

synthCube

SEM VCF

(oberheim circuit realized by jeff farr)

module documents v5.0

05/29/2014

NOTE: This version of the SEM VCF does NOT include implementation of the VC control for resonance. The PCB includes circuitry for VC RES should you choose to implement it, but this panel does not allow for the required controls. The panel and PCB can be modified to allow for the additional controls at the builder's option.

V5.0 includes revisions and fixes documented in the muffwiggler thread including the pcb trace cut, with thanks to roglok, consumed, makers and other 'early adopter' wigglers

Errata:

Circuit Description

Designed by Tom Oberheim, the SEM VCF....

A discussion thread about the jeff farr circuit layout is here:

<http://www.muffwiggler.com/forum/viewtopic.php?t=69431&highlight=sem>

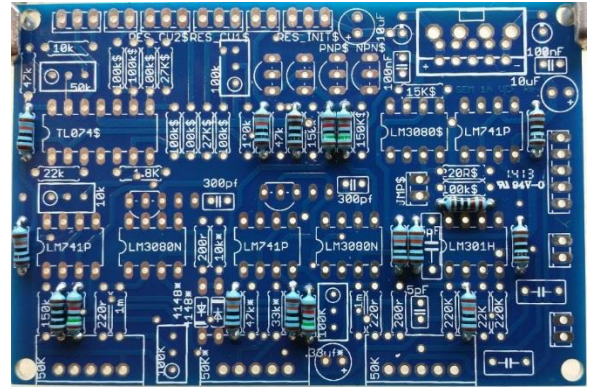
A build thread for the SEM is here:

<http://www.muffwiggler.com/forum/viewtopic.php?t=82093&highlight=sem>

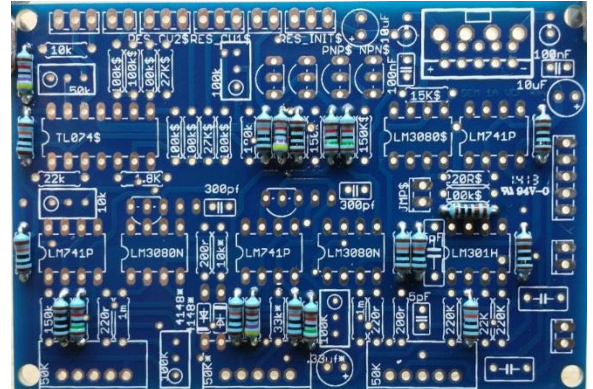
General Assembly Preparation

- 1) Check that the kit contains all of the parts noted in the BOM; notify us of any discrepancy using info@synthcube.com
- 2) Gather additional materials and tools necessary for your build:
 - a. Tools (soldering iron, pliers, wire cutters/stripper, etc)
 - b. Solder of your choice
 - c. Digital camera to document your progress
- 3) Prepare the pots for assembly:
 - a. Break off the metal tab
 - b. Slide the metal tab into the slot in the shaft- this adds some strength to the shaft for mounting knobs

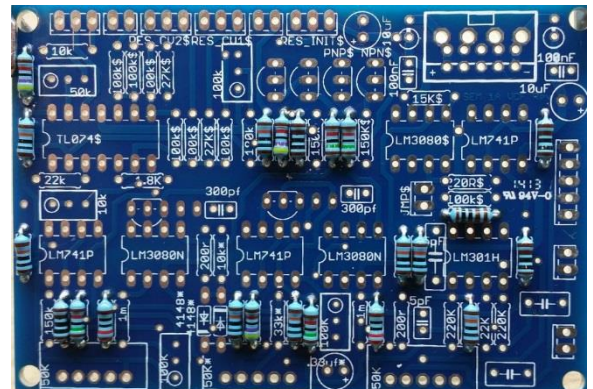
5. Next install the 1K resistors.



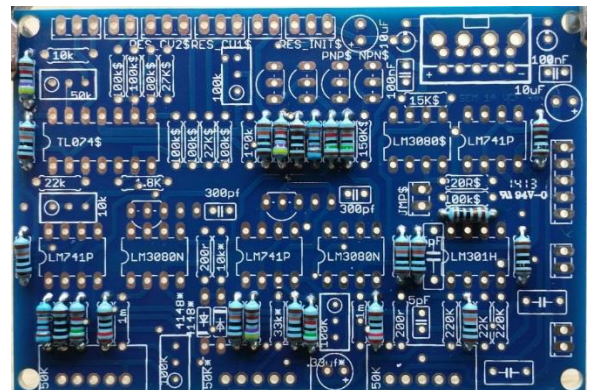
6. Install the 47k resistors.



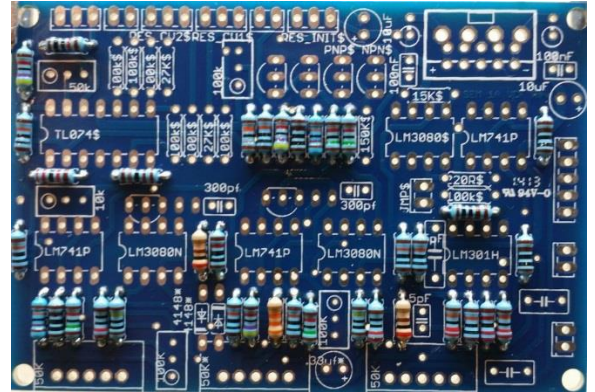
7. Next install the 220ohm resistors. Only two of the three supplied resistors are used this build.



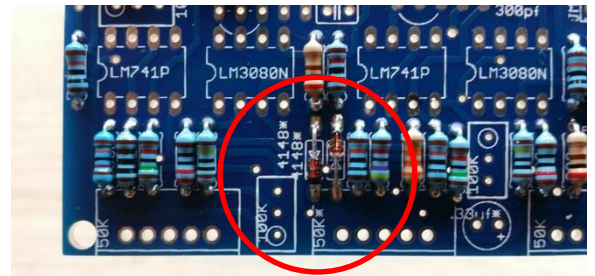
8. Next install the 120 K resistors. For this build the 120 K resistors replace the 150 K resistors. This is for compatibility with 12V systems including euro format. For 15V system builds, install 150K resistors, not 120K resistors. The 120K resistors are installed in the two spots labelled '150K'. For this build, the resistor labelled '150K\$' is not installed.



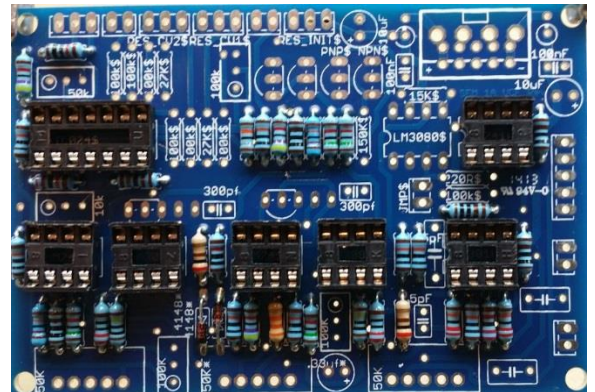
9. Next install the remaining single and double resistors. The only unpopulated resistor spaces should be those required for the VC resonance circuit (indicated by the symbol '\$') which are not used in this build.



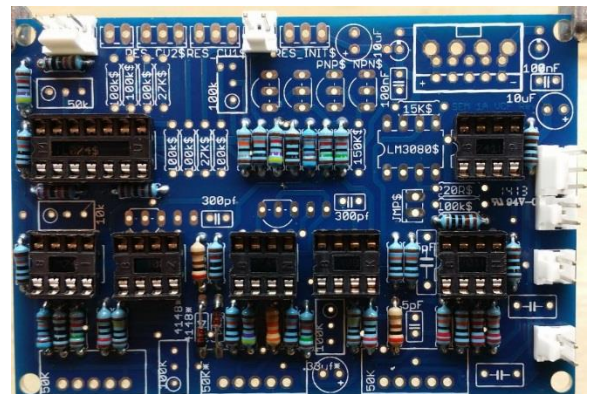
10. Next install the diodes. Orient them so the black bands are over the white bands on the PCB silkscreen.



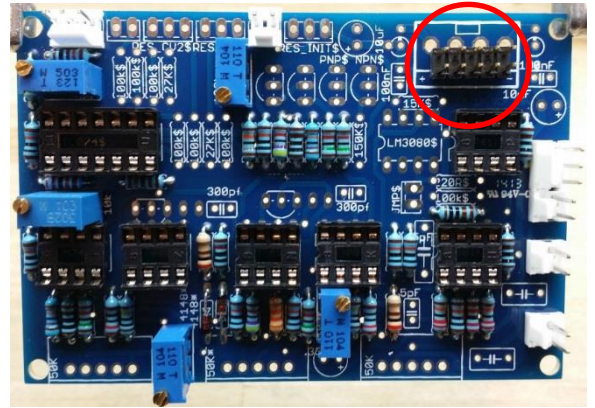
11. Next install the IC sockets. Note that the TL074 socket **MUST BE INSTALLED** for the module to work. Although the silkscreen erroneously uses the '\$' symbol, the TL074 is required. Orient the sockets properly to match the silkscreen image.



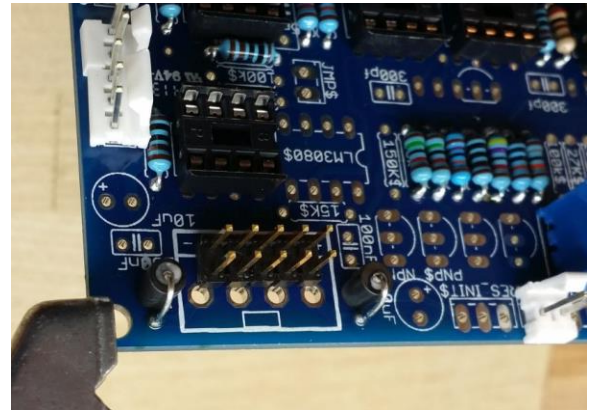
12. Next install the two-pin, three-pin and five-pin .10 MTA headers along the edge of the PCB. Headers required for the VC Res circuit are not included for this build.



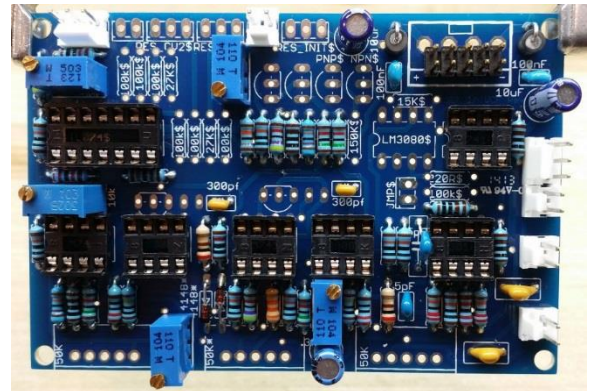
13. Next install the header for the power cable connection. In this build we are using a 12V euro-compatible connector in the form of a ten-pin two row header. Also install the trimmer resistors, paying attention to the orientation. The 100 K resistors have the number "104" on the top; 10K resistor has the number "103" on the top and the 50 K trimmer has the number "503" on the top.



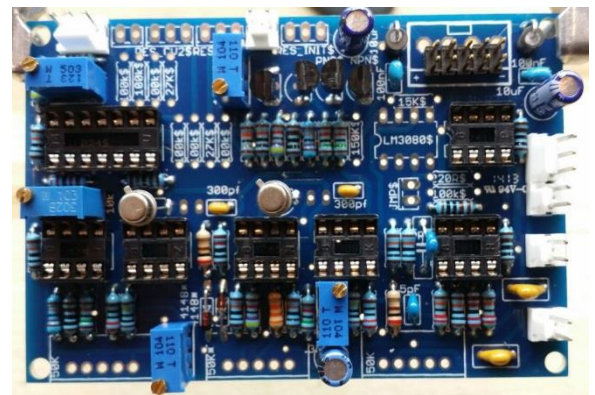
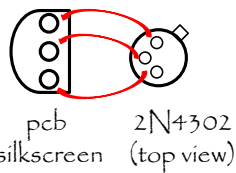
14. Next install the ferrite beads. Note the ferrite beads have to be installed vertically to sit on the PCB properly. Bend the leads so the ferrite beads sit vertically as shown



15. Install the MLCC capacitors. Note that the original BOM specifies 0.15nF caps, but these are 0.15uF caps and are installed in the lower left corner in the unmarked silkscreen borders next to the two-pin header (green or yellow caps) For one of the two 5.0pF caps, you will have to bend the leads to make them fit the wider spacing. Next install the electrolytic capacitors. Pay attention to the orientation for the electrolytic caps- don't reverse them.



16. Install the transistors on the PCB. Install the 2N4302 transistors as shown, since the silkscreen is not labeled. The 2N4302 are provided in a 'can' housing. The center pin should be bent slightly and installed in the center pin of the silkscreen layout. Leave room below the can housing for the IC to fit in the socket below. Note: one of the 2N3906 must be rotated 180 degrees and is installed OPPOSITE how the silkscreen image is printed. See the diagram showing the back of the PCB at the end of this build guide.



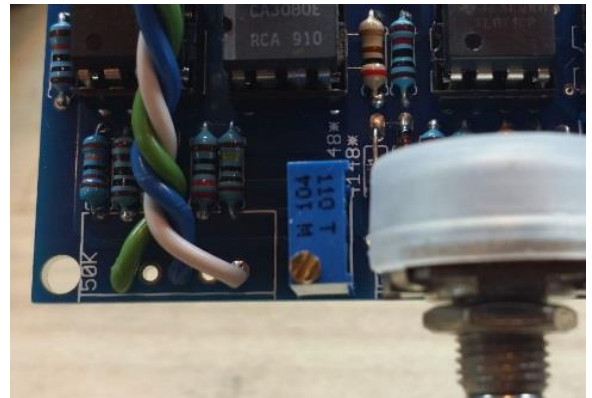
17. Wash the PCB if you are using water-washable solder. Carefully inspect each solder joint with a magnifying glass. Reflow any suspect joints. Visually check each component for orientation, solder joint, and correct part number to eliminate possible trouble spots.

18. Next install the two PCB- mounted potentiometers on the PCB. Optional: You can install an extra nut on the pot shaft before mounting the PCB to the panel this helps align the PCB with the panel. The extra nuts are not included in the kit.

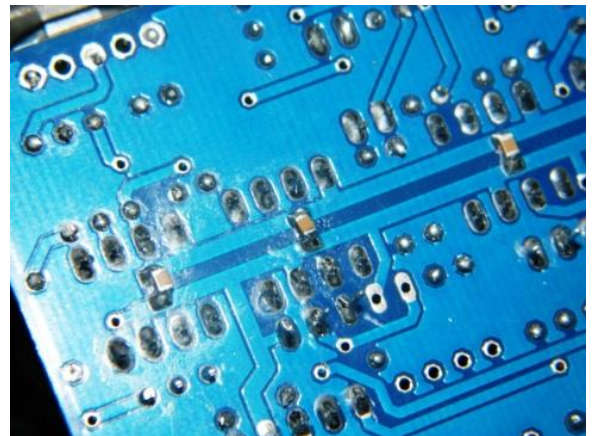
19. Install the integrated circuits into the IC sockets. Please make sure to orient the ICs properly.



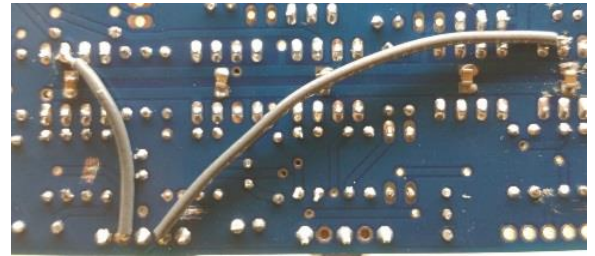
20. Install the wire for the frequency pot. The kit provides a twisted three-wire in Grn/Lav/Wh or Grn/Bl/Wh. Strip and tin the ends of the wires. Solder the PCB ends to the Frequency pot pads, noting the correct orientation.



21. Solder the SMT caps to the *back* of the PCB. These are very small parts but can be installed carefully using traditional equipment. Apply your iron to one pad, and touch your solder to the pad to create a small drop of solder. Using tweezers, grip the SMT cap and touch one end of the cap to the hardened solder drop, then touch your iron to the solder to liquefy it. Pull the iron away and allow the solder to harden, holding the cap in place, before removing the tweezers. Touch your iron to the other pad and allow solder to flow under the other end of the capacitor. Voila. There are good youtube videos available for more information about this technique.



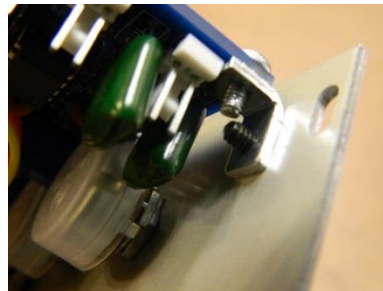
22. Install the jumpers used to fix the PCB issue related to bleed-through. (thanks to 'roglok' and 'consumed' on muffwiggles for documenting the fix!) Install one end of each jumper to the unused pot pads, and the other ends to the outputs on the op-amps as shown. Inspect the area carefully for continuity and shorts since the op-amp outputs are right next to the SMT caps.



23. Assemble the panel—install the pots and jacks. Also install the 'L' bracket with the #4-40 screw through the panel. Note the orientation of pots and jacks.



24. Mount the PCB to the panel using the 'L' bracket and a #4-40 screw. Thread the two PCB-mount pots carefully through the panel. Install washers and nuts on the pots.



25. Solder the twisted three-wire to the frequency pot, paying attention to the pin orientation. We use heat shrink wrap over soldered pot, jack and switch lugs but that is the builder's option.



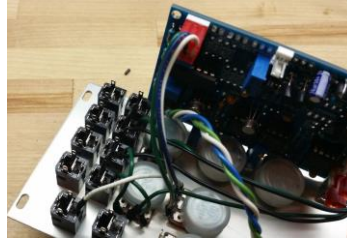
26. Next, solder the 'Audio 2' input jacks and pots. Strip and tin the green and black wires attached to the two-pin red socket. Solder the green wire from the two-pin socket to the center lug of the Audio Input 2 pot. Solder a black wire to connect the ground (sleeve) lugs of the first Audio Input 2 jack to the second Audio Input 2 jack, and at the same time solder the black wire from the two-pin socket to one of 'Audio Input 2' jack ground lugs. Last, solder green wires from the two 'Audio 2' jack signal lugs to the remaining Audio Input 2 pot lugs according to the wiring diagram. Attach the red socket to the Audio 2 two-pin header on the PCB.



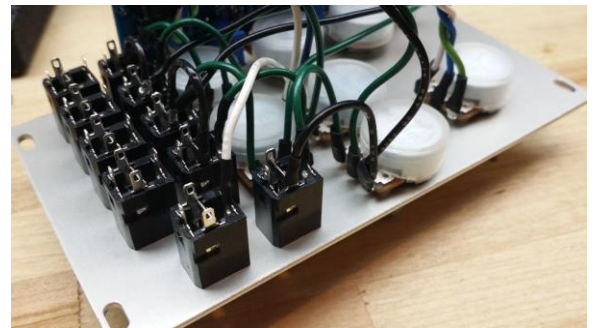
27. Repeat these steps for the 'Audio 1 Input' input jacks and pot.



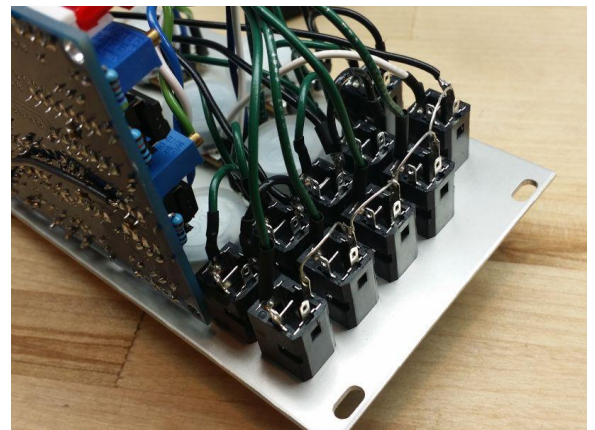
28. Next wire the attenuverted cv jack and pot. Strip and tin the wires attached to the three-pin red socket connector. Strip and tin the 4" white wire. Solder both white wires to the solder lug on the cv attenuator pot. Solder the other end of the white wire to the CV jack. Next solder the green and lavender wires to the other lugs on the cv attenuator pot per the diagram. Attach the connector to the three-pin header on the PCB.



29. Next wire the attenuated cv pot and jack. Strip and tin the green and black wires attached to the two-pin red socket connector. Strip and tin the 2" green and black wires. Attach the green wires to the left and center (from the rear) lugs of the cv pot. Solder other end of green wire to signal lug on cv jack. Solder the two black wires to the correct lug on the pot. Then solder the other end of the black wire to the ground lug on the jack. Attach the connector to the two-pin header on the PCB



30. Next wire the output jacks. Solder the ground lugs of the outputs jacks together using scrap resistor leads to create neat connections. Next solder the black wire from the five-pin red socket connector to one of the output jack ground lugs. Solder the green wires to the correct lugs on each output jack. Attach the connector to the five-pin header on the PCB. Be sure to connect the ground lug on the CV attenuverter pot at the same time.



31. Visually inspect each component.

32. Attach the knobs to the pots. Polish the panel with a soft cloth to remove fingerprints. Attach the power cord to the ten-pin header on the PCB. Be CERTAIN to keep the +12V and -12V sides oriented to your power supply by noting the direction of the red wire on the power cable ribbon.



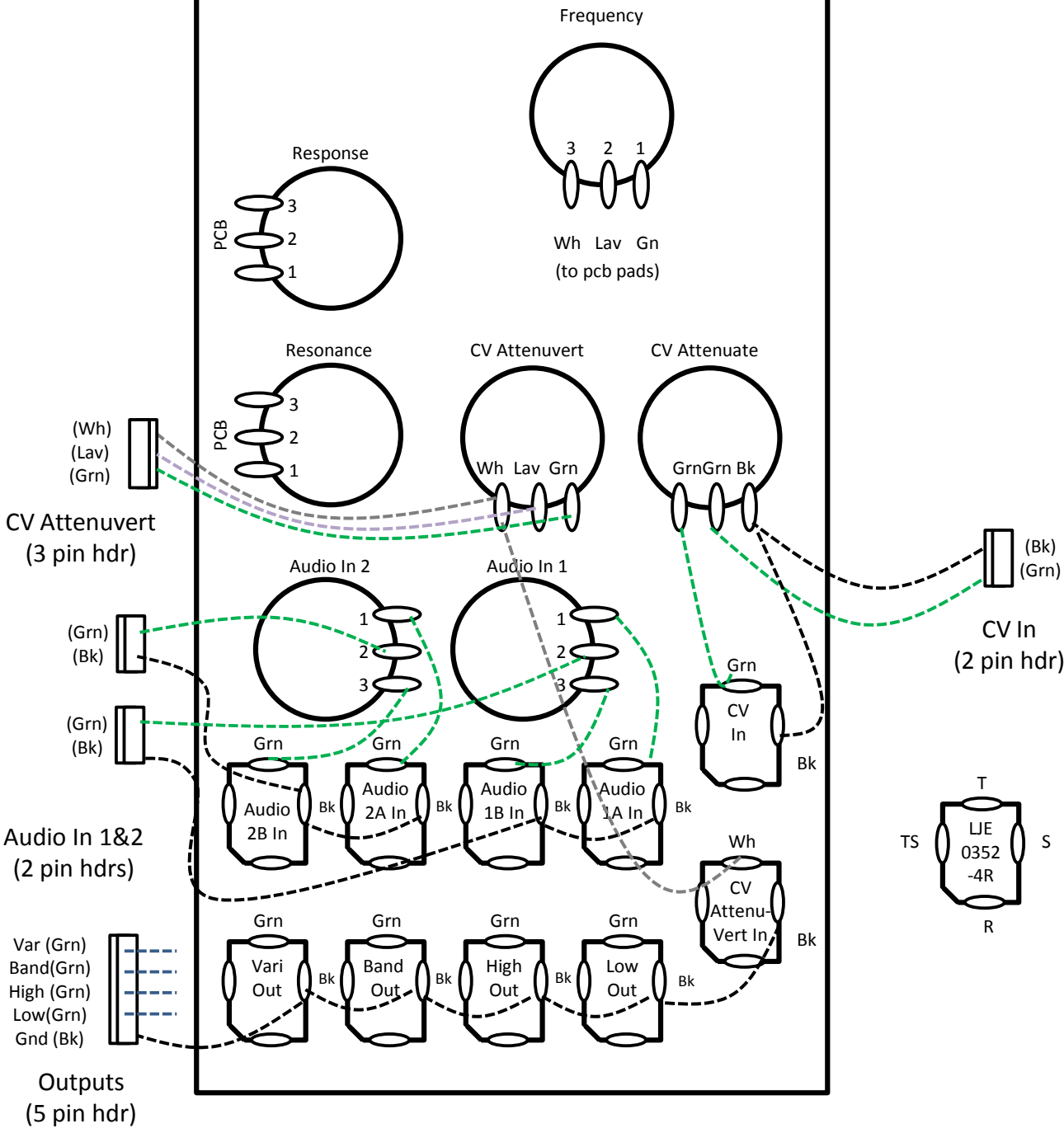
Description	Prov	Quant	Part Notes	Ref Designator
Mylar Film Capacitors Film Capacitors .15uF 10%		2		C2, C10
Aluminum Electrolytic Capacitors - Leaded 50volts 10uF		2		C6, C7
Aluminum Electrolytic Capacitors - Leaded 50volts 0.33uF		1		C5
5.0pF Ceramic Capacitors		2		C3, C19
MLCC - SMT 50volts 0.1uF		8		C1, C11, C12, C13, C14, C15, C16, C17
MLCC 0.1uF		2		C8, C9
300 pF Ceramic Capacitors		2		C4, C20
Transistors 2N3904		2		T2, T4
Transistors 2N3906		2		T3, T5
JFET 2N4302		2		Q1, Q2
1N4148 Diode		2		D2, D3
Filter Beads		2		L1, L2
Metal Film Resistors 1.8Kohms 1% 50PPM		1		R18
Metal Film Resistors 1Kohms 1% 50PPM		1		R140
Metal Film Resistors 1Kohms 1% 50PPM		4		R10, R25, R27, R28
Metal Film Resistors 1.0Mohms 1% 50PPM		2		R7, R143
Metal Film Resistors 10Kohms 1% 50PPM		2		R12, R15
Metal Film Resistors 15Kohms 1% 50PPM		5		R1, R2, R8, R147
Metal Film Resistors 22Kohms 1% 50PPM		2		R21
Metal Film Resistors 27Kohms 1% 50PPM		2		R39, R40
Metal Film Resistors 33Kohms 1% 50PPM		1		R11
Metal Film Resistors 47Kohms 1% 50PPM		3		R13, R16, R24
Metal Film Resistors 100Kohms 1% 50PPM		15		R4, R9, R14, R17, R19, R22, R30, R31, R35, R36, R37, R38, R142

moogah SEM Filter BOM- PCB				
Metal Film Resistors 150Kohms 1% 50PPM		3	replace for 12V versions	R23, R26
Metal Film Resistors 180Kohms 1% 50PPM		1		R20
Metal Film Resistors 200ohms 1% 50PPM		2		R5, R144
Metal Film Resistors 220Kohms 1% 50PPM		2		R29, R155
Metal Film Resistors 220ohms 1% 50PPM		3		R6, R145
resistor 120K		2	replaces 150K for 12V	R23, R26
Trimmer Resistors - 3/8" 50Kohms Sealed Vertical Adjust		1		ATV
Trimmer Resistors - 3/8" 100Kohms Sealed Vertical Adjust		3		T1, T9, INITIAL_FREQ
Trimmer Resistors - 3/8" 10Kohms Sealed Vertical Adjust		1		V/OCT
LM301ANG		1		IC4
TL-071		3		IC5, IC6, IC8
CA3080E		3		A1, A2, A13
TL-074		2		IC1
8 pin IC sockets		7		
14 pin IC sockets		2		
VERT BRKAWAY HDR 10P dual row tin		1		Power
FRICITION LCK HDR 2P Straight Post tin		5		CV2, IN1, IN2, RES_1VOCT, RES_DRIVE
FRICITION LCK HDR 3P Straight Post tin		4		CV1, RES_CV, RES_CV2, RES_INIT
FRCTN LK HDR STR 4P Square post tin		1		Power
FRICITION LCK HDR 5P Straight Post tin		1		OUT
Housings CLOSED W/O TABS 2P red tin 22 AWG		5		
Housings CLOSED W/O TABS 3P red tin 22 AWG		4		
Housings CLOSED W/O TABS 5P red tin 22 AWG		1		

SEM VCF BOM- euro clarke68 panel



<u>Description</u>	<u>Prov</u>	<u>Quant</u>	<u>Part Notes</u>	<u>Ref Designator</u>
16mm pots 50K linear PCB mount, knurled split shaft		2		PCB
16mm pots 50K linear solder lug, knurled split shaft		5		Panel
3.5mm mono jack		10		
Knob- large 33.3mm		1		
Knob- small 13mm		4		
Knob- med 19.8mm		2		
Panel- clarke68		1		
Panel Screws M3X.5X6mm Pan		3		
Panel Screws M3X.5X6mm Pan , black		1		
Washers, Nylon, #4		4		
10-16 power cable, euro		1		
L Brackets- 4-40 threaded		1		
4-40 X1/4" machine screw pan head		2		for L bracket
Wire- 3 color twist- 6"		1		Off Board Freq Pot
Wire- Black 6"		4	crimped	
Wire- Black 2"		3		
Wire- Green 6"		7	crimped	
Wire- Green 2"		1		
Wire- Green 4"		5		
Wire- White 4"		2	one crimped	
Wire- Lav 4"		1	crimped	



SEM VCF Rear View
Euro Clarke Panel

Calibration and Tuning

Complements of 'makers' and 'roglok' on the muff's forum:

"Makers: " I found a calibration procedure for the original SEM VCF adjustments on the following page and I've gone through that procedure. I believe that Q8 as described below is Notch output. I was not able to set Initial Frequency well based on the procedure because the procedure assumes you have SEM VCO's. I still get some High Frequency Bleed on the Low Pass but all in all I am very satisfied with the sound.

<http://www.angelfire.com/music2/theanalogcottage/sem1.htm>

- 1) Using an oscilloscope, check pin Q8 on board -to-board connector (this pin is the nearest to connector I1).
- 2) Turn "Notch" pot to "HP".
- 3) Adjust "OFFSET 1" trimmer to zero volt.
- 4) Turn "Notch" pot to "LP".
- 5) Adjust "OFFSET 2" trimmer for zero volt.
- 6) Center VCO1 frequency pot and VCF frequency pot (at 12 o'clock).
- 7) Apply VCO1 pulse waveform into VCF and rotate resonance (Q) pot fully clockwise.
- 8) Adjust VCF "INIT FREQ" trimmer until fundamental (F1) is prominent.
- 9) JuMper CV input to pin H1.
- 10) Depress key one octave above lowest keyboard key and adjust VCF "Volt/octave" trimmer for maximum signal.
- 11) Repeat steps 1 through 5..

Roglok:

"The calibration procedure that **makers** posted is not 100% correct. I found a [scan of the original manual](#) with instructions that make more sense. Here's the relevant excerpt for adjusting the DC offset trimmers:

- Observe connector pin Q5 and adjust T9 trimmer for zero volts
- Observe connector pin Q2 and adjust T10 trimmer for zero volts

Q5 being the Lowpass output before the 1K protection resistor (or pin 6 of IC5 on moogah's schematics) and Q2 being the Bandpass output before the 1K protection resistor (or pin 6 of IC6 on moogah's schematics)...

T9 is not for adjusting HP but BP offset...