SERGE Variable Slope Filter (VCFS) for Eurorack

According to the 1983 Serge catalogue, the "*VARIABLE SLOPE VCF (VCFS) offers unique control of sound quality offered by no other synthesizer manufacturer. All VCF's offer voltage control of the cut—off frequency, that is, control of which frequencies the filter lets pass. The VCFS allows the amount of filtering to be dynamically controlled as well, from barely perceptible filtering to highly resonant, sharp cut-offs.*

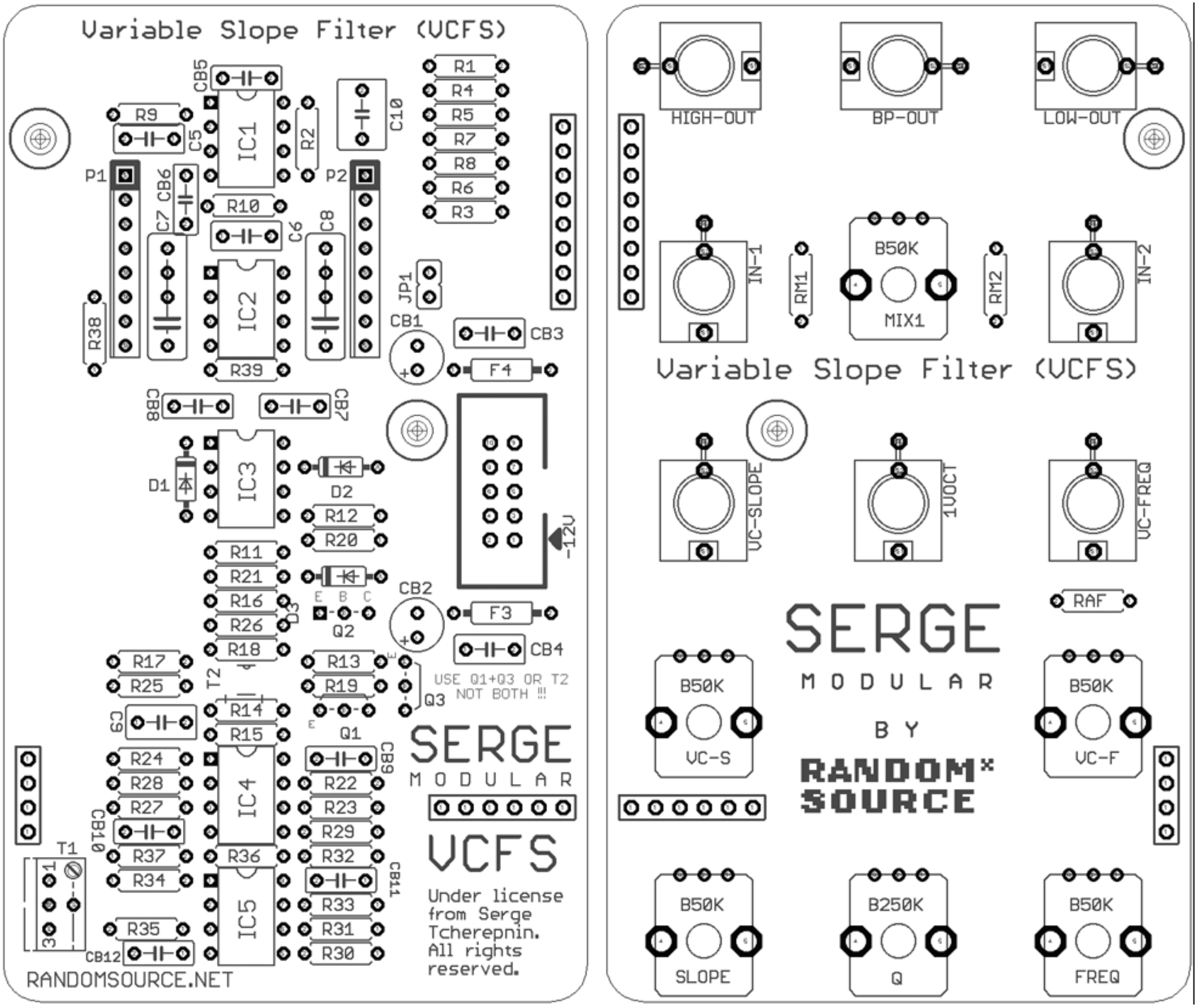
With the variable slope control in the center position, the VCFS acts as a typical flat-response VCF, with high, low, and band~pass outputs available simultaneously. The slope of the cut-off is 12 db/octave. As the control is moved toward the maximum position, the resonance of the filter increases, so that the cut-off becomes sharper.

Although the VCFS will not ring like the VCFQ, it will resonate enough at the maximum setting to pick out harmonics from a complex signal input. As the control is moved to the minimum position, the cut-off slope will decrease to 6 db/octave. This type of change of filter slope has been found to be an effective synthesis technique corresponding well with some of the transformations in acoustic instrument sounds. There are two signal inputs to the VCFS which can be mixed and manually cross faded from the associated knob."

The Random*Source version of the filter is a licensed and authorized adaption of the legendary Serge filter for Eurorack. It is an improved design that uses 2 **high-end THAT2180 VCAs** for superior audio performance - Serge Tcherepnin not only approved this, but even stated that he would have used those VCAs if they had been available in the 70s.

Moreover, the **Random*Source edition of the VCFS features an added Q knob**, which allows to (substantially) increase the resonance, and has been **optimized for an even wider range between minimum slope and maxiumum slope**. Both changes increase the sonic range covered by the filter on both ends, from very subtle to drastic.

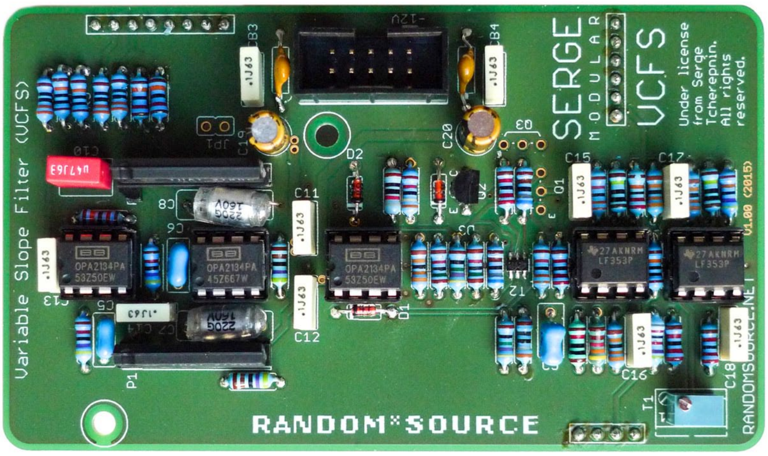
The Serge Variable Slope Filter consists of a main pcb and a matching component pcb serving as an interface to the front panel:



Please note:

- The 2180 is available in 3 different versions - A being the most selected (and most expensive) while C has slightly lower specifications, B being in between. Each version has far better audio quality than the VCAs in the original circuit, so any differences between A, B or C are neglectible.
- The module is a faithful implementation of the original Serge circuit designed for Serge voltage levels (black = bipolar, white = CV / unipolar 0-5V).
- **Pay attention to the orientation of the THAT2180s** - pin 1 is marked by a notch on top.
- The value of the resonance potentiometer determines the maximum resonance - 250k provides a nice slope range, 500k lead to quite drastic resonance behavior when turning Q up. The values of the main board have been adjusted to allow a wider slope range than the original circuit had - when slope and Q pots are turned down, the slope should be softer than the original would ever get, while slope turned up plus some Q should lead to a characteristic that's more aggressive than the original at maximum slope.

- The main pcb provides for a BCM847DS (matched transistor pair in SMT) - marked T2 - **or alternatively** two through-hole, general purpose NPN transistors (Q1, Q3), e.g. 2N3904. The original (vintage/obsolete) 2N3567 of course works as well. **Don't install the BCM847DS and Q1, Q3!** Matching the 2 transistors does not seem to make much of a difference. In other words: **you can, but you don't have to solder SMT here.**
- Pay attention to match the pinouts of the transistors to the markings on the board (**C**ollector / **B**ase / **E**mitter)! The direction of the transistor will depend on which type you use (check the datasheet!).
- SMT bypass caps on the bottom side of the main pcb are an alternative to the 100n bypass caps.
- Board is designed to be powered by a +/-12V stabilized PSU. Please **pay attention to the direction of the power header and the -12V / +12V markings**! Incorrectly connecting the power would cause considerable damage to the ICs. You may want to use resettable fuses instead of ferrit beads (marked F1, F2) on the board - see BOM below.



Bill of Materials

For some resistors which determine key characteristics of the filter, the values may be adjusted according to need/taste - there is not necessarily a “correct value” (this is DIY!).

Resistors (1%)

2	10R	F1, F2	alt: FERRIT BEAD
3	330R	R1, R7, R8	
1	390R	R25	
2	475R	R16, R17	
1	820R	R12	
1	4k32	R28	
2	4K7*	R38, R39	(*increase to 5k1 in case of a +/-15V power supply - unsupported!)
2	6.8k	R18, R19	
1	10k	R24	
2	15k	R21, R26	
4	22k	R13, R15, R34, R35	
2	22k	RM1*, RM2*	*on component pcb!
2	33k	R14, R22	
1	39k	R20	use 39k for increased slope range (if you feel that is insufficient, lower to 33k or even to 28k)
2	47k	R9, R10	
1	36k	RAF*	*on component pcb!
1	82k	R37	
3	100k	R4, R5	control input level - increase to 200k if needed
	100k	R36	
3	120k	R2, R3, R6	to bring the output level down to unity gain - original value of 220k leads to a fairly high output level (i.e. more distortion in resonance peeks)
1 (4)	220k	R29	(4 only if you use them for R2 R3, R6 as well)
4	330k	R11, R27, R32, R33	
1	470k	R30	
2	1M	R23, R31	
1	10K	T1 (1V/OCT)	Trimpot (S64YW or anything that fits) to adjust the tracking of the 1V/Oct input.

Capacitors (min. 35V)

2	15pF (Mica)	C5, C6	or 10pF, Mica or C0G , 5mm lead spacing
2	220pF	C7, C8	Styrene/Styroflex
1	270pF (Mica)	C9	or 220pF, Mica or C0G (or Styrene), 5mm lead sp.
10	100n	C3, C4, C11, C12, C13, C14, C15, C16, C17, C18	Bypass caps (alternatively, use SMT 1206 on solder side), 5mm lead spacing
1	470n (Film)	C10	WIMA MKS-2-5 or similar, 5mm lead spacing
2	10uF	C1, C2	Electrolytic

ICs

1	BCM847DS (SMT) or 2x 2N3904 (or BC550A, BC550B)	T2 Q1/Q3	Use either a BCM847DS (matched pair) in SMT or 2 single transistors (matched if possible, but matching is not essential) as Q1, Q3. Original used: 2x 2N3567 - mind the E (=Emitter) markings when using alternatives! -
1	2N3904	Q2	- mind the C B E marking! -
2	OPA2134	IC1, IC2	Burr-Brown dual op-amp
2	THAT2180	ICG1, ICG2	THAT2180B (or THAT2180A or C) Mouser: 887-2180CL08-U, 887-2180BL08-U
3	TL072	IC3, IC4, IC5	or OPA2134 as well
3	1N4148	D1, D2, D3	

Misc

1	Jumper	JP1	- ignore / leave open - (closing the jumper bypasses the Q knob)
1	Euro Power header		MTA-100 power connector, Reichelt: WSL 10G
1	SIL header 4pol		pin connectors, linking main pcb to component
1	SIL header 6pol		pcb - using precision strips allows to break off
1	SIL header 8pol		pieces as needed
8	Thonkiconn Jacks		3.5mm Jack Sockets (PJ301M-12) from Thonk
5	Potentionmeter 50k or 100k	linear (B50K or B100K)	Alpha 9mm vertical pcb mount available from Thonk, Tayda
1	250k	linear (B250k) - Q-Potentiometer	Alpha 9mm vertical pcb mount available from Thonk, Tayda Controls amount of (additional) resonance

Building

This is simply a suggestion - you might find a different workflow more practical:

1. Use a side-cutter to separate main pcb and component pcb.

2. Mount the Thonkiconn jacks, the pots and the switch onto the component pcb. Pots should sit on the side facing the front panel (as marked on the board). Don‘t solder them in yet.

3. Carefully mount the component pcb (with the pots etc. inserted) onto the front panel. You may then have to wiggle each pot a bit to get the pots through. Make sure the threads of the pots go through completely and the pots sit right at the front panel. You can even screw the jacks and pots to the panel to make sure of that, but you will have to unscrew them again later.

4. Once everything is nicely in place, solder the pots, jacks und switch onto the component pcb (while the front panel is attached).

5. Stuff the main board, beginning with the resistors, then caps etc.

6. Main pcb and component pcb are to be connected through precision DIP socket and pins. It is recommended to use the pins on the main pcb (facing down, soldered from above) and the pin sockets on the component pcb (standing up, soldered from the front panel side). Break or cut off the pieces you need and stick them together so that main pcb and component pcb form a nice sandwich (don‘t solder yet). Check that you didn‘t leave out any pins / holes and that the sockets are all on the same side (component pcb). Solder all the pins in while keeping the sandwich together - this avoids any misalignments.

7. Carefully separate the sandwich - if you used precision sockets, this may not to too easy - they stick together nicely (giving a good connection).

8. Mount the component pcb onto the front panel again and screw on the pots from the front side.

9. Make sure everything is in place.

10. Attach any screws / spacers if desired and mount the main pcb onto the component pcb.

11. Connect a power cord supplying +12V, GND, GND, -12V to the power-header on the main board and double check the direction of the power header before you turn power on. you should be ready to go :-)

Calibration

The main pcb provides a single trimpot that allows adjusting the **1V/Oct** input. As the filter does not self oscillate by design, calibrating the 1V/Oct tracking precisely is tricky and probably not too relevant. The 1V/Oct input is a musical feature, designed to simulate the behavior of acoustic instruments. The best way might be to simply do this by ear.

Also, be aware that this input has no temperature compensation.

Power Consumption

Power consumption: appr. 40mA @ +12V and 40mA @ -12V

(Version 27 January 2016)