

Description

This is a PCB for a 4-channel “attenuverter”, i.e. 4 buffered attenuators that can also invert the signal. It further has a mixing section with sum output of all attenuverters and an additional unity-gain auxiliary input.

By default, the first attenuverter input is normalled to a fixed 5V reference, making the unit a flexible fixed voltage source or CV processor with offset. Solder jumpers allow the other channels to be also normalled to the 5V reference.

The PCB fits behind a 6hp eurorack panel, with all controls and circuitry integrated on the single PCB. It uses 0805-size SMD components and SOIC-size ICs. If you are comfortable soldering basic SMD parts, this should be an easy build.

Features

- Four attenuverters
- Adjustable control curves
- Offset voltage option
- Optional zero trimmers for use with center-detent potentiometers
- 9mm pots, Thonkiconn jacks
- Single PCB construction
- 0805 and SOIC SMD
- 6hp Eurorack format

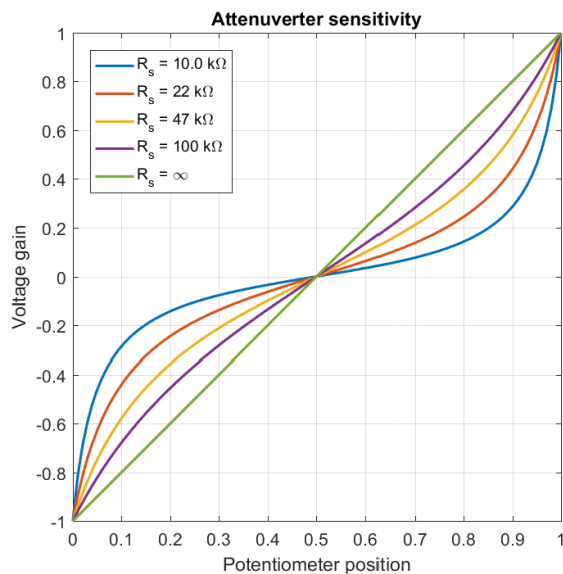


Schematics, PCB layout and documentation © 2019 Caspar Ockeloen-Korppi.

Special features and build notes

Adjustable control curves

A special feature of this circuit is that the attenuverter response curve can be chosen by using different resistor values, as shown in the following figure:



With the suggested value of $R_s = 47\text{ k}\Omega$, the potentiometer curve will be “logarithmic” like (yellow curve). This gives extra sensitivity near the center for small amounts of modulation. The value of R_s can be adjusted for a stronger or weaker logarithmic effect, or R_s can be omitted completely to get the standard linear response (green curve). Note that there are 8 resistors in total for this purpose: R22 through R29 all have the value R_s .

Fixed voltage reference

The board includes a fixed 5V reference, comprised of U4 and R30. By default, this voltage is connected to the normal (switch) contact of the first attenuverter input. Hence, if no jack is connected to that input, the topmost knob generates a constant voltage between -5 V and +5 V. It can be either be used directly from the

first channel output, or combined with other channels through the sum out jack.

If the offset function is not desired, simply leave out U4 and R30. If you want to have the offset function also on other channels, there are solder-bridge jumpers J13, J14 and J15 behind each input jack. Bridge the pads with a blob of solder to connect the normal contact of the respective input jack to +5V. Similarly, there is a jumper J12 behind the first input jack. J12 is pre-connected with a short track between the pads; cut the track to disable the jumper, and use a solder blob to re-enable it if needed.

Zero trimming potentiometers

There is provision for a zero trimming potentiometer on each channel. These are only useful if you use center-detent potentiometers, and allow you to set exactly¹ 0 gain when the pot is centered (without a center detent, you can adjust the knob position to achieve this goal). Note that the trimmers replace the R_s resistors: a 100k trimmer has the same effect as $R_s = 47\text{ k}\Omega$. If you can’t find 100k center-detent pots, any other similar value will do (say in the 50k to 1M range). Keep the trimmer value the same as the potentiometer value.

Component selection

- If you use the trimmer pots, they should be of the Bourns 3296X style, or compatible (I use T910X series from TME). The X version has the screw on the side, which is important so the trimmer fits under the front panel.
- Capacitors C1 and C2 must not exceed the recommended diameter and height, otherwise they won’t fit. Nichicon UST1H100MDD is known to fit.

¹Note that Alpha center detent potentiometers have about 1% or so of backlash and repeatability error, so every time you center the potentiometer you will get a slightly different result.

Alternative builds

The same PCB can be used to make normal (non-inverting) buffered attenuators or a mixer. Some examples are:

1. Buffered attenuators with logarithmic-like (audio) response:
Do not place R2,R5,R8,R11,R22,R24,R26,R28,RV1,RV3,RV5,RV7 (leave open)
2. Buffered attenuators with linear response:
In addition to alternative 1, also do not place R23,R25,R27,R29.
3. Four-channel mixer (either log or linear):
In addition to alternative 1 or alternative 2, also do not place the output jacks J2,J4,J6,J8. Place a wire link between the tip and normal connections of J2,J4,J6,J8. Use the mixer panel layout, which has fewer holes.

In these applications you may also want to leave out the 5V reference circuit (do not place U4 and R30).

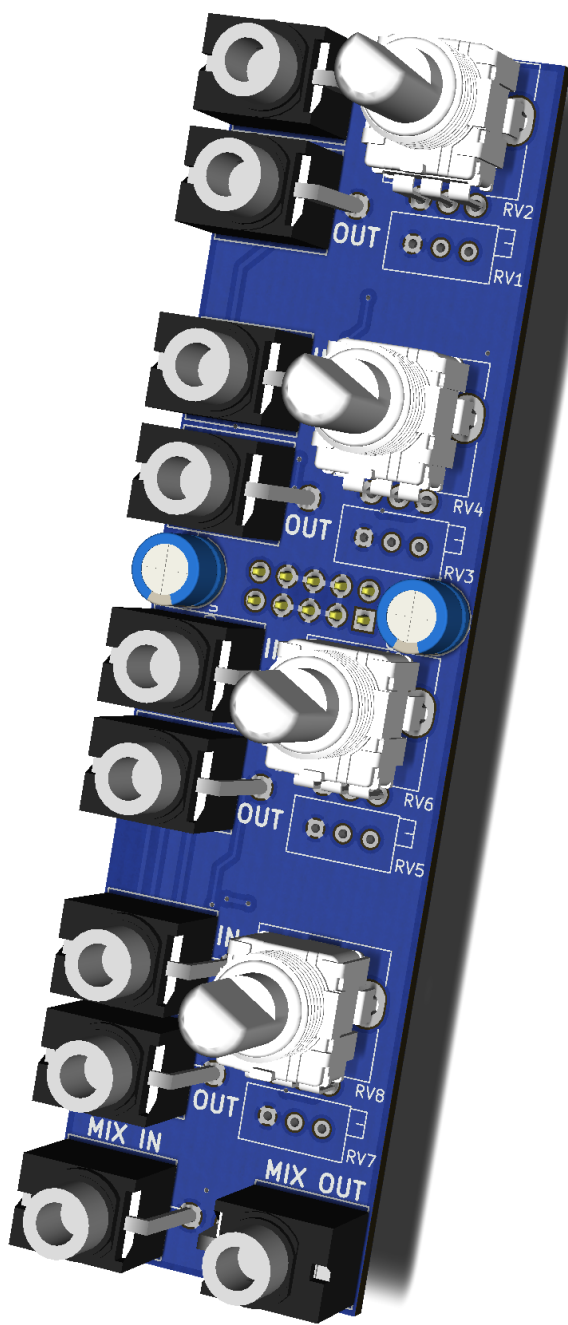
Important notes for Rev 1.0 PCBs

- The rev 1.0 PCB and schematic specify 100 μ F power supply capacitors. This is overkill, I recommend 10 μ F (C1 and C2). Note the package dimensions in the BOM. I also recommend using a maximum height of 9mm and placing them on the front side of the board (the silk screen for C1 and C2 is on the back side). That way there is enough space to plug in the power connector without hassle. I use Nichicon UST1H100MDD.

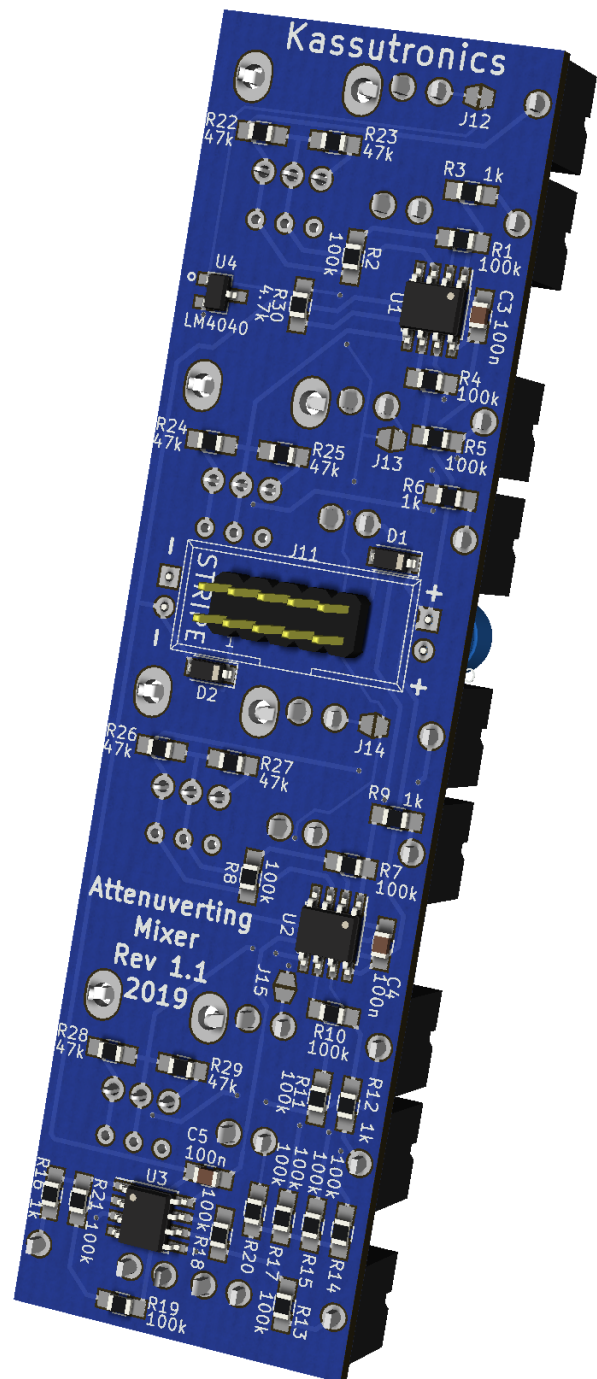
Bill of materials

Qty	Designator	Value	Note
2	C1,C2	10u	Diameter 6.3 mm, pitch 2.5mm, max height 9mm. I recommend Nichicon UST1H100MDD.
3	C3,C4,C5	100n	X7R ceramic, 0805
2	D1,D2	BAT54J	Any Schottky diode in SOD-323 or SOD-123 package
10	J1,J2,J3,J4,J5,J6,J7,J8,J9,J10		Thonkiconn jack
1	J11		2x5 pin header, boxed or unboxed
5	R3,R6,R9,R12,R16	1k	All resistors 0805 SMD
1	R30	4.7k	
8	R22,R23,R24,R25,R26,R27,R28,R29	47k	Value = Rs (see documentation)
16	R1,R2,R4,R5,R7,R8,R10,R11,R13, R14,R15,R17,R18,R19,R20,R21	100k	
4	RV2,RV4,RV6,RV8	100k	Alpha 9mm vertical
4	RV1,RV3,RV5,RV7	100k	Optional, replaces R22-R29 (see documentation)
3	U1,U2,U3	TL072**D	SOIC package
1	U4	LM4040 5V	SOT-23 package (DMZ)

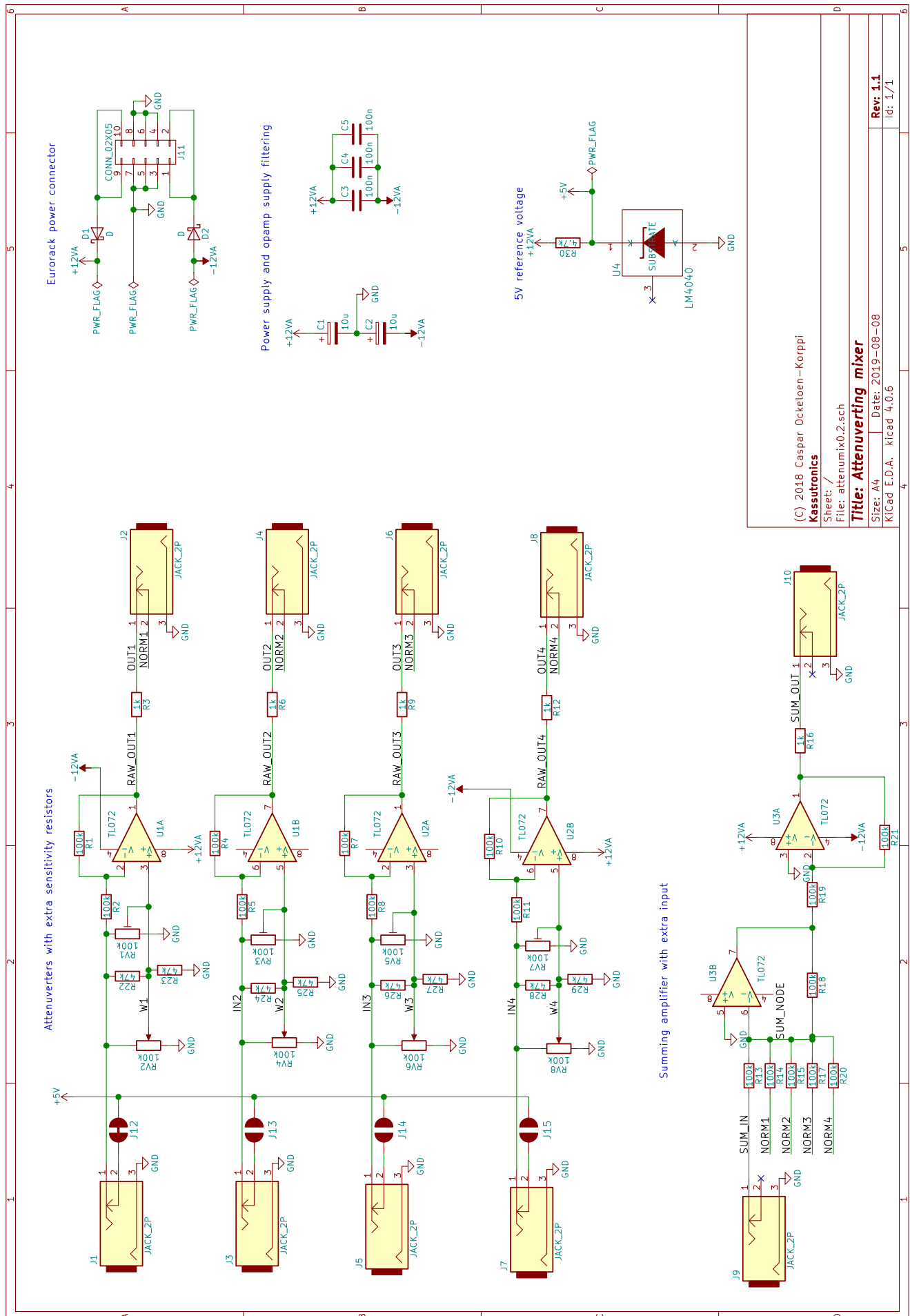
Board view



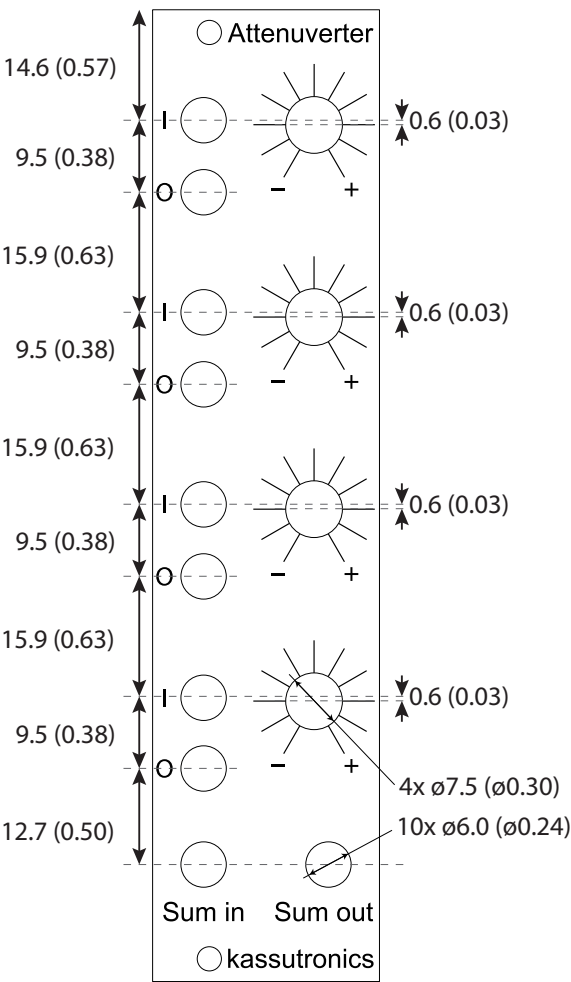
Front



Back



Front panel dimensions



Dimensions in mm (inch)

Revision history

Board revisions

- 1.0 Initial design.
- 1.1 Added 5V reference (U4, R30); added reverse voltage protection (D1, D2); re-arranged power header to fit shrouded headers; improved silkscreen

Documentation revisions

- A Initial documentation for board revision 1.0
- B Re-formatted small fixes to documentation.
- C Fixes to alternative build 3.
- D Updated for PCB revision 1.1; added front panel dimensions

Contact

Check for updated documentation and other information on my blog at kassu2000.blogspot.com.
I am always happy to answer questions and receive feedback at kassutronics@gmail.com.