SERGE Resonant Equalizer (EQ) for Eurorack

The RESONANT EQUALIZER (EQ) is a unique ten-band filter designed specifically for electronic sound synthesis and processing. Except for the top and bottom frequency bands, the bands are spaced at an interval of a major seventh. The Resonant Equalizer is designed to produce formant peaks and valleys similar to those in acoustic insstruments.

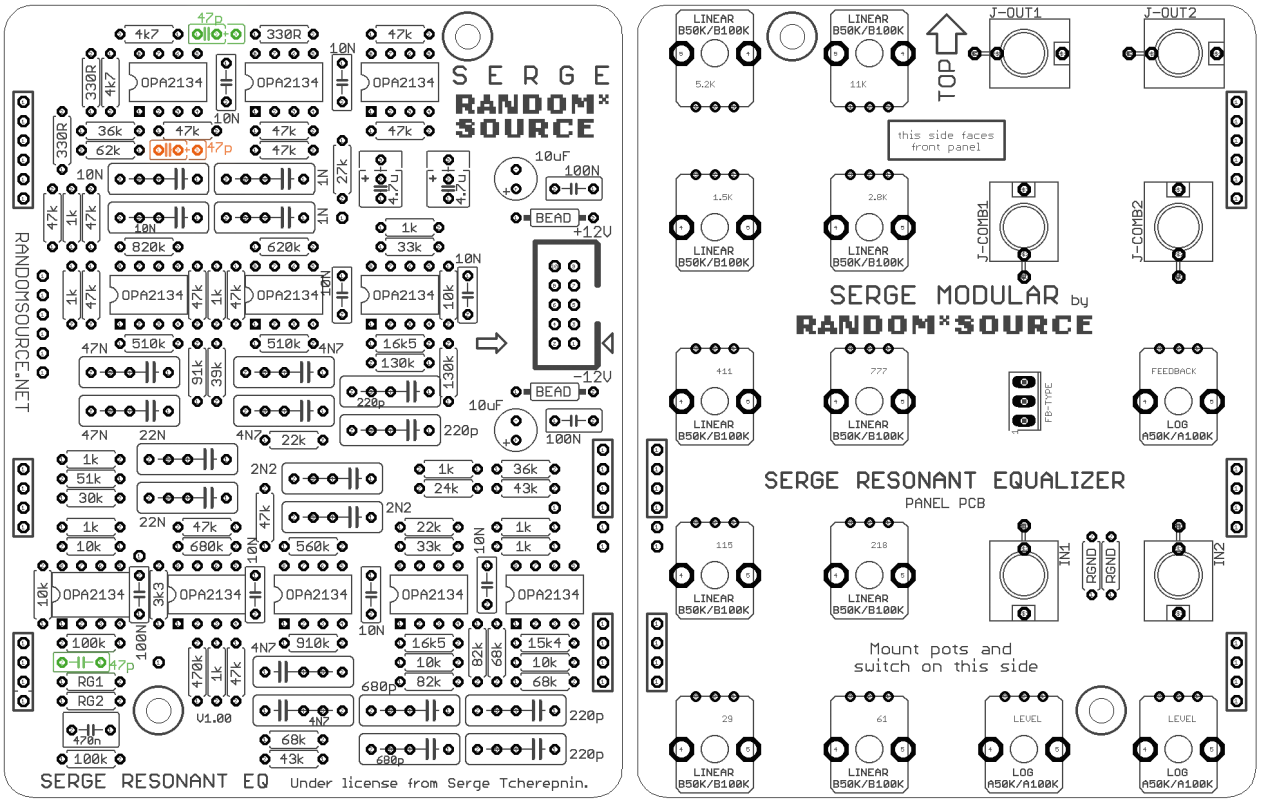
There are three equalized outputs: the two COMB outputs provide the sums of the two alternate filter bands, while the two top outputs are the (identical) mix of all filter bands. Please note that there is no sharp separation between the bands, moving any frequency knob will have an influence on both COMB outputs.

COMB1 provides the outputs of the (left) frequency bands at 29 Hz, 115 Hz, 411 Hz, 1.5 kHz and 5.2 kHz while COMB2 mixes the bands at 61 Hz, 218 Hz, 777 Hz, 2.8 kHz, and 11 kHz. This equalizer differs from other equalizers in that the bands can be set to be resonant. When the knobs are in the middle position, the response at the main EQ Output is flat. When the knobs are positioned between the 9 and 3 o'clock position, up to 12 db of boost or cut is set at the band. If the knob is set beyond the 3 o'clock position, the band will become resonant, simulating the natural resonance of acoustic instrument formant structures. Below the 9 o'clock position, increased band rejection is achieved.

The Random*Source version of the filter is a licensed and authorized adaption of the legendary Serge equalizer for Eurorack. It adds an input mixer with 2 inputs that can be built/used to attenuate or amplify/ distort the input signal(s) before they are sent into the EQ and a Feedback section (knob and phase switch) which allows to feed back the output signal or the inverted output signal (depending on the Phase switch). Adding inverted feedback leads to phase cancellation effects - unlike the "normal" feedback, level decreases and the signal thins out.

The Random*Source version also specifies some of the best parts and components available today for optimum audio performance.

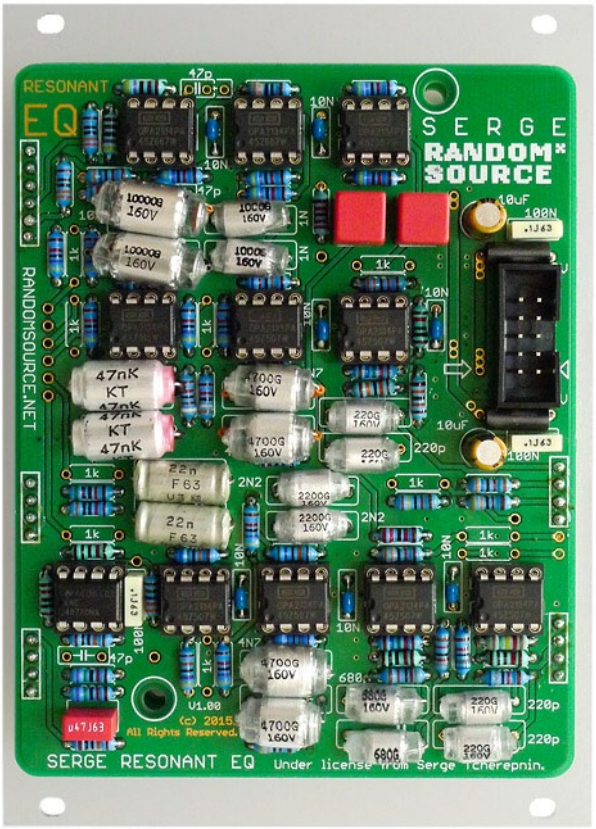
The Serge Resonant EQ kit consists of a main pcb and a matching component pcb serving as an interface to the front panel:



Please note:

- For best audio performance, it is essential to use high-quality parts. For the op-amps, Burr-Brown OPA2134 is a good choice. Alternatively feel free to try (more expensive) LM49720 or OPA2604 ICs. For the capacitors, it is recommended to use Styrene/Styroflex (ideally with low tolerances) where possible - the values 22N and 47N may be hard to find as Styrene, alternatively use C0G or Film.
- Resistors RG1 and RG2 set the maximum gain level of each of the two inputs. 100k will make the input knobs pure attenuators, i.e. full CW is 100% of the input signal (unity gain). 75k will allow a maximum amplification of +33% (100k/75k = 133.3%), i.e. unity gain will be at around 2 o'clock position of the knob. Similarly, 68k will allow a maximum amplification of +47%. Depending on the input signal's level, this can be used to add distortion / clipping before entering the EQ. How that distortion will occur (and sound) depends on the op-amp used - that's the one above RG1 in the picture above. The OPA2604 has a nice clipping behavior.
- The main pcb provides for three 47pF capacitors to prevent unwanted self-oscillations of the op-amps. The 47pF shown in orange above seems to be essential, the other two shown in green are probably not necessary, so it's recommended to start with the orange one and install the others if need be. Beware, the EQ and in particular the feedback option is designed to generate all kinds of (self-)oscillations, making it not so easy to distinguist unwanteded self-oscillations from the "normal" oscillations, especially those generated by the high frequency bands.

- SMT bypass caps on the bottom side of the main pcb are an alternative to the 10n bypass caps.
- The module has a power consumption around 100mA on each rail. That means **a (standard) resettable fuse for a current of 0.1A cannot be used**. Use ferrit beads instead (or a resettable fuse with a theshold of at least 150mA).
- 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 - especially if you use ferrit beads. (Resettable fuses instead of ferrit beads may prevent that, but make sure you get the right kind - see above.



Bill of Materials

Resistors (1%)

2	BEAD	Ferrit Beads. Alternativly, resettable Fuse (PPTC with I_{Trip} >= 150mA should work here - untested! - e.g. Bourns MF-R010-0 - carefully bend legs to fit the footprint)
2	RG1, RG2	Sets maximum amplification of input mixer (attenuator) - see text above - 68k (for +47%) or 75k (for +33%) recommended, 100k for no amplification
2	RGND	100k (optional - normalizes unused inputs to GND to avoid noise)
3	330R	
10	1k	NOT NEEDED - OMIT
1	3k3	
2	4k7	
5	10k	
1	15k4	
2	16k5	
2	22k	
1	24k	
1	27k	
1	30k	
2	33k	
2	36k	
1	39k	
2	43k	
13	47k	
1	51k	
1	62k	
3	68k	
2	82k	
1	91k	
2	100k	
2	130k	
1	470k	
2	510k	
1	560k	
1	620k	
1	680k	
1	820k	
1	910k	

Capacitors (min. 35V, 5mm lead spacing except Styroflex / electrolytic caps)

2	47p	C0G, 5mm lead spacing - (optional)
1	47p	C0G, 2.5 or 5mm lead spacing. Install!
7	10N	Bypass caps (C0G or other) - between ICs
3	100N	Bypass caps (Film)

11	10N / 100N	SMT (1206 or 0805) bypass caps (optional / alternative)
4	220p	Styrene/Styroflex
2	680p	Styrene/Styroflex
2	1N	Styrene/Styroflex
2	2N2	Styrene/Styroflex
4	4N7	Styrene/Styroflex
2	10N	Styrene/Styroflex
2	22N	Styrene/Styroflex or C0G, Film
2	47N	Styrene/Styroflex or C0G, Film
1	470N	Film (WIMA or similar)
2	4.7uF	Film (e.g. WIMA)
2	10uF	Electrolytic

ICs

11	OPA2134	Burr-Brown dual op-amp, alternatives: OPA2604, LM49720

Misc

1	Switch SPDT	Phase (for Feedback)	Sub-Miniature Switch, sets the Phase of the feedback added with the knob, e.g. Mountain Switch. 2 positions (ON - NONE - ON, Mouser: 108-0042-EVX) means you have to turn the knob down for no feedback, 3 positions (ON - OFF - ON, Mouser: 108-0044-EVX) means center positions turns feedback off.
1	Euro Power header		MTA-100 power connector, Reichelt: WSL 10G
1	SIL header 6pol		pin connectors, linking main pcb to component
4	SIL header 4pol		pcb - using precision strips allows to break off pieces as needed
6	Thonkiconn Jacks		3.5mm Jack Sockets (PJ301M-12) from Thonk
10	Potentionmeter 50k or 100k	linear (B50K or B100K) for the 10 frequency bands	Alpha 9mm vertical pcb mount available from Thonk, Tayda
3	Potentionmeter 50k or 100k	logarithmic (A50K or A100K) for IN1, IN2, FEEDBACK	Alpha 9mm vertical pcb mount available from Thonk, Tayda

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. Solder the 2 resistors onto the component pcb.
3. 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.
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. 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.
6. 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. Screw the jacks and pots to the panel.
7. Once everything is nicely in place, solder the pots, jacks und switch onto the component pcb (while the front panel is attached).
8. Stuff the main board, beginning with the resistors, then caps etc.
9. Attach any screws / spacers if desired and mount the main pcb onto the component pcb.
10. 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

No calibration required.

Power Consumption

Power consumption: appr. 100mA @ +12V and 100mA @ -12V

(Version 15 December 2015)