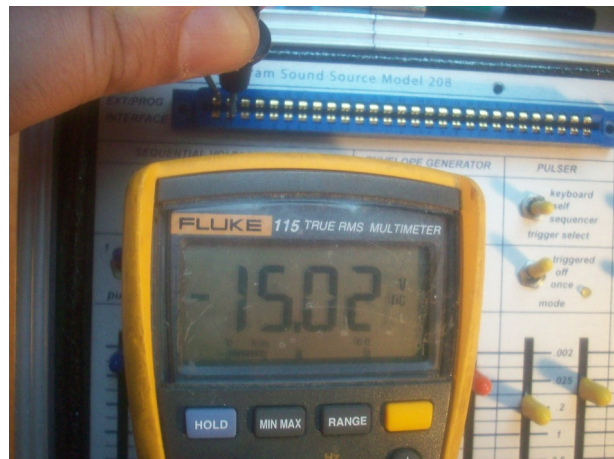
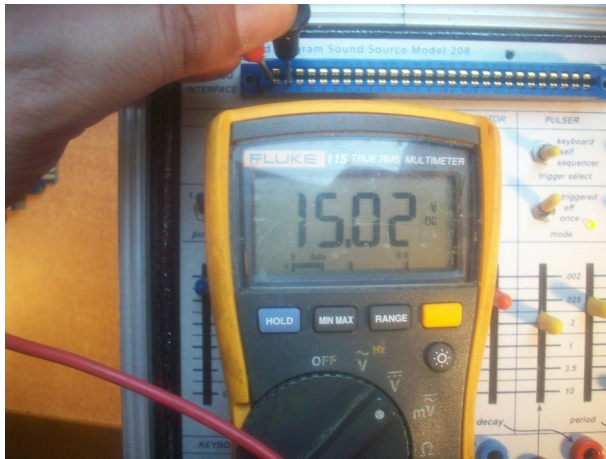
White cable is -Vout connected to the edge connector -15V contact
Red cable is +Vout connected to the edge connector +15V contact
Black cable is 0V connected to the ground banana
Blue cable is +Vin connected to +12V of the DC socket, after the mains switch
Green cable is -Vin connected to 0V of the DC socket

The modified wiring of the jack sockets is not related to this upgrade.

It was done previously to reduce the noise at the outputs by disconnecting the sockets from the PSU/output/midi board.







BEMI / Buchla USA 218e keyboard buffer upgrade

With the BEMI / Buchla USA 218e keyboard, switching the VCS3 Card's "key" switch slightly upsets the 208 CO and MO pitch. The cause is the "main" output is not buffered.

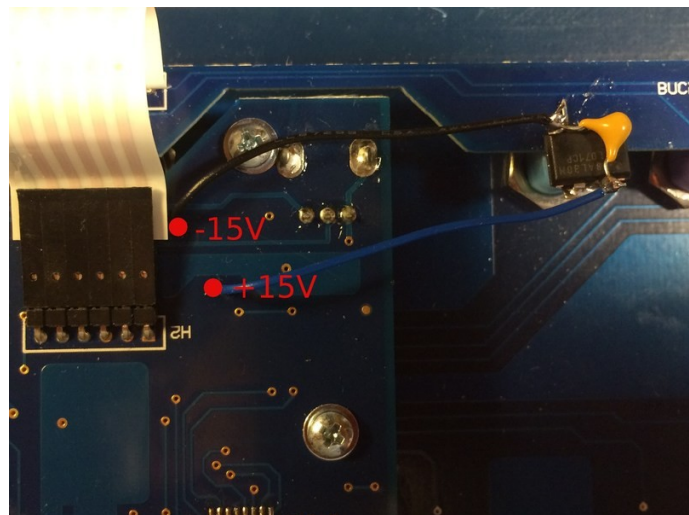
Adding a simple buffer to this output makes the CO and MO pitch stable regardless of the VCS3 Card "key" switch position.

It's a very easy upgrade without permanent or risky modification of the 218.

All needed is a TL071 opamp, a 100nF cap and wire.

Any person with basic soldering skills should be able to do it.

1. Disconnect the blue banana tab from the PCB and bend it
2. Snip pins 1 - 5 - 8 of the TL071 to ease mounting
3. Bend and connect together pins 2 and 6 of the TL071
4. Solder the TL071 to the bent banana tab using it's soldered pins 2 and 6
5. Solder pin 3 of the TL071 to the PCB tab where the blue banana tab was formerly soldered
6. Connect pin 4 of the TL071 (black cable) to a -15v pad on the 218PCB : the via spotted in red on the pic or where the power connector is wired
7. Connect pin 7 of the TL071 (blue cable) to a +15v pad on the 218 PCB : the via spotted in red on the pic or where the power connector is wired
8. Solder the 100nF cap across pins 4 and 7 of the TL071



Disclaimer

The PCBs come unpopulated and without parts.

I assume no liability for personal injury or damage to equipment or loss of use caused directly or indirectly by the use of these PCBs, the VCS3 Card or the info shared in these build notes.

The build and modifications should only be performed by those experienced in electronics with access to a good oscilloscope and frequency meter.

I'm happy to help in some cases but I don't offer free support. These PCBs are for the ones able to debug their own builds.

Please understand remote troubleshooting of a wrong build might be very difficult and time consuming.

In case of unsolvable problem, contact me to have your unit shipped to me and fixed at cost.

As I'm constantly improving my designs, some changes might occur without notice and can't be subject to complaint.

I'll answer any question, feel free to ask before ordering.

Thank you and happy building !

Most recent update : June 23 2022