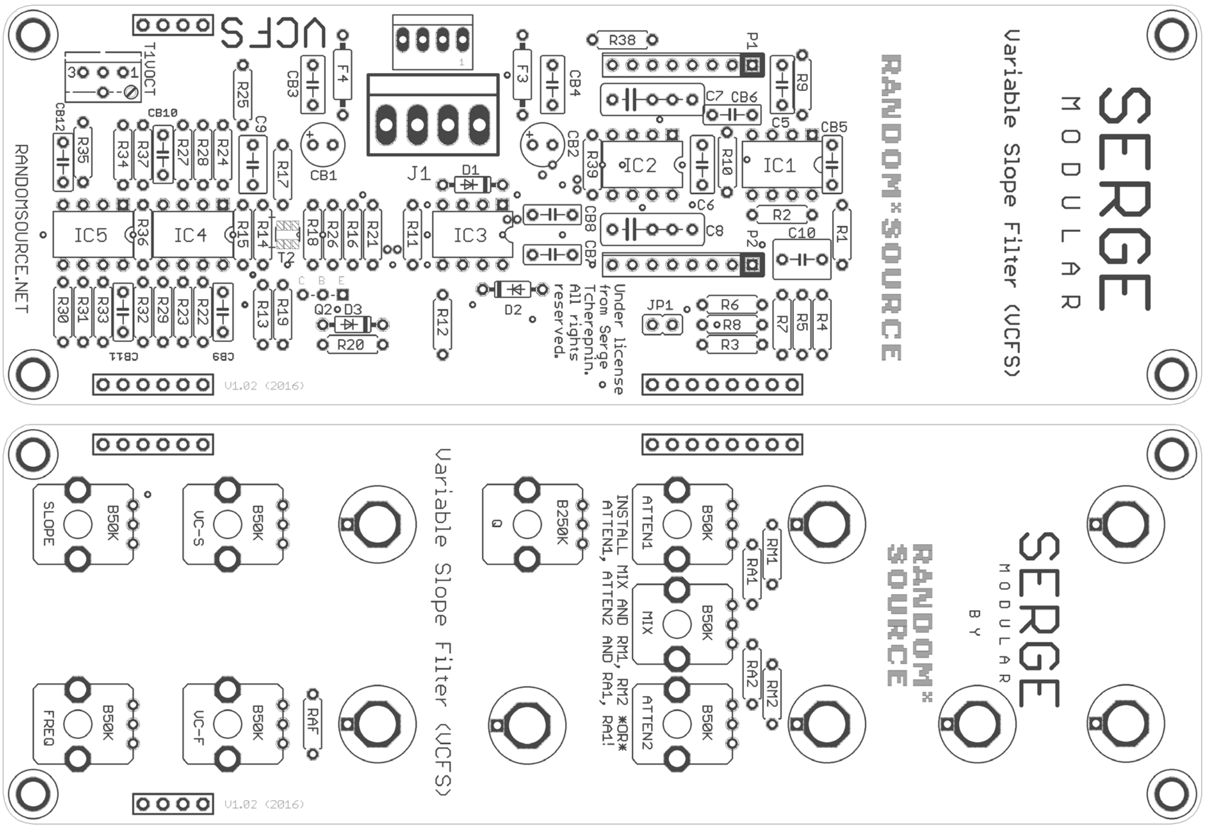
SERGE Variable Slope Filter (VCFS) Version 1.02

The Random*Source version of the filter is an updated design that uses 2 **high-end THAT2180 VCAs** for superior audio performance and provides **variable Resonance (Q)** to shape overall filter characteristic - both changes have been approved by Serge Tcherepnin.

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

Main pcb and panel pcb (v1.02 - please refer to the previous version of this document for v < 1.02!):



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 value of the resonance potentiometer determines the maximum resonance - 250k provides a nice slope range, 500k (or even 1Meg) 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 v 1.02 provides for a BCM847DS (matched transistor pair in SMT) - marked T2. This is the only SMT part required to be soldered. It is recommended to solder it first - before populating the resistors, otherwise it's harder to reach.
- The BOM below shows a (fancy) reference build featuring some slightly radical part choices compared to the vintage versions (in particular omitting various capacitors, original values are shown as well) that should work well as long as you also use the (high-end) op-amps specified. Feel free to experiment with other op-amps, but bear in mind that you may have to add capacitors as compensation.
- Pay attention to match the pinouts of the transistor Q2 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!).
- Orientation of the main pcb: **power header is at the bottom** (when looking at the module upright, e.g. in a rack).
- Board is designed to be powered by a +/-12V stabilized PSU only
(+/-15V is untested/unsupported and will require some value changes to not kill the THAT2180s).

Bill of Materials

The following resistor values are suggestions - for a number of resistors, which determine key characteristics of the filter, ther values should be adjusted according to need/taste. The values stated here should be understood as a starting point rather than the "correct value" (this is DIY!).

Resistors (1%)

2	BEAD / 10R	F1, F2	FERRIT BEAD
3	330R	R1, R7, R8	
1	390R	R25	
2	475R	R16, R17	
1	820R	R12	
1	4k32	R28	or 4k02
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	36k	R20	use 36k 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 if you use them for R2m 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

2	OMIT (15pF)	C5, C6	Original design: 15pF or 10pF Mica - install C0G if needed
2	220pF	C7, C8	Styrene/Styroflex - match for best performance
1	OMIT (270pF)	C9	Original design: 270pF Alt: 220pF, Mica or C0G or Styrene
10	100n	C3, C4, C11, C12, C13, C14, C15, C16, C17, C18	Bypass caps
1	470n (Film)	C10	
2	10uF	C1, C2	Electrolytic

ICs

1	BCM847DS	T2	SMT
1	2N3904	Q2	
1	OPA2227 or OPA2134	IC2	Burr-Brown dual op-amp
2	OPA2134	IC1, IC3, IC4, IC5	Burr-Brown dual op-amp
2	THAT2180	ICG1, ICG2	THAT2180B (or THAT2180A or C)
3	1N4148	D1, D2, D3	

Misc

1	Jumper	JP1	JUMPER - leave open - (closing makes Q-pot useless)
1	MTA-156		MTA-156 power connector
2	SIL header 10pol		10-pin connector, links main pcb to component pcb

0	Banana Jacks	TRIG IN (red)	Emerson-Johnson Mouser: 530-108-0902-1 (red) or Thonk
5	Banana Jacks	IN, BIPOLAR (black)	Emerson-Johnson Thonk / Mouser: 530-108-0903-1 (black)
3	Banana Jacks	CV / unipolar (blue or white)	Emerson-Johnson Thonk / Mouser: 530-108-0910-1 (blue), 530-108-0901-1 (white)
5	Potentionmeter 50k or 100k	linear (B50K or B100K)	Alpha 9mm vertical pcb mount available from Thonk, Tayda
1	Potentionmeter 250k (or 500k)	linear (B250K or B500K)	Resonance (Q) - see text! Alpha 9mm vertical pcb mount available from Thonk, Tayda

Building

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

1. Mount the Banana jacks, and the switch onto the front panel.

2. Screw (10mm) spacers to the panel pcb if desired - this is easiest done while the panel pcb is not yet attached to the front panel.

3. Main pcb and panel 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 panel 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 (panel pcb). Also make sure the pcbs have the right orientation (so that the pots will sit outside!). Solder all the pins in while keeping the sandwich together - this avoids any misalignments.

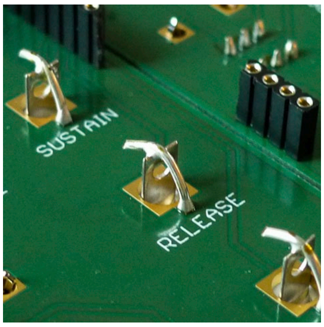
4. Carefully separate the sandwich - if you used precision sockets, this may not to too easy - they stick together nicely (giving a good connection).

5. Solder the resistors onto the panel pcb.

6. Mount the pots onto the component pcb. Pots should sit on the printed side - this side faces the front panel. Don‘t solder them in yet.

7. Carefully mount component pcb (with the pots 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 pots to the panel to make sure of that, but you have to unscrew them again later (for Step 5).

8. Solder the banana jacks in. You can either solder them directly to the surrounding vias (rings around) or - which makes removing easier should you ever need to do that - by inserting a stiff (bare) wire into the little hole (via) and solder that wire to the top of the banana jack:



9. Solder the SMT part (BCM847DS) onto the main board. Using thin (0.5mm) solder wire recommended.

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

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

12. Connect a power cord supplying +12V, GND, GND, -12V to the MTA-header on the main board and 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. Also, be aware that this input has no temperature compensation. The best way might be to simply do this by ear.

(Version 11 January 2017)