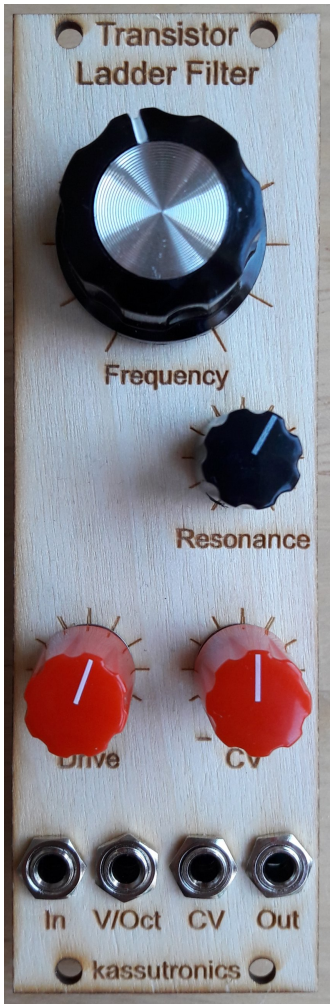
Transistor Ladder Filter

Kassutronics

Board revision 1.1, documentation revision E

May 15, 2020

Description



This is a PCB for a classic Minimoog-style transistor ladder low pass filter in Eurorack format. The main design goal was to make a compact filter (8hp) that is easy and affordable to build. For this reason, front panel controls and main circuitry are integrated on a single PCB, and all components are through hole (but see build choices below).

Internally there are a Minimoog-style core and an opamp output stage. Many thanks to Yves Uson (yusynth.net) for sharing his version of the filter, the core is based on his implementation and was the starting point for me to learn synthesizer DIY. Apart from the obvious frequency and resonance pots, there is a V/Oct CV input (which can track over a few octaves but has rather limited temperature compenstation), an CV input with attenuverter and a Drive control.

Current usage (typical): +12V 20mA, −12V 10mA.

Note: This documentation is valid for both Rev 1.0 and Rev 1.1 PCBs. If you have a Rev 1.0 board, a few component values should be changed, see the special section below.

Please read this document through before starting, so you place the correct values and buy the right components!

Features

- Classic 4-pole low pass filter
- V/Oct input
- FM input with attenuverter
- Drive (input level) control
- 9mm pots, Thonkiconn jacks and all throughhole components on a single PCB
- 8hp Eurorack format

Build instructions

Optional components and build choices

THT or SMD transistors: There are two options for the ladder transistors, depending on your preference:

1. Five hand-matched pairs (so 10 transistors total) in standard TO-92 through-hole package. The specified transistor is BC547C, but most small-signal NPN transistors should work just fine[1]. Each pair should be matched to 2 mV or better in VBE. Matching transistors is easy, just buy plenty (say 50 or 100) and with one evening of work you will have enough matched pairs for a huge synth. For instructions and more info, see my blog article: http://kassu2000.blogspot.fi/2015/10/transistor-matching.html.
2. Five dual matched transistor packages BCM847BS, in tiny SOT-363 surface mount package. These are to be soldered at the SMD pads between the regular THT transistor pair locations. The long silkscreen line indicates pin 1.

Just to emphasize, choose either option 1 or 2 and not both!

Optional trimmer RV5: There is a trimmer RV5 that allows to zero the CV attenuverter. This is only really useful if you use a center-detent pot for the attenuverter. RV5 has a second function though: it makes the attenuverter curve more "logarithmic" towards both sides, making it easier to apply small amounts of modulation. If you omit RV5, the attenuverter curve is linear.

Value changes for Rev 1.0 boards

If you have a Rev 1.0 PCB, there are a few values wrong on the silkscreen. Please make the following changes, the updated values agree with the Rev 1.1 PCB and schematic.

Designators	Silkscreen value (wrong)	Correct value
R9	15k	10k
R28	10k	6.8k
C8, C10	100 μF	10 μF

Component and build notes

- The board is designed for small metal film transistors, with 5.08mm (0.2") hole spacing. These have a body length around 3.7 mm, and are usually either 1/8 W or 0.4 W rated. Standard 0.6W metal film resistors will fit if you place them vertically.
- While not critical, it is often suggested to hand-match the filter capacitors C1 – C4. To do this, measure a bunch of caps with a multimeter (in capacitance mode), and choose four that are most similar. Accuracy of a few percent is fine.
- Pots are standard ALPHA 9mm vertical type, available at e.g. Thonk or Tayda.
- Jacks are Thonkiconn.
- Trimmers are the Bourns 3296X multiturn type, or compatible (I use T910X series from TME). The X is important: it means the screw is sideways. Other versions with vertical screw don't fit under the front panel, however if needed they can be mounted on the backside of the PCB
- The 10uF caps should be 2.5mm lead spacing, max diameter 6.3mm and max height 9mm. I use Nichicon

[1] For example 2N3904. Beware that the pinout of the 2N3904 is the reverse of the BC547, so they should be installed rotated 180 degrees.

UST1H100MDD. They should be installed on the front side of the board, even though on Rev 1.0 boards the silkscreen is on the back. The square pad is the positive lead.

- The 220uF capacitor (Diameter max 8.0mm, Lead spacing 3.5mm) should be placed on the back of the board as indicated on the silkscreen (because it is too tall to fit under the panel)
- The power header is a 5x2-pin 2.54mm unboxed header, and should be installed on the backside of the board.

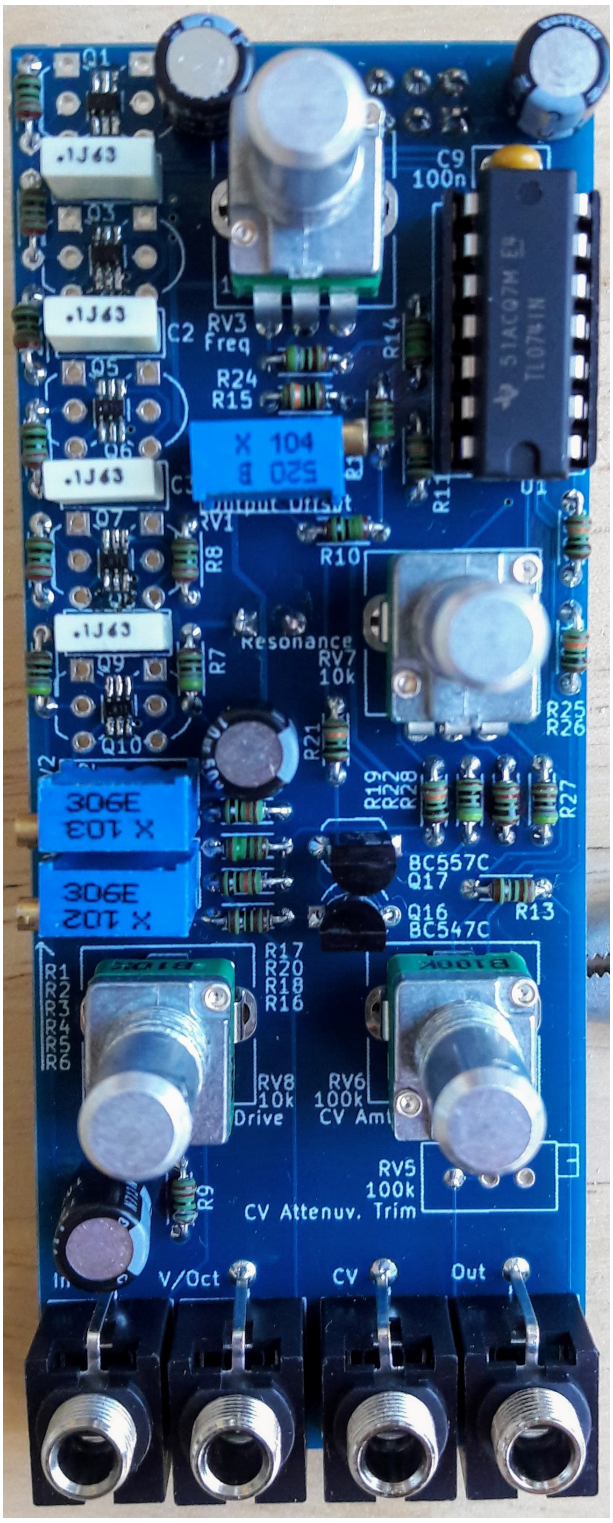
Trimming

- Output offset (RV1) should be adjusted such that the output voltage is 0 with no input
- V/Oct trim (RV4) should be adjusted with the resonance at maximum by trial and error with a CV keyboard. I recommend trimming it for tracking around the 100-400Hz range, and don't expect good tracking at high octaves.
- Frequency offset (RV2) can be used to adjust the frequency range
- If installed, the attenuverter trim (RV5) can be used to zero the CV input when the control is at center position. Apply a CV signal like an LFO and adjust until it has no effect.

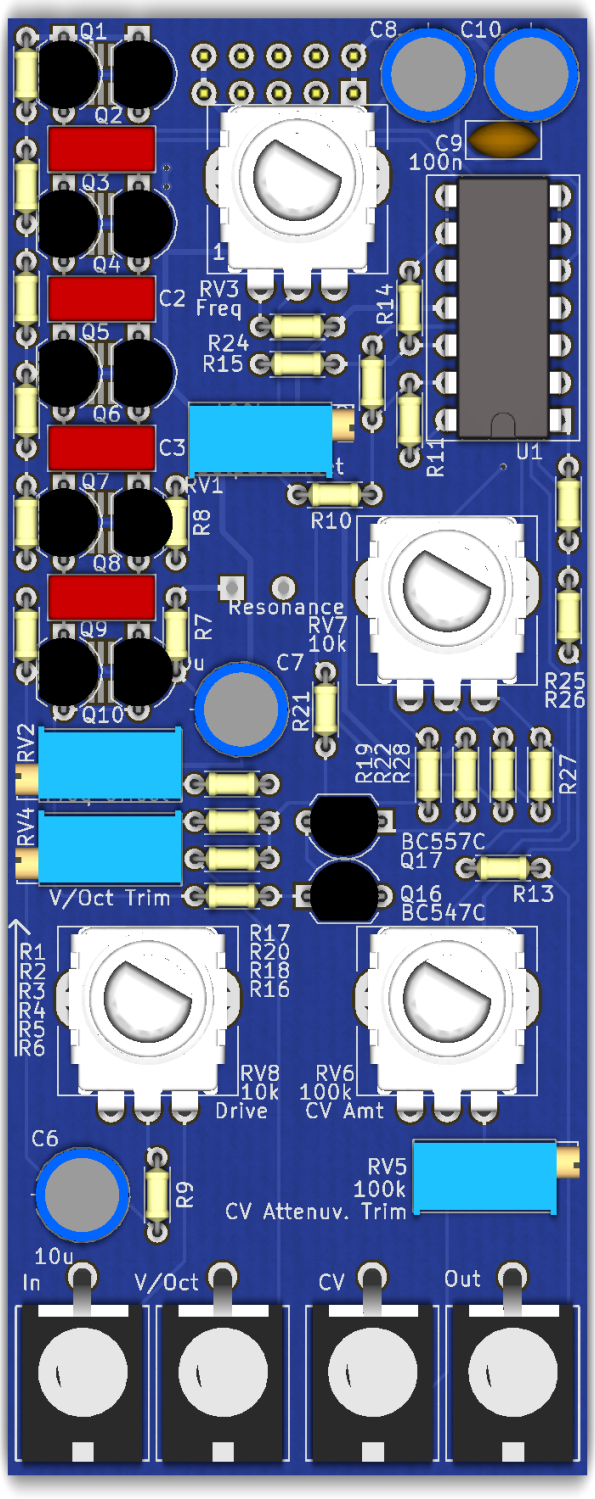
Bill of materials

Qty	Designator	Value	Note
4	C1,C2,C3,C4	47n	Film, 5mm pitch
1	C9	100n	X7R ceramic, 2.5mm pitch
4	C6,C7,C8,C10	10u	Min. 25V, 2.5mm pitch, max dia. 6.3mm, max height 9mm. Example: Nichicon UST1H100MDD
1	C5	220u	Min. 25V
4	J1,J2,J3,J4	Thonkiconn	
1	J5		Unboxed pin header
10	Q1 – Q10	BC547C	Match pairs Q1&Q2, Q3&Q4, ...
5	Q11 – Q15	BCM847BS	Optional: alternative to Q1-Q10
1	Q16	BC547C	
1	Q17	BC557C	PNP
6	R1,R2,R3,R4,R5,R8	220	All resistors 1% metal film small type (1/8 W or 0.4W)
3	R6,R7,R22	470	
1	R20	1.2k	
2	R12,R14	1.5k	
8	R11,R15,R17,R18, R19,R21,R25,R26	100k	
1	R28	6.8k	
1	R9	10k	
2	R13,R16	1k	
1	R10	1M	
2	R24,R27	4.7k	
1	RV1	100k	3296X or T910X
1	RV2	10k	3296X or T910X
1	RV4	1k	3296X or T910X
1	RV5	100k	Optional. 3296X or T910X
3	RV3,RV7,RV8	10k linear	Alpha 9mm vertical
1	RV6	100k linear	Alpha 9mm vertical
1	U1	TL074	DIP, add socket if desired

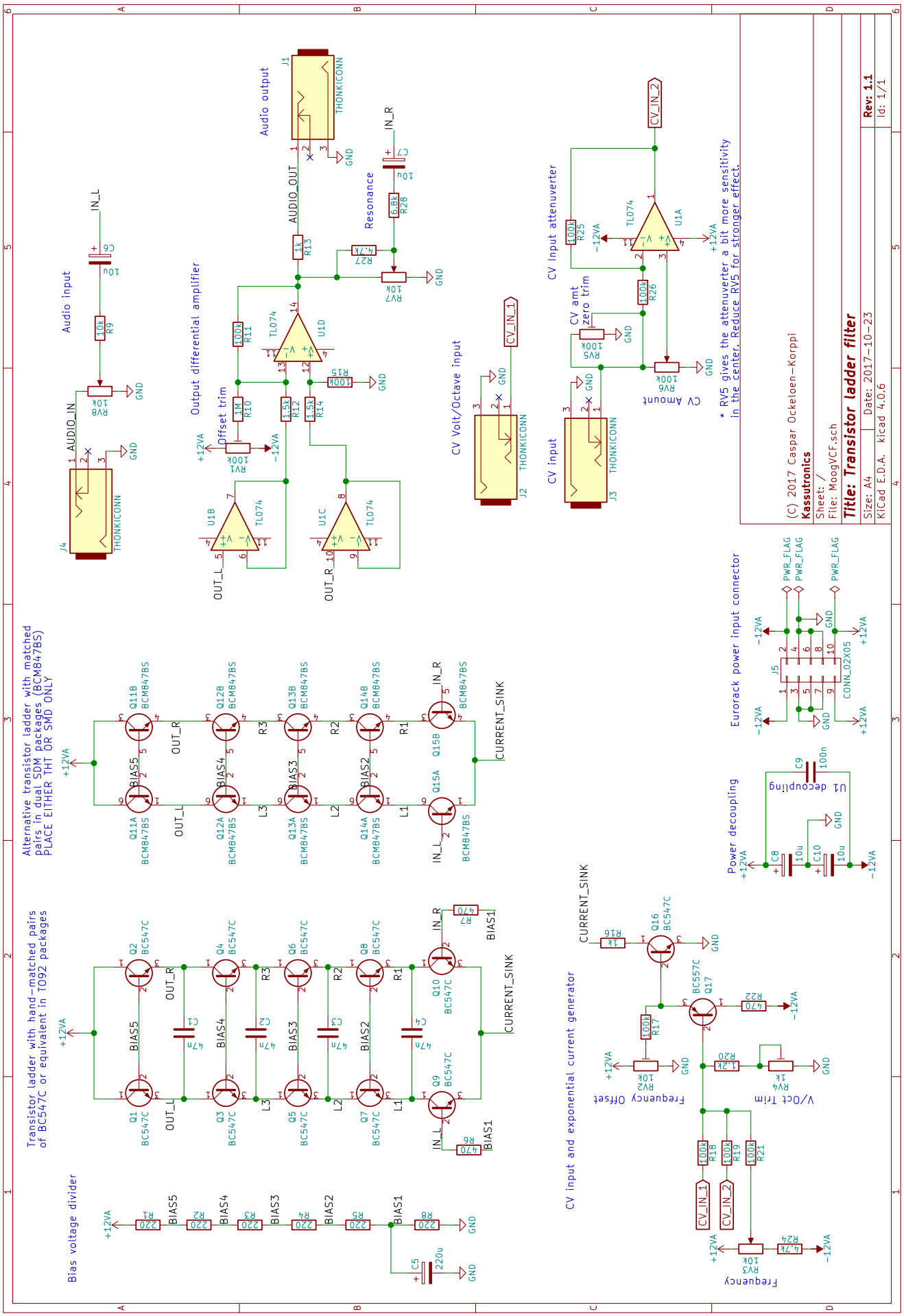
Board view



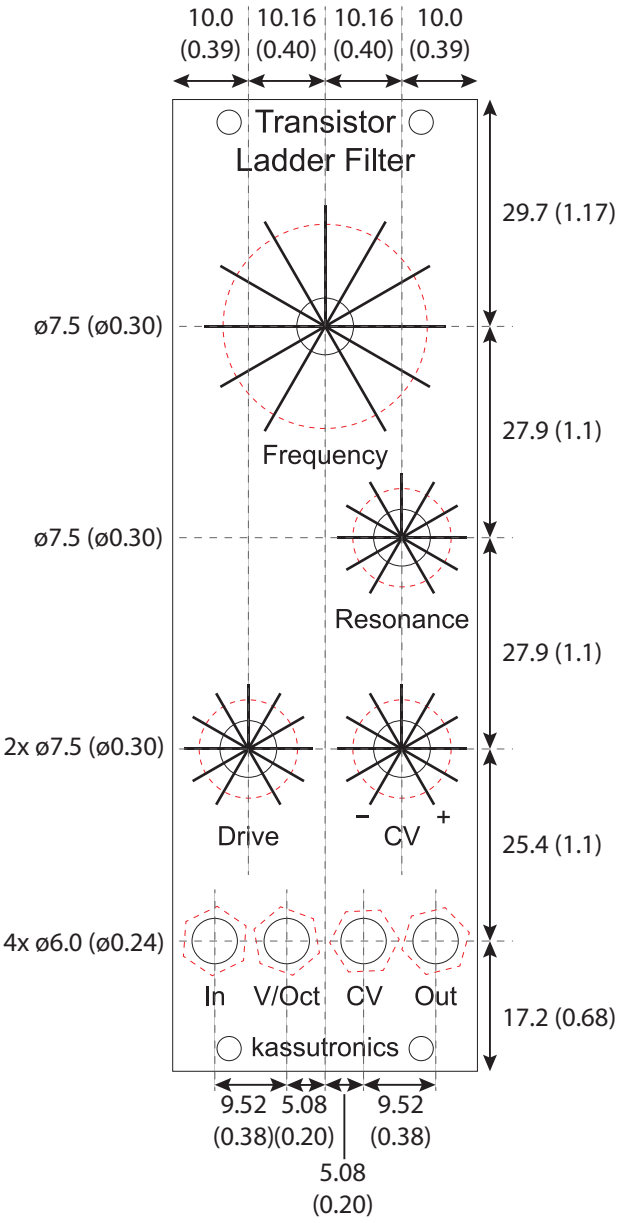
Rev 1.0 PCB
(shown with SMD transistors)



Rev 1.1 PCB
(shown with THT transistors)


Audio input
Audio output
Output differential amplifier
Offset trim
Resonance
CV Input attenuverter
CV amt zero trim
CV Amount
CV input
CV Volt/Octave input
* RV5 gives the attenuverter a bit more sensitivity in the center. Reduce RV5 for stronger effect.
(C) 2017 Caspar Ockeloen-Korppi
Kassutronics
Sheet: /
File: MoogVCF.sch
Title: Transistor ladder filter
Size: A4
Date: 2017-10-23
KiCad E.D.A. kicad 4.0.6
Rev: 1.1
Id: 1/1
Alternative transistor ladder with matched pairs in dual SDM packages (BCM847BS)
PLACE EITHER THT OR SMD ONLY
Transistor ladder with hand-matched pairs of BC547C or equivalent in TO92 packages
Bias voltage divider
CV input and exponential current generator
Eurorack power input connector
Power decoupling
U1 decoupling
Frequency Offset
V/Oct Trim
Frequency

Front panel dimensions



Dimensions in mm (inch)

Troubleshooting

In case the module does not function properly, the following measurements may help to narrow down the issue.

- Measure the ladder bias voltages labeled BIAS1 through BIAS5. These should be 2, 4, 6, 8, 10 V respectively. If some steps are smaller or zero, this may indicate a short in the transistor ladder near that step.
- Check if the exponential current generator is working: measure the voltage across R16. It should be around 25mV with the frequency knob fully counter-clockwise, and around 1.3V with the frequency knob fully clockwise. If the values are very different, check the circuitry around Q16 and Q17, as well as the orientation and type of the transistors. Note that Q17 is a PNP transistor (BC557C).

  The following table lists expected voltages for different measurement points, and actions to try if the measured voltage doesn't match. Note that the exact voltages depend on component variation as well as the calibration, but once the module is working and fully calibrated the voltages should agree within 10% or so.

Measurement	CCW value	CW value	Action
Wiper of RV3	-4V	12 V	Check RV3, R24
Base of Q17	-70 mV	220 mV	Adjust RV4, check R21, R20, RV4
Base of Q16	470 mV	640 mV	Adjust RV2, check R22, R17, RV2

Revision history

Board revisions

1.0 Initial design.

1.1 Updated component values of R9, R28, C8, C10;
Re-arranged C8, C10 and power header.

Documentation revisions

A Initial documentation for board revision 1.0

B Re-formatted and added info for board revision 1.1; Small fixes to documentation.

C Added note regarding 2N3904 pinout.

D Added front panel dimensions.

E Added trouble shooting information; Small improvements to documentation.

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.