MSK 009 Coiler Multi-Mode Filter and Rectifier

North Coast Synthesis Ltd.
Matthew Skala

May 2, 2022

Contents

General notes

This manual documents the MSK 009 Coiler Multi-Mode Filter and Rectifier, which is a module for use in a Eurorack modular synthesizer. The module contains a voltage-controlled two-pole state variable filter implemented using inductors (coils, hence the name) as the main energy-storing components in the integrators. It also uses capacitors, which have their main effect at bass frequencies, with the filter's behaviour shading from capacitor-based to inductor-based between about 500Hz to 2kHz. There are separate outputs for high-pass, band-pass, and low-pass transfer functions, and two audio inputs, one of which goes through a full-wave rectifier before being fed into the filter.

Controls and connections

The front panel of the module is shown in Figure 1.

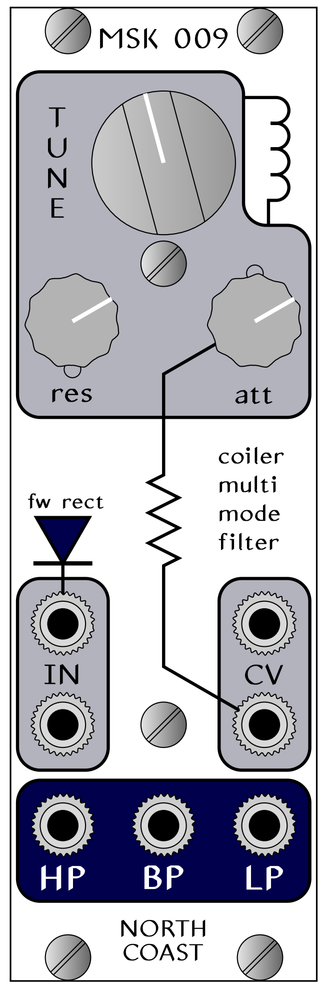


Figure 1: Module front panel.

TUNE knob This knob adjusts the overall frequency of the filter, for all three outputs. Its setting is added to the control voltage inputs. It should cover the entire usable range of the filter, with a little bit of excess at the low end to allow for "closing" the filter more completely when using voltage control in a low-pass gate patch.

res knob This sets the "resonance" of the filter by attenuating one of the feedback paths. Counter-clockwise for a flatter response curve, clockwise for a sharper peak. Near the clockwise maximum resonance, the filter will oscillate. Because of the way the inductors respond differently to phase at different points on the audio spectrum, this knob's effect interacts with the current frequency setting; the height of the resonance peak and the point at which oscillation begins will change with the cutoff frequency, creating a wide range of varying-timbre effects.

att knob This is an attenuator for the CV2 input—the lower of the two CV inputs, to which this knob is joined by the zigzag resistor line in the panel art, symbolizing attenuation. With the knob fully clockwise the sensitivity of this input is approximately

1V/octave, the same as the unattenuated input. At lower settings, the CV2 input is less sensitive.

IN inputs Audio inputs to the filter. The upper input, marked with a diode symbol and the notation **fw rect**, is subjected to full-wave rectification (positive and negative voltages translated into their absolute values) before being applied to the filter. The lower input is a direct connection. Both inputs may be used at once; their effects are summed.

The rectified input includes a phase inverter (both positive and negative voltages are translated into *negative*) to cancel out the naturally-occuring phase inversion between the input and LP output in this filter topology. As a result, if you feed an audio signal into the rectified input with the filter cutoff significantly below the frequency of the audio, it will be rectified and filtered into a *positive* voltage tracking the overall amplitude of the input signal. This way the module can be used as an envelope follower.

The inputs can accept any voltages between the module's power rails (−12V to +12V) without damage. The module my be overdriven, creating significant distortion, with inputs beyond about ±5V.

CV inputs Exponential control voltages for filter cutoff frequency. The upper socket (CV1) has a nominal sensitivity of 1V/octave; but the tracking of this filter is not meant to be very accurate, and it cannot be made highly accurate because of the somewhat unpredictable properties of the inductors. Tracking will differ in different parts of the audio spectrum. The CV-processing circuit is partially temperature-compensated, with zeroth-order "offset" compensation but not first-order "tracking" compensation. The lower socket's (CV2) sensitivity is adjustable with the att knob, to a maximum of the same sensitivity as CV1. The CV1 input, attenuated CV2 input, and TUNE knob setting are all summed to produce the control value for the filter core.

Both CV inputs can accept voltages anywhere between the module's power rails (−12V to +12V) without damage. Which voltages are useful depends on the patch and the setting of the TUNE knob, but a typical user might aim for 0V to 5V.

HP, BP, and LP outputs These are the three outputs of the filter core: high-pass, band-pass, and low-pass. Because this is a two-pole filter, the asymptotic slopes of the response curves are 12dB/octave for the high-pass and low-pass, and 6dB/octave on each of two slopes for the band-pass.

All three outputs are active simultaneously, driven by the combined input from the two IN jack sockets. The phase relationships among the three outputs will change with frequency as the filter shifts between using its capacitors and its inductors; that means mixing outputs to produce other filter functions (such as notch filtering) may produce results that sound good, but they are unlikely to be strong on measures like stopband attenuation.

Voltage levels on the audio outputs will normally be similar to the voltage levels on the inputs, with the maximum possible voltage limited by possible clipping in the op amp chips at around ±10V. Output level at maximum oscillation will be about ±5V. At the lowest resonance setting, the BP output will be a little quieter than the other two, an effect which tends to disappear at higher resonance.

Specifications

The nominal input impedance is 100kΩ for all inputs except the rectifier input, which varies between 50kΩ and 100kΩ. Nominal output impedance is 1kΩ for all outputs.

Any voltage between the power supply rails (nominally ±12V) is safe for the module, on any input; output voltages are limited by the capabilities of the op amps to about ±10V and will clip if the inputs are driven sufficiently hard. Distortion resulting from limiting in internal feedback paths may show up before the outputs actually clip.

The circuit is DC-coupled throughout; as a result, it can operate at very low frequencies, but small DC offsets may appear on the outputs. Trimmers are provided for minimizing offset effects.

Briefly shorting any input or output to any fixed voltage at or between the power rails, or shorting two to each other, should be harmless to the module. Patching the MSK 009's output into some other module's output should be harmless to the MSK 009, but doing that is not recommended because it is possible the non-MSK 009 module may be harmed.

This module (assuming a correct build using the recommended components) is protected against reverse power connection. It will not function with the power reversed, but will not cause or suffer any damage. Some other kinds of power misconnection may possibly be dangerous to the module or the power supply.

In normal operation the maximum current demand of this module is 25mA from the +12V supply and 25mA from the -12V supply. Placing an unusually heavy load on the outputs (for instance, with so-called passive modules) can increase the power supply current beyond those levels.

Voltage modification

This circuit is designed for ±12V power. It should work acceptably on ±15V power without modification, assuming all components are rated for the increased voltage, but some current levels and adjustment ranges are related to the power supplies and so just applying ±15V power with no changes may not give optimal results. In particular, I would expect doing that to create "dead zones" at the ends of the tuning control range. My suggestion if using ±15V power would be to increase all four 220kΩ resistors (R5, R8, R19, and R29) to 270kΩ; that should restore the intended current levels and adjustment ranges.

I have calculated but not tested these resistor changes.

Source package

A ZIP archive containing source code for this document and for the module itself, including things like machine-readable CAD files, is available from the Web site at `https://northcoastsynthesis.com/`. Be aware that actually building from source requires some manual steps; Makefiles for GNU Make are provided, but you may need to manually generate PDFs from the CAD files for inclusion in the document, make Gerbers from the PCB design, manually edit the .csv bill of materials files if you change the bill of materials, and so on.

Recommended software for use with the source code includes:

- GNU Make;
- LaTeX for document compilation;
- LaTeX.mk (Danjean and Legrand, not to be confused with other similarly-named LaTeX-automation tools);
- Circuit_macros (for in-document schematic diagrams);
- Kicad (electronic design automation);
- Qcad (2D drafting); and
- Perl (for the BOM-generating script).

The kicad-symbols/ subdirectory contains my customised schematic symbol and PCB footprint libraries for Kicad. Kicad doesn't normally keep dependencies like symbols inside a project directory, so on my system, these files actually live in a central directory shared by many projects. As a result, upon unpacking the ZIP file you may need to do some reconfiguration of the library paths stored inside the project files, in order to allow the symbols and footprints to be found. Also, this directory will probably contain some extra bonus symbols and footprints not actually used by this project, because it's a copy of the directory shared with other projects.

The package is covered by the GNU GPL, version 3, a copy of which is included in the file COPYING.

PCBs and physical design

The enclosed PCB design is for two boards. Board 1 is 3.90″×1.50″ or 99.06mm×38.10mm. Board 2 is a little shorter, 3.40″×1.50″ or 86.36mm×38.10mm. The two boards are intended to mount in a stack parallel to the Eurorack panel, held together with M3 machine screws and male-female hex standoff hardware. See Figure 2. Including 18mm of clearance for the mated power connector, the module should fit in 46mm of depth measured from the back of the front panel.

Use and contact information

This module design is released under the GNU GPL, version 3, a copy of which is in the source code package in the file named `COPYING`. One important consequence of the license is that if you distribute the design to others—for instance, as a built hardware device—then you are obligated to make the source code available to them at no additional charge, including any modifications you may have made to the original design. Source code for a hardware device includes without limitation such things as the machine-readable, human-editable CAD files for the circuit boards and panels. You also are not permitted to limit others' freedoms to redistribute the design and make further modifications of their own.

I sell this and other modules, both as fully assembled products and do-it-yourself kits, from my Web storefront at `http://northcoastsynthesis.com/`. Your support of my business is what makes it possible for me to continue releasing module designs for free. The latest version of this document and the associated source files can be found at that Web site.

Email should be sent to
`mskala@northcoastsynthesis.com`.

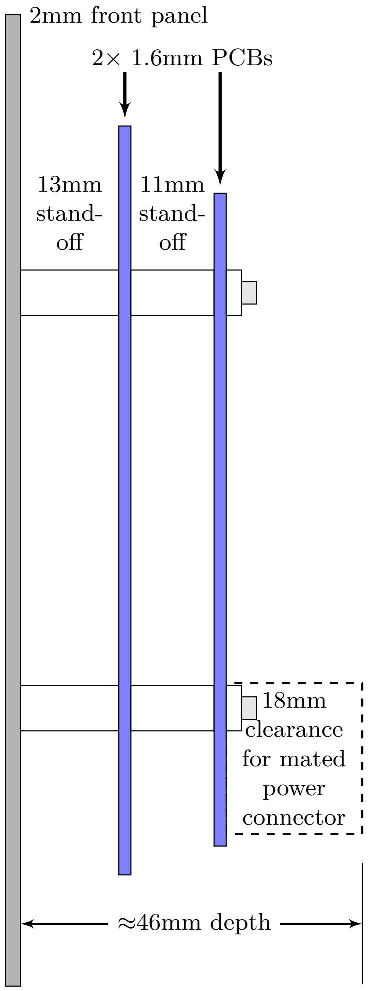


Figure 2: Assembled module, side view.

Safety and other warnings

Ask an adult to help you.

North Coast Synthesis Ltd. does not offer warranties or technical support on anything we did not build and sell. That applies both to modules built by you or others from the kits we sell, and to fully-assembled modules that might be built by others using our plans. Especially note that because we publish detailed plans and we permit third parties to build and sell modules using our plans subject to the relevant license terms, it is reasonable to expect that there will be modules on the new and used markets closely resembling ours but not built and sold by us. We may be able to help in authenticating a module of unknown provenance; contact us if you have questions of this nature.

For new modules purchased through a reseller, warranty and technical support issues should be taken to the reseller *first*. Resellers buy modules from North Coast at a significant discount, allowing them to resell the modules at a profit, and part of the way they earn that is by taking responsibility for supporting their own customers.

We also sell our products to hobbyists who enjoy tinkering with and customizing electronic equipment. Modules like ours, even if originally built by us, may be quite likely to contain third-party "mods," added or deleted features, or otherwise differ from the standard specifications of our assembled modules when new. Be aware of this possibility when you buy a used module.

Soldering irons are very hot.

Solder splashes and cut-off bits of component leads can fly a greater distance and are harder to clean up than you might expect. Spread out some newspapers or similar to catch them, and wear eye protection.

Lead solder is toxic, as are some fluxes used with lead-free solder. Do not eat, drink, smoke, pick your nose, or engage in sexual activity while using solder, and wash your hands when you are done using it.

Solder flux fumes are toxic, *especially* from lead-free solder because of its higher working temperature. Use appropriate ventilation.

Some lead-free solder alloys produce joints that look "cold" (i.e. defective) even when they are correctly made. This effect can be especially distressing to those of us who learned soldering with lead solder and then switched to lead-free. Learn the behaviour of whatever alloy you are using, and then trust your skills.

Water-soluble solder flux must be washed off promptly (within less than an hour of application) because if left in place it will corrode the metal. Solder with water-soluble flux should not be used with stranded wire because it is nearly impossible to remove from between the strands.

Residue from traditional rosin-based solder flux can result in undesired leakage currents that may affect high-impedance circuits. This module does not use any extremely high impedances, but small leakage currents could possibly reduce its accuracy. If your soldering leaves a lot of such residue then it might be advisable to clean that off.

Voltage and current levels in some synthesizer circuits may be dangerous.

Do not attempt to make solder flow through the board and form fillets on both sides of every joint. Some soldering tutorials claim that that is desirable or even mandatory, it does look nicer, and it may happen naturally when the conditions are good and the leads happen to be small in relation to the holes. But with large wire leads that just fit in the holes, when the holes are connected to the ground plane (even through thermal reliefs), on some harder-to-wet lead finishes, with lead-free solder, and so on, you may only end up dumping excessive heat into the joint and damaging the components while you fuss over perfect fillets. A well-made solder joint that just covers the pad and makes good contact to the lead on one side of the board, is good enough.

Building your own electronic equipment is seldom cheaper than buying equivalent commercial products, due to commercial economies of scale from which you as small-scale home builder cannot benefit. If you think getting into DIY construction is a way to save money, you will probably be disappointed.

Bill of materials

This table is not a substitute for the text instructions.

Qty	Ref	Value/Part No.	
1	C12	33pF	radial ceramic, 0.2″ lead spacing
1	C1	100pF	radial ceramic, 0.2″ lead spacing
2	C2, C3	6800pF	film, 0.2″ lead spacing
6	C6–C11	0.1μF	axial ceramic
2	C4, C5	10μF	radial aluminum electrolytic, 0.1″ lead spacing
1	D1	1N4148	or 1N914; switching diode
2	D4, D5	1N5230B	4.7V Zener
2	D2, D3	1N5818	or SB130; Schottky rectifier
2	H1, H2		nut for M3 machine screw
2	H3, H4	M3x11	M3 male-female standoff, 11mm body length
2	H7, H8	M3x13	M3 male-female standoff, 13mm body length
6	H15–H20	M3x6	M3 machine screw, 6mm body length
4	H11–H14		nylon washer for M3 machine screw
7	J1–J7	1502 03	switched mono 3.5mm panel jack, Lumberg
1	J8		female single-row socket, 10 pins at 0.1″
2	L1, L2	22mH	EPCOS 5% axial ferrite choke, B82144A2226J
1	P1		male single-row header, 10 pins at 0.1″
1	P2		male Eurorack power header, 2×5 pins at 0.1″
2	Q1, Q2	SS8550D	or PN200A; PNP high gain, TO-92 EBC
4	R22, R23, R32, R33	510Ω	
3	R15, R27, R37	1kΩ	
1	R6	1.8kΩ	
1	R7	2.7kΩ	
2	R24, R34	9.1kΩ	
2	R25, R35	10kΩ	
1	R10	18kΩ	
1	R38	22kΩ	
4	R21, R26, R31, R36	27kΩ	
1	R11	36kΩ	
2	R9, R17	51kΩ	
2	R20, R30	100kΩ	horizontal single turn, Vishay T73YP or similar
3	R2, R4, R16	100kΩ	vertical conductive plastic panel pot, BI Technologies P260T series, linear taper
7	R1, R3, R12–R14, R18, R28	100kΩ	

Qty	Ref	Value/Part No.	
4	R5, R8, R19, R29	220kΩ	
1	U3	LM13700	dual operational transconductance amp
2	U1, U2	TL074	quad JFET-input op amp

Fixed resistors should be 1% metal film throughout. RoHS-certified zinc-plated steel hardware is recommended, not stainless steel because of galvanic-corrosion incompatibility with aluminum parts.

Newer kits may include TL074B op amps instead of TL074, to make offset nulling easier.

Also needed: solder and related supplies, PCB, panel, knobs, a cable tie, Eurorack power cable, etc.

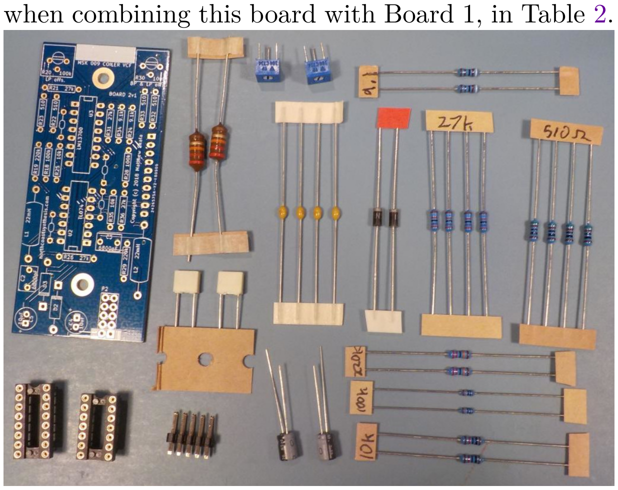
Building Board 2

The recommended order for building this module is to assemble Board 2, the one further from the front panel, first. That will make it easier to get all the physical positioning right for the components that bridge between the boards or pass through the panel.

Note that although I'm describing a separate step for each component value, and that's how I built my prototype so as to have plenty of photo opportunities, if you are reasonably confident about your skills you may find it easier to populate all or most of the board (i.e. put the components in place) and then solder them in a single step. Except where noted, the order in which you add components does not matter much.

Preliminaries

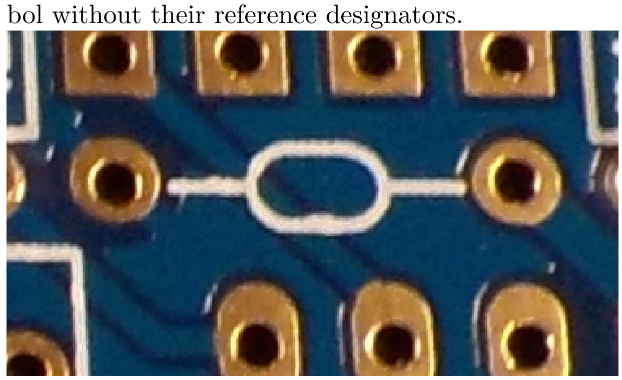
Count out the right number of everything according to the bill of materials. There is an abbreviated BOM for Board 2, excluding a few items that will be added when combining this board with Board 1, in Table 2.



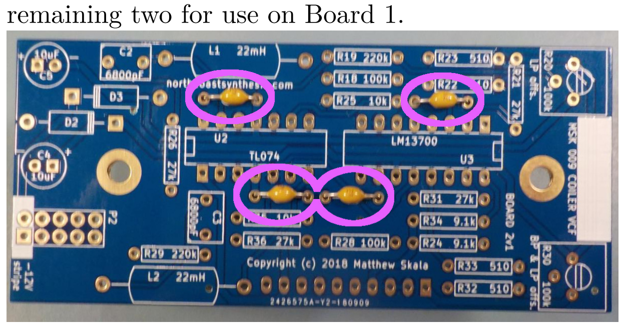
There are two trimmers to be installed on this board. Before installing them, use an ohmmeter to adjust each one to 50% of its range. Measure the resistance along the track, then measure the resistance from the wiper to one end and adjust to make the wiper half the total track resistance. This need not be exact, but having them start near their midpoints will help with adjustment later, by reducing issues with interaction among the different settings. With both trimmers pre-set to 50%, the module should basically work even if it is not at its best, whereas if they are installed at extreme values instead, then you may have trouble getting it up and running enough to adjust it more accurately.

Decoupling capacitors

The four axial ceramic 0.1μF decoupling capacitors, C8 to C11, are shown on the board by a special symbol without their reference designators.



Install these four capacitors where the symbol appears. They are not polarized and may be installed in either orientation. These capacitors act as filters for the power supplies to the op amp and OTA chips. An MSK 009 kit should include six of these capacitors, and only four are used on this board; save the remaining two for use on Board 1.



Fixed resistors

Resistors are never polarized. I like to install mine in a consistent direction for cosmetic reasons, but this is electrically unnecessary. In this module, the fixed

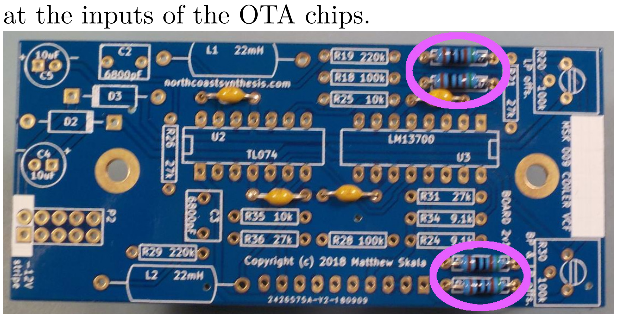
This table is not a substitute for the text instructions.

Qty	Ref	Value/Part No.	
2	C2, C3	6800pF	film, 0.2″ lead spacing
4	C8–C11	0.1μF	axial ceramic
2	C4, C5	10μF	radial aluminum electrolytic, 0.1″ lead spacing
2	D2, D3	1N5818	or SB130; Schottky rectifier
2	L1, L2	22mH	EPCOS 5% axial ferrite choke, B82144A2226J
1	P2		male Eurorack power header, 2 × 5 pins at 0.1″
4	R22, R23, R32, R33	510Ω	
2	R24, R34	9.1kΩ	
2	R25, R35	10kΩ	
4	R21, R26, R31, R36	27kΩ	
2	R20, R30	100kΩ	horizontal single turn, Vishay T73YP or similar
2	R18, R28	100kΩ	
2	R19, R29	220kΩ	
1	U2		14-pin DIP socket
1	U3		16-pin DIP socket

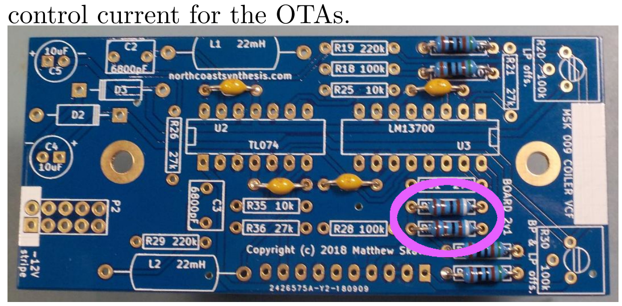
Table 2: Bill of Materials for assembling Board 2. Also needed is the PCB itself.

resistors are metal film 1% type. They usually have blue bodies and four colour bands designating the value, plus a fifth band for the tolerance. The tolerance band is brown for 1%, but note that we may occasionally ship better-tolerance resistors in the kits than the specifications require, if we are able to source them at a good price. Accordingly, I mention only the four value band colours for this type of resistor; if you are using resistors with other codes, you are responsible for knowing them. Note that colour codes on metal film 1% resistors are often ambiguous (reading from one end or the other end may give two different values, both plausible) and some of the colours are hard to distinguish anyway. If in doubt, always measure with an ohmmeter before soldering the resistor in place.

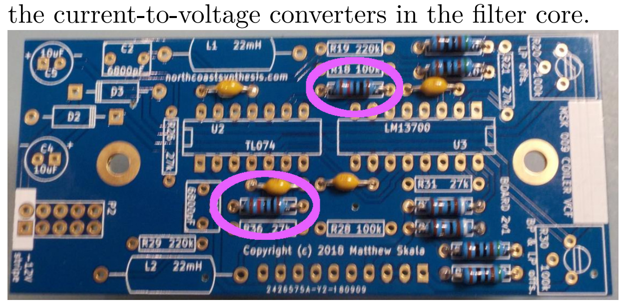
Install the four 510Ω (green-brown-black-black) resistors R22, R23, R32, and R33. These resistors, with the 100kΩ ones added later, set the signal levels at the inputs of the OTA chips.



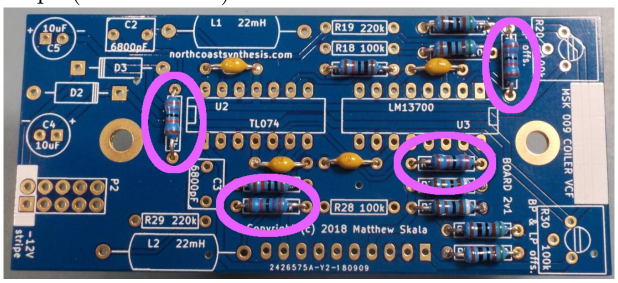
Install the two 9.1kΩ (white-brown-black-brown) resistors R24 and R34. These limit the maximum control current for the OTAs.



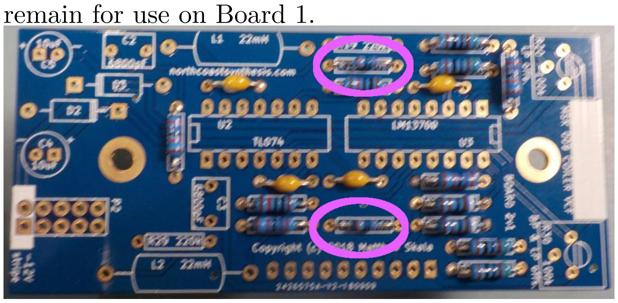
Install the two 10kΩ (brown-black-black-red) resistors R25 and R35. These are feedback resistors for the current-to-voltage converters in the filter core.



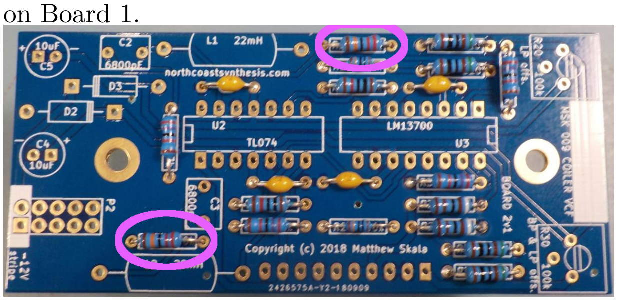
Install the four 27kΩ (red-violet-black-red) resistors R21, R26, R31, and R36. These are feedback resistors for the integrators (R26 and R36), and set the current for the linearizing diodes in the LM13700 chips (R1 and R31).



Install the two 100kΩ (brown-black-black-orange) resistors R18 and R28. These resistors participate in setting the input levels for the OTA chips. A full kit contains seven resistors of this value; five should remain for use on Board 1.

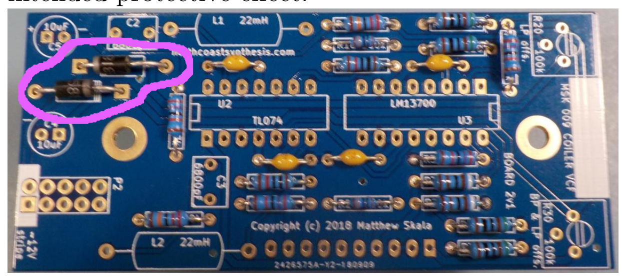


Install the two 220kΩ (red-red-black-orange) resistors R19 and R29. These resistors set the adjustment ranges for the DC offset trimmers. A full kit contains four resistors of this value; save two for use on Board 1.



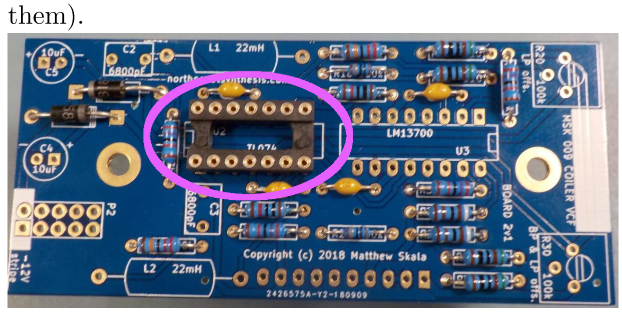
Semiconductors

Install the two 1N5818 or SBA130 Schottky rectifier diodes D2 and D3. These are for reverse-voltage protection; they cut off power to the module when the power plug is backwards. They are polarized and it is important to install them in the right direction. Each diode is packaged inside a black or dark grey plastic slug with a white or light grey stripe at one end; that end is the *cathode*. The silkscreen markings on the board have a corresponding stripe and the diodes should be installed with their stripes matching the markings on the board. The solder pads for the cathodes are also square instead of round. Installing these backwards means they will have the opposite of the intended protective effect.



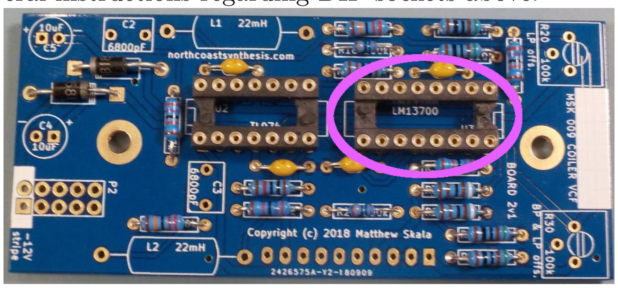
Install the 14-pin DIP socket for the operational amplifier chip U2. This chip does most of the amplification in the filter core. DIP sockets themselves do not care which direction you install them, but it is critically important that the chips installed in the sockets should be installed in the right direction. To help with that, the sockets will probably be marked with notches at one end (indicating the end where Pin 1 and Pin 14 are located) and you should install the sockets so that the notched ends match the notches shown on the PCB silkscreen. The solder pad for Pin 1 is also distinguished by being rectangular instead of rounded.

Installing DIP sockets without having them tilted at a funny angle can be tricky. I recommend inserting the socket in the board, taping it in place on the component side with vinyl electrical tape or sticking it there with a small blob of putty at each end, then soldering one pin on one corner and checking that the socket is snug against the board before soldering the other pins. That way, if you accidentally solder the first pin with the socket tilted, it will be easier to correct (only one pin to desolder instead of all of them).



If you somehow manage to solder an entire socket in backwards, don't try to desolder it to turn it around. Just leave it as it is and remember that when you insert the chip, you must insert it so the chip matches the markings on the *board*, not the turned-around socket.

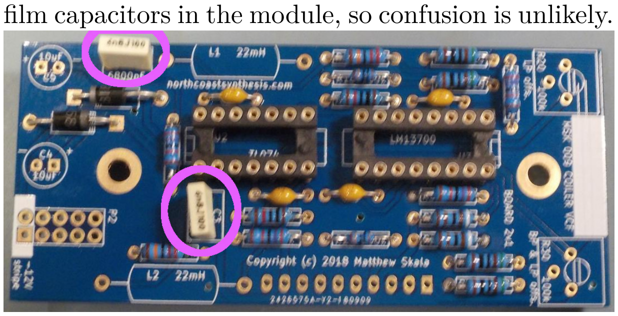
Install the 16-pin DIP socket for the OTA (operational transconductance amplifier) chip U3. This chip contains two current-controlled amplifiers, which, by means of a frequency-dependent control current, tune the filter core to the desired frequency. See the general instructions regarding DIP sockets above.



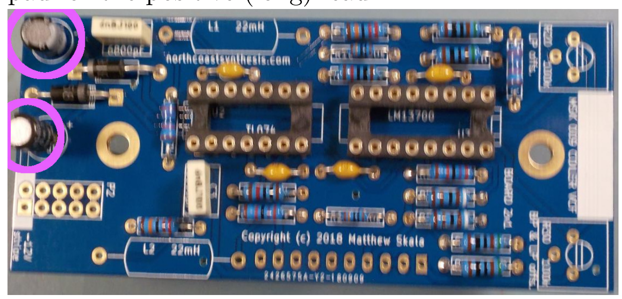
Electrolytic and film capacitors

Install the two 6800pF film capacitors C2 and C3. These are timing components used in the integrators at low frequencies to complement the inductors used at medium to high frequencies. They are unpolarized components and may be installed in either orientation.

The markings on film capacitors may vary depending on the manufacturer and model. These ones might be marked "682" (for 68 followed by two 0s number of picofarads), "6n8" (for 6.8nF), or even "0.0068" (value in μF). However, these are the only film capacitors in the module, so confusion is unlikely.



Install the two 10μF electrolytic capacitors C4 and C5, which filter the power supply for the module as a whole. These are polarized components and they may explode if installed backwards. Each one will be marked on its casing with a stripe and minus signs to indicate the negative lead; the positive lead will probably also be longer. These clues should be matched with the markings on the PCB: plus and minus symbols in the silkscreen and a square solder pad for the positive (long) lead.



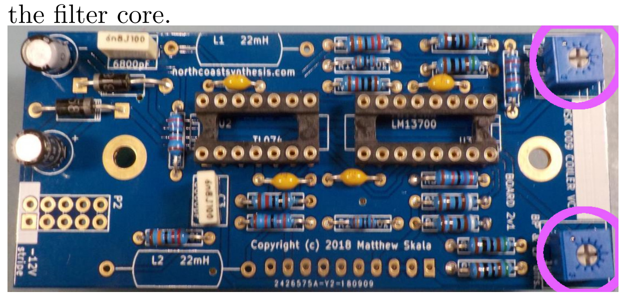
Trimmer potentiometers

If you have not already set the trimmers to 50% of their full scale value as described under "Preliminaries" above, then do it now.

Trimmers usually are not washable, so if you plan to clean your boards by full immersion in water or other solvent, your last chance is now; future cleaning will have to be done with a brush and some care to avoid letting liquid seep into the trimmers. Even now you should take some care with the DIP sockets, because solvent can carry flux residue into them and form a varnish-like layer if not carefully rinsed away.

Trimmers are not exactly polarized, but the three legs of each trimmer serve different functions and need to be connected to the right holes. The physical arrangement of the legs and corresponding holes should make it impossible to install the trimmers wrong way round.

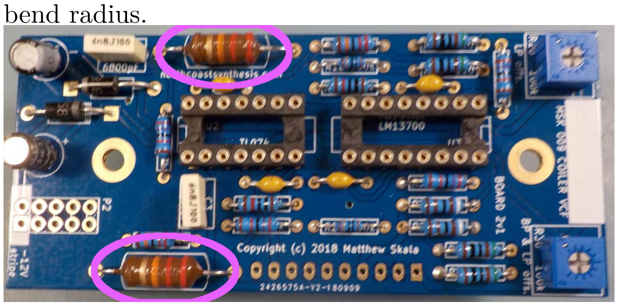
Install the two 100kΩ trimmers R20 and R30. These trimmers are for compensating DC offsets in the filter core.



Inductors

The two 22mH ferrite-bobbin inductors, that is, *coils*, L1 and L2 give this module its name. Install them now. They are the main timing components in the filter core, serving at medium to high frequencies. Single inductors like these have no polarity and may be installed in either direction; the situation is more complicated with transformers made of two or more interacting inductors.

The inductors are delicate, especially in the area where the leads attach to the bodies, because the windings that connect to the leads are made of very fine wire. The ferrite core material is also somewhat brittile. It is important not to bend the leads too close to the bodies. There is some extra space for the inductors on the circuit board to allow for a gentle bend radius.



Eurorack power connector

Install the 2×5-pin Eurorack power connector J2. This connector is not polarized in itself, although the connection it makes is polarized. As with the DIP sockets, you should be careful to get it installed snugly against the board, not tilted at an angle. Use tape or putty to hold it in place, solder one pin, then check that it is straight before you solder the other pins.

The six pins in the centre of the connector, that is all except the four corner pins, are for grounding and they are all connected together on the board. Thus, if you accidentally form solder bridges among these six pins while installing the connector, don't waste effort trying to remove them; they will have no electrical effect.

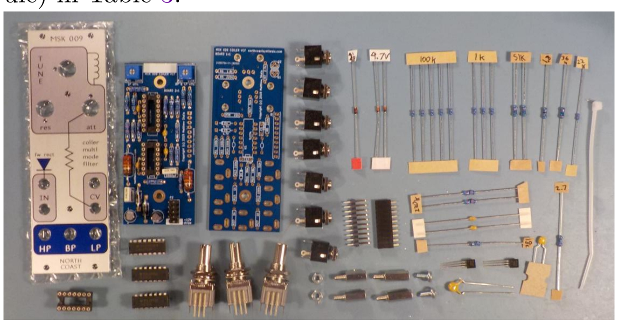
In between completed boards is a good time to take a break.

Building Board 1

Board 1 has components on both sides, and for best results, it is important to install them in the right order. Build Board 2 first, and see the general comments in the Board 2 chapter about how to approach the task.

Preliminaries

Count out the right number of everything according to the bill of materials. There is an abbreviated BOM for the items needed in this chapter (including the connection to Board 2 and final assembly of the module) in Table 3.



Some notes on knobs

The first batch of knobs I ordered for North Coast products turned out to have serious quality problems, specifically with the setscrews that hold the knobs onto the potentiometer shafts. Some of the screws had marginal threads that would strip when the screw was tightened, and I ended up having to do a bunch of extra testing and ship extra knobs to some customers to replace any that might fail. Later batches have also had issues, although they're under better control now because the bad first batch served as a warning to step up the testing procedures. Starting with kits prepared in August 2019, I switched to blue knobs with 100% testing; in September 2020, I switched to a new manufacturer, and knobs that are a slightly darker shade of blue. Although all the knobs I ship in kits now have been tested and passed at least twice, and should be fine to use, I am also shipping spare setscrews in any kits with knobs from batches where a signficant number of knobs failed testing.

Here are some things to be aware of as a kit builder.

- Some photos in these instructions were taken with the older grey knobs, and some dealers will still have kits containing grey knobs in their stock, but newer kits will have blue knobs.
- Do not overtighten the setscrews when attaching the knobs! The screw should be tight enough to hold the knob onto the shaft, but there's no advantage to making it tighter than that, and overtightening may risk destroying the screw thread or damaging the drive slot.
- If, despite my efforts to make sure no bad screws get sent to customers, you still get a bad screw that cannot be tightened and no spare for it, then please contact me.
- If you want to source an exact replacement for the setscrew, it should be an M3×3mm flat-tip slotted setscrew, which is also sometimes called a "grub screw," made of RoHS-compliant brass (possibly by exemption). Stainless steel is fine too, and I may sometimes ship stainless steel screws instead of brass if I can find a reliable source for them; plain steel should not be used here for galvanic corrosion reasons. Hex-socket screws are fine if you have the driver for them, but I don't ship those because I'm not sure all DIY builders do have the right driver.
- Because it's a standard M3 thread, in a pinch it's possible to substitute a plain M3 machine screw such as are commonly used with Eurorack cases, although one of those would obviously look less nice.

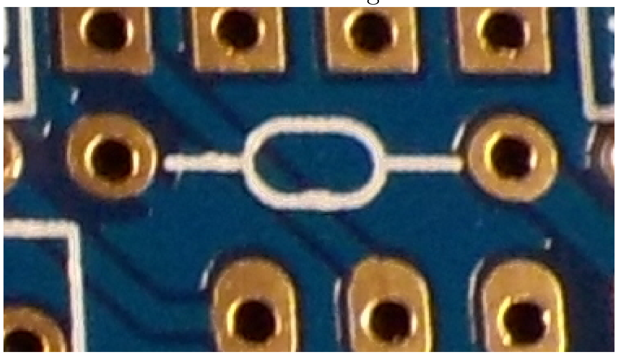
This table is not a substitute for the text instructions.

Qty	Ref	Value/Part No.	
1	C12	33pF	radial ceramic, 0.2″ lead spacing
1	C1	100pF	radial ceramic, 0.2″ lead spacing
2	C6, C7	0.1μF	axial ceramic
1	D1	1N4148	or 1N914; switching diode
2	D4, D5	1N5230B	4.7V Zener
2	H1, H2		nut for M3 machine screw
2	H3, H4	M3x11	M3 male-female standoff, 11mm body length
2	H7, H8	M3x13	M3 male-female standoff, 13mm body length
2	H15, H16	M3x6	M3 machine screw, 6mm body length
7	J1–J7	1502 03	switched mono 3.5mm panel jack, Lumberg
1	J8		female single-row socket, 10 pins at 0.1″
1	P1		male single-row header, 10 pins at 0.1″
2	Q1, Q2	SS8550D	or PN200A; PNP high gain, TO-92 EBC
3	R15, R27, R37	1kΩ	
1	R6	1.8kΩ	
1	R7	2.7kΩ	
1	R10	18kΩ	
1	R38	22kΩ	
1	R11	36kΩ	
2	R9, R17	51kΩ	
3	R2, R4, R16	100kΩ	vertical conductive plastic panel pot, BI Technologies P260T series, linear taper
5	R1, R3, R12–R14	100kΩ	
2	R5, R8	220kΩ	
1	U3	LM13700	dual operational transconductance amp
2	U1, U2	TL074	quad JFET-input op amp
1	U1		14-pin DIP socket

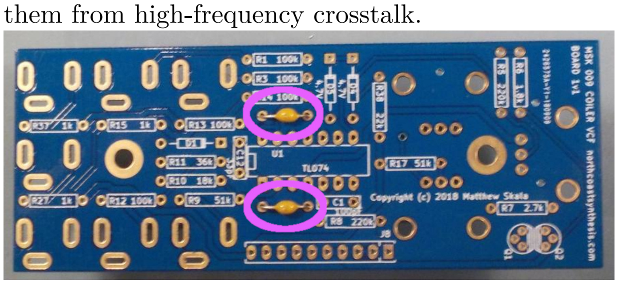
Table 3: Bill of Materials for Board 1. Newer kits may include TL074B op amps instead of TL074, to make offset nulling easier. Also needed: the PCB itself, the aluminum front panel, knobs, a cable tie, the assembled Board 2, and panel-to-rack mounting hardware.

Decoupling capacitors

The two axial ceramic 0.1μF decoupling capacitors C6 and C7 are shown on the board by a special symbol without their reference designators.



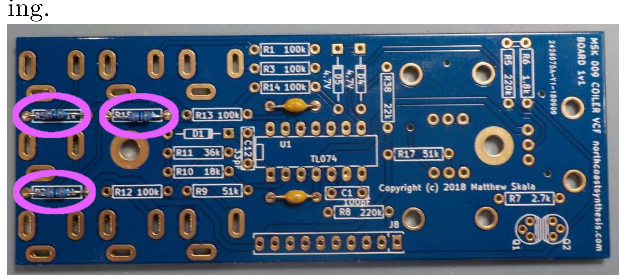
Install these two capacitors where the symbol appears. They are not polarized and may be installed in either orientation. These capacitors act as filters for the power supplies to the op amp chip, protecting them from high-frequency crosstalk.



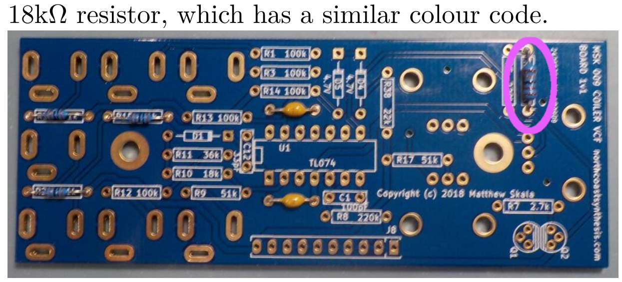
Fixed resistors

Resistors are never polarized. I like to install mine in a consistent direction for cosmetic reasons, but this is electrically unnecessary. In this module, the fixed resistors are metal film 1% type. They usually have blue bodies and four colour bands designating the value, plus a fifth band for the tolerance. The tolerance band is brown for 1%, but note that we may occasionally ship better-tolerance resistors in the kits than the specifications require, if we are able to source them at a good price. Accordingly, I mention only the four value band colours for this type of resistor; if you are using resistors with other codes, you are responsible for knowing them. Note that colour codes on metal film 1% resistors are often ambiguous (reading from one end or the other end may give two different values, both plausible) and some of the colours are hard to distinguish anyway. If in doubt, always measure with an ohmmeter before soldering the resistor in place.

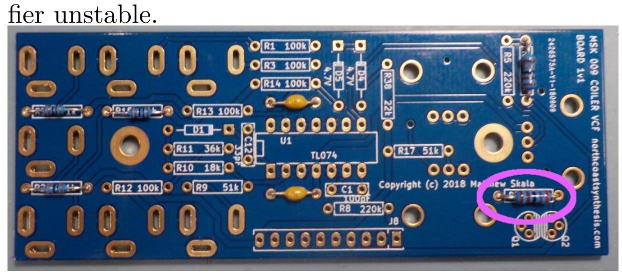
Install the three 1kΩ (brown-black-black-brown) resistors R15, R27, and R37. These are current-limiting resistors to protect the audio outputs, and other modules, in case of short circuits or bad patching.



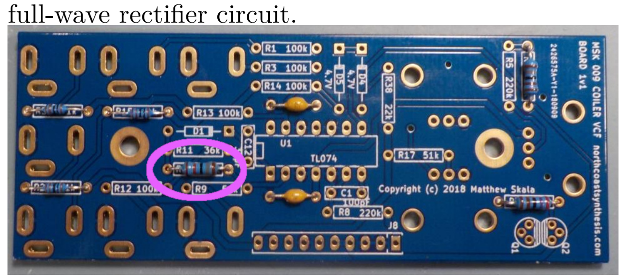
Install the 1.8kΩ (brown-grey-black-brown) resistor R6. This is the feedback resistor for the CV-processing op amp, setting the CV sensitivity to approximately 1V/octave. Do not confuse this with the 18kΩ resistor, which has a similar colour code.



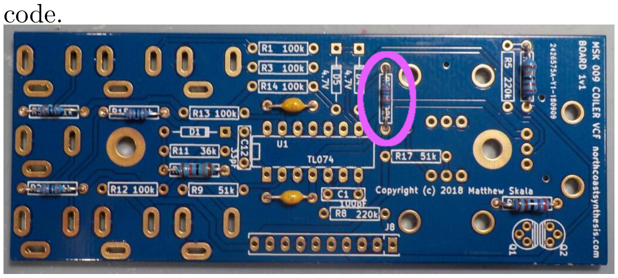
Install the 2.7kΩ (red-violet-black-brown) resistor R7. This is a ballast resistor for the temperature-servo op amp, preventing the voltage gain of the transistor in the feedback loop from rendering the amplifier unstable.



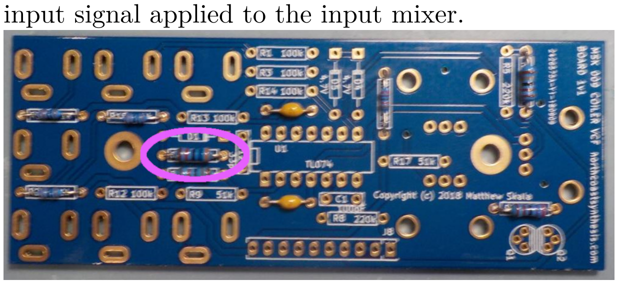
Install the 18kΩ (brown-grey-black-red) resistor R10. This resistor is used to control the gain in the full-wave rectifier circuit.



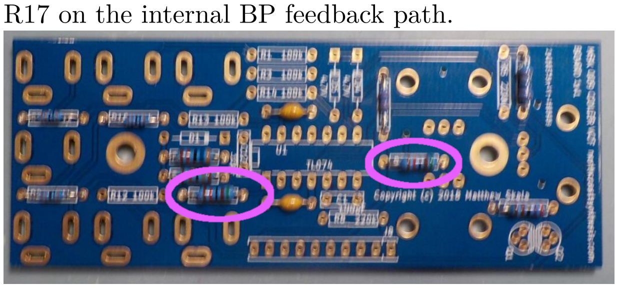
Install the 22kΩ (red-red-black-red) resistor R38. This resistor controls the amount of clipping applied on an internal feedback path, to set the amplitude level during oscillation. Do not confuse this resistor with the 220kΩ resistors, which have a similar colour code.



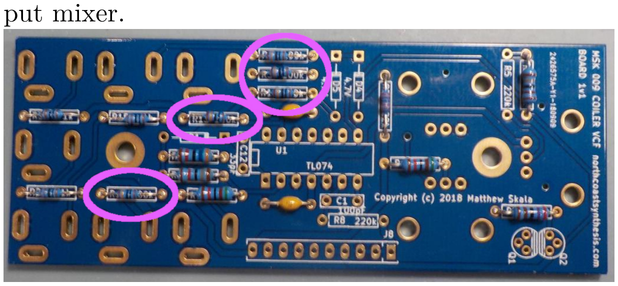
Install the 36kΩ (orange-blue-black-red) resistor R11. This controls the level of the full-wave rectified input signal applied to the input mixer.



Install the two 51kΩ (green-brown-black-red) resistors R9 and R17. These are used for level and impedance control: R9 on the rectifier input, and R17 on the internal BP feedback path.



Install the five 100kΩ (brown-black-black-orange) resistors R1, R3, R12, R13, and R14. The first three of these are used to set input impedances for the unrectified audio input and both CV inputs. The remaining two, R13 and R14, set gain levels in the input mixer.



Install the two 220kΩ (red-red-black-orange) resistors R5 and R8. The first, R5, controls the scale of the main tuning knob; the second, R8, contols the reference current for the exponential converter.

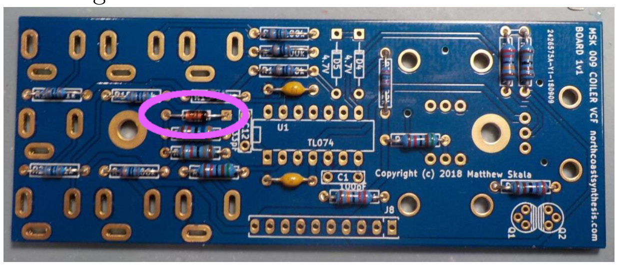
Semiconductors

There are two different kinds of diodes to install on this board and they look almost exactly alike: one 1N4148 or 1N914 switching diode named D1, and two 1N5230B or equivalent 4.7V Zener diodes named D4 and D5. All three diodes will be packaged in little pink glass beads with near-microscopic etched numbers indicating their type. Be careful not to confuse them; swapping the switching diode with a Zener will result in incorrect behaviour of the full-wave rectifier at high input voltages, and incorrect feedback levels probably causing either very strong oscillation at all resonance settings, or preventing oscillation entirely.

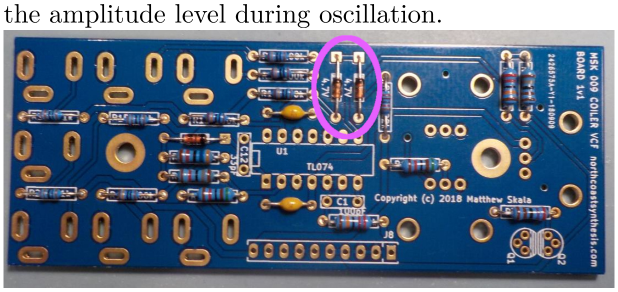
If you are unsure which diode is which and you cannot confidently read the etched markings, hook up a diode in series with a 10kΩ resistor reverse-biased across a 12V power supply and measure the voltage drop across the diode. If it is near 12V, then you are testing the switching diode; if it is near 4.7V, you are testing one of the Zener diodes; if it is near 0.6V, you probably have the diode connected forward-biased and should reverse the polarity of it or the power supply.

Both kinds of diodes are polarized and must be installed in the correct direction to function properly. One end of the glass body of the diode package will be labelled with a black band or stripe; that end is the *cathode*. The direction for the cathode is marked on the PCB silkscreen by a matching stripe in the printed symbol; and the solder pad for the cathode is square rather than round. The note “4.7V” is also on the PCB silkscreen next to the locations of the two Zener diodes as an added reminder of which diode goes where.

Install the switching diode D1, bearing in mind the notes above. This diode is used in the full-wave rectifier to switch between inverting and non-inverting modes.



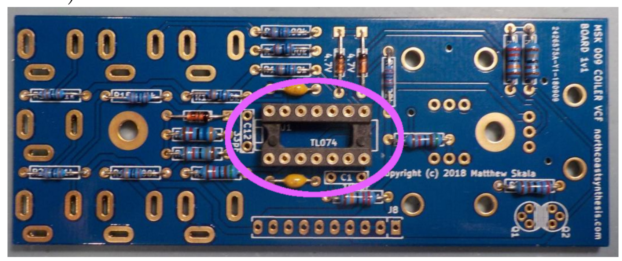
Install the two 4.7V Zener diodes D4 and D5. These apply clipping to the BP feedback path, setting the amplitude level during oscillation.



Install the 14-pin DIP socket for the TL074 quad operational amplifier (op amp) U1. Two of the four amplifiers on this chip form the exponential converter, which converts the tuning inputs into a control current for the core. One more is the full-wave rectifier for the rectified input, and the last amplifier is part of the filter core, mixing the input signals with the feedback paths to generate the HP signal.

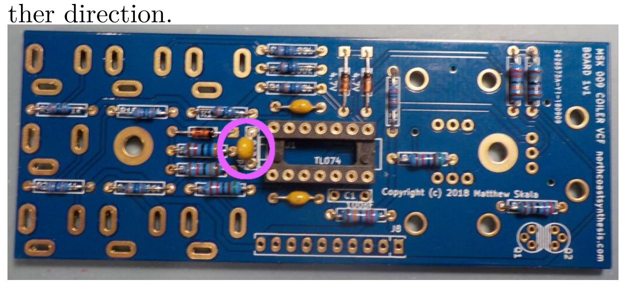
DIP sockets themselves do not care which direction you install them, but it is critically important that the chip installed in the socket should be installed in the right direction. To help with that, the socket will probably be marked with notches at one end (indicating the end where Pin 1 and Pin 14 are located) and you should install the socket so that the notched end matches the notch shown on the PCB silkscreen. The solder pad for Pin 1 is also distinguished by being rectangular instead of rounded.

Installing DIP sockets without having them tilted at a funny angle can be tricky. I recommend inserting the socket in the board, taping it in place on the component side with vinyl electrical tape or sticking it there with a small blob of putty at each end, then soldering one pin on one corner and checking that the socket is snug against the board before soldering the other pins. That way, if you accidentally solder the first pin with the socket tilted, it will be easier to correct (only one pin to desolder instead of all of them).

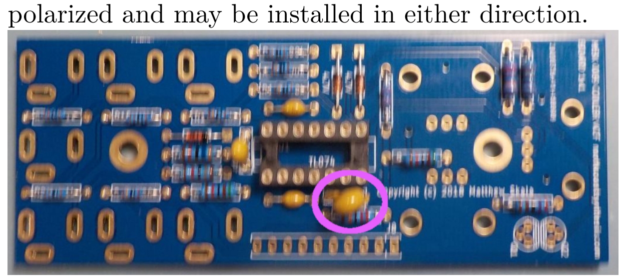


Compensation capacitors

Install the 33pF radial ceramic capacitor C12. It will probably be marked “330,” which means 33pF in a scheme similar to the resistor colour code: significant digits 3 3 to be followed by 0 zeroes. This capacitor helps ensure stability of the input mixer op amp, and the filter core as a whole, by killing the frequency response at frequencies above audio. Without it, the nonideal behaviour of the inductors could turn the filter core into a low-frequency radio transmitter. This capacitor is not polarized and may be installed in either direction.



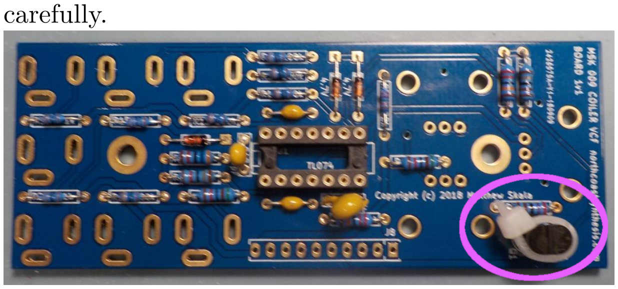
Install the 100pF radial ceramic capacitor C1. This similarly is a stability enhancer for the servo op amp in the exponential converter, preventing the amplifier from oscillating at high frequency. The capacitor will probably be marked “101,” for 1 0 followed by one more 0 number of picofarads. It is not polarized and may be installed in either direction.



Exponential converter cluster

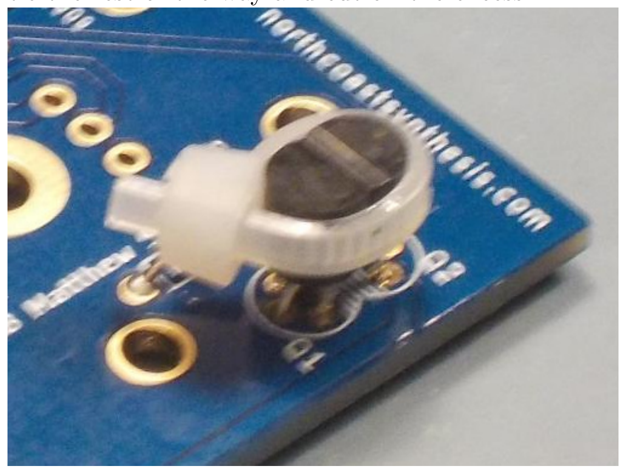
Because of the somewhat unpredictable behaviour of the coils, it is not realistic to expect really accurate voltage-to-frequency tracking of the Coiler VCF. The frequency CV inputs at maximum sensitivity are called 1V/octave to give some idea of the scale of voltages users might like to use, not as an accurate frequency standard. However, the circuit does contain partial temperature compensation to reduce unintended frequency changes as the module warms up.

For the temperature compensation to work as intended, the two transistors Q1 and Q2 should be kept at the same temperature. This is accomplished by mounting them face to face and tightening a nylon cable tie around them to keep them pressed together. If you have built a North Coast Synthesis Leapfrog Filter you will recognize the similar design; the Coiler, however, omits the thermistor used in the Leapfrog. Constructing this cluster of components is a little tricky and annoying; follow the directions carefully.



Insert the two SS8550D or PN200A transistors Q1 and Q2 in the board. Do not solder them yet. Put a nylon cable tie around the transistors and tighten it far enough to hold them loosely together. Concentrate in particular on having the two transistors meet as cleanly as possible, with the two fully flat sides in contact. Do not overtighten the cable tie or cut it off yet. This is probably the hardest step, and North Coast kits include a spare cable tie for use in case you ruin one. Be aware that the components should not stick out any further above the board than is normal for other TO-92 components; if you seat the transistors too high up, at the full length of their legs, you may exceed the 11mm of clearance between this board and Board 2 above it.

Solder the components. Then tighten the cable tie the rest of the way and cut off the excess.

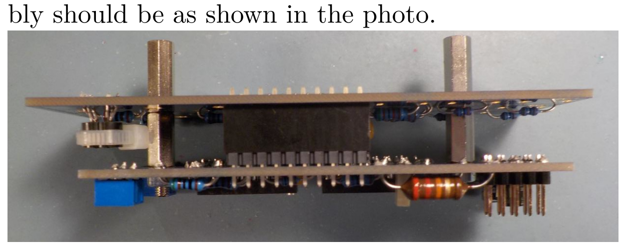


Board to board connectors

Fasten the two 13mm standoffs on the back of Board 1; that is the side opposite the components already installed. The male ends of the 13mm standoffs should pass through the mounting holes in the board and mate with the female ends of the 11mm standoffs on the front or component side of the board.

Mate the 10×1 header connectors J8 and P1 and place them (do not solder yet) in the J8 footprint on Board 1 with the legs of the female connector J8 going through the board.

Place your completed Board 2 from the previous chapter on top of the assembly, component side up with the legs of P1 going through the P1 footprint, and fasten the board to the 11mm standoffs with the two hex nuts. The resulting temporary assembly should be as shown in the photo.

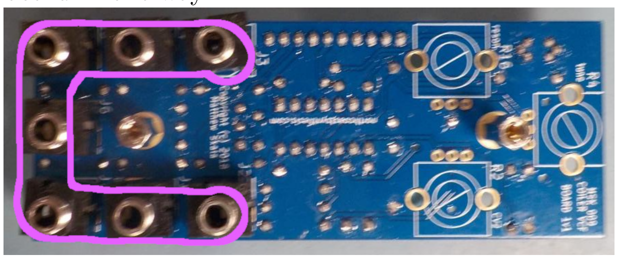


Solder J8 and P1 in place on the two boards. Then remove Board 2 and the hex nuts holding it in place, but keep the standoffs that go through the holes in Board 1.

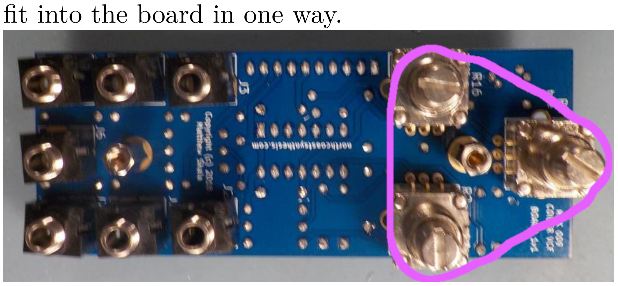
Panel components

Flip Board 1 over; you will now be installing the components that go between it and the panel. The pieces fit together in a straightforward way, but see the exploded assembly diagram on page 43 if further clarification is needed.

Place (do not solder yet) the seven phone jack sockets J1 through J7 in their footprints. These are for patching signals to and from other modules. These components should only be able to fit into the board in one way.



Place (do not solder yet) the three panel control potentiometers R2, R4, and R16 in their footprints. These are for patching signals to and from other modules. These components, too, should only be able to fit into the board in one way.



Line up the panel on top of the asembly and fasten it in place by driving the two machine screws through their corresponding holes into the 13mm standoffs.

Install all the hardware for the panel components. In the case of the potentiometers, the sequence is first (nearest the panel) the conical spring washer, high side in the middle and low side around the outside; then the toothed lock-washer; then the nut. If the teeth on the lock-washer seem sharper on one side, that side should be up, facing the nut.

In the case of the jack sockets, the knurled nuts provided for these will have screwdriver slots on one side, and those should face the outside with the smoother side facing the panel. You may need to tilt the assembly and jiggle it a bit to get the jack sockets to fall into the right alignment with their bushings poking through the panel. On the other side, when correctly installed their solder legs will just barely pass through the circuit board.

Do not overtighten any of this hardware, and be careful, if you are using wrenches or pliers, to avoid scratching the panel. Wrapping the tool jaws with tape may help.

Final assembly

Insert the TL074 chip in its socket on Board 1. Be careful to insert it right way round: the end with Pin 1 will be marked by an indentation at one corner or a notch in the end and this end of the chip should be inserted to match the notch in the socket and on the board silkscreen and the rectangular Pin 1 solder pad. The Pin 1 end of the chip is at the bottom when the module is inserted in a rack.

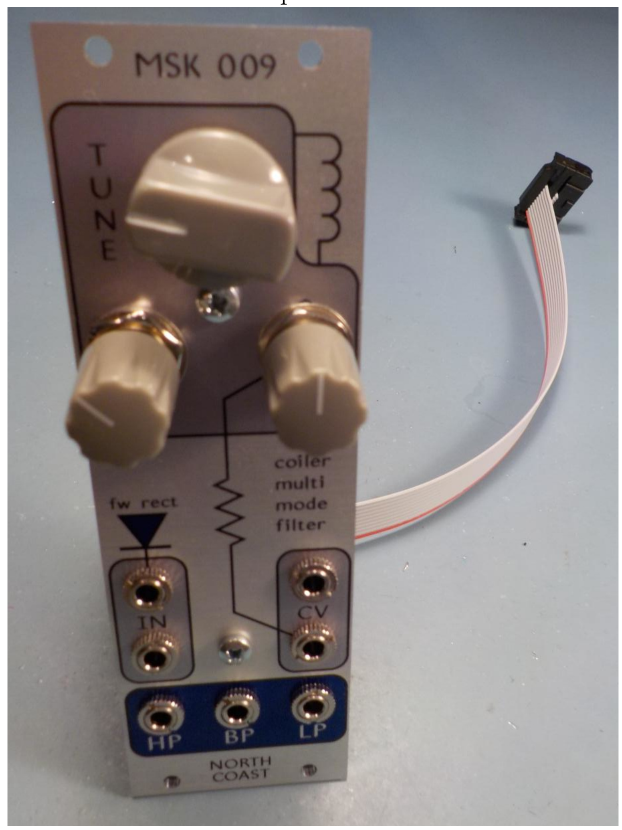
Also be careful all the legs of the chip go into the corresponding holes in the socket. These chips, when new, usually have their legs splayed outward a little bit to help them fit snugly into circuit boards when used without a socket and you must gently bend the legs inward to fit them in the sockets. If you apply pressure to a chip prematurely, without all the legs properly fitting into the holes, it is easy to have the legs fold up or even break off.

It should not be necessary to remove the panel from Board 1 again. Just attach Board 2, carefully fitting its header plug into the header socket on Board 1 and the male ends of the standoffs through the corresponding holes in Board 2. Then use the hex nuts to fasten Board 2 in place.

Insert the TL074 and LM13700 chips in their sockets on Board 2. Be careful to insert them right way round, with the Pin 1 markings on the chips matching those on the board. The two chips are oriented in opposite directions, with the Pin 1 marking on each chip at the end furthest from the other chip. As with the TL074 on Board 1, be careful all the legs are in the holes of the socket before you press each chip down, lest you fold up the delicate legs.

There is a rectangular white area at the top of Board 2 reserved for adding a serial number, signature, quality control marking, or similar. Use a fine-tipped permanent marker to write whatever you want there. Isopropyl alcohol will probably dissolve marker ink, so do this step after any board-cleaning.

Your module is complete.



Adjustment and testing

It is inevitable that a circuit of this kind will have a certain amount of DC offset and control feedthrough, and since the MSK 009 was never meant to be a "well-behaved" filter anyway, it doesn't seem appropriate to attempt to eliminate it entirely. Should you wish a completely DC-free signal, I recommend running the filter output through a North Coast Synthesis Ltd. MSK 011 Transistor Mixer with its AC-coupled output. However, the MSK 009 does have two trimmers for minimizing offset, and this chapter describes how to properly adjust them, along with some tips on troubleshooting should everything not work perfectly at initial power-up.

You will need a multimeter and a Eurorack power supply. An oscilloscope may be useful for troubleshooting but should not be needed for basic adjustment.

Short-circuit test

With no power applied to the module, check for short circuits between the three power connections on the Board 2 Eurorack power connector. The two pins at the bottom, marked with white on the circuit board, are for -12V. The two at the other end are for +12V; and the remaining six pins in the middle are all ground pins. Check between each pairing of these three voltages, in both directions (six tests in all). Ideally, you should use a multimeter's "diode test" range for this; if yours has no such range, use a low resistance-measuring setting. It should read infinite in the reverse direction (positive lead to −12V and negative lead to each of the other two, as well as positive lead to ground and negative to +12V) and greater than 1V or 1kΩ in the forward direction (reverse those three tests). If any of these six measurements is less than 1kΩ or 1V, then something is wrong with the build, most likely a blob of solder shorting between two connections, and you should troubleshoot that before applying power.

Optional: Although we test all cables before we sell them, bad cables have been known to exist, so it might be worth plugging the Eurorack power cable into the module and repeating these continuity tests across the cable's corresponding contacts (using bits of narrow-gauge wire to get into the contacts on the cable if necessary) to make sure there are no shorts in the cable crimping. Doing this *with the cable connected to the module* makes it easier to avoid mistakes, because the module itself will short together all wires that carry equal potential, making it easier to be sure of testing the relevant adjacent-wire pairs in the cable.

Plug the module into a Eurorack power supply and make sure neither it nor the power supply emits smoke, overheats, makes any unusual noises, or smells bad. If any of those things happen, turn off the power immediately, and troubleshoot the problem before proceeding.

Optional: Plug the module into a Eurorack power supply *backwards*, see that nothing bad happens, and congratulate yourself on having assembled the reverse-connection protective circuit properly. Reconnect it right way round before proceeding to the next step.

Output offset adjustment

Identify the two trimmers on the back of the module (Board 2). When you are looking at the back of the module, R20 is at upper left and R30 is at upper right. There is some interaction between them, but R20 primarily affects the offsets on the HP and LP outputs, and R30 the offsets on the BP and LP outputs. The labels on early versions of the circuit board do not mention R20's effect on HP. Bearing in mind that with two trimmers we cannot perfectly control all three offsets, and offset on LP is more important than on BP because of the intended application of the module as an envelope follower, the recommended procedure is to use R20 to minimize the offset on the HP output, then R30 to minimize the offset on the LP output, then repeat to deal with their interaction. The offset on the BP output is not directly controlled but should be small once the other two are minimized.

Disconnect any signal cables from the filter. Turn the resonance and CV attenuator controls fully counterclockwise and the tuning control not quite fully counterclockwise (knob pointer around eight o'clock). Apply power.

Measure the DC voltage on the HP output and adjust R20 to bring this voltage as near zero as you reasonably can. Within ±0.10V is good.

Measure the DC voltage on the LP output and adjust R30 to bring this voltage near zero also. Again, within ±0.10V is good.

Repeat the last two steps a second time: measure the HP voltage and adjust R20 to null it, and then measure the LP voltage and adjust R30 to null it. This completes the adjustment.

If you find that you cannot zero the voltages because the trimmers have insufficient adjustment range, try repeating the procedure with the tuning knob turned a little further counterclockwise. In general, the module will be easier to adjust the lower the tuning setting, but the behaviour of the module will be less predictable at high frequencies if adjusted at lower frequencies. On the other hand, if the knob is turned too far clockwise during the adjustment, then the module may tend to couple noise from the power supply into its outputs when set to very low frequencies. My current recommendation is to turn the knob to about eight o'clock as the best compromise.

If you cannot bring the voltages close to zero even with changes to the tuning, it may indicate a problem with the build, but one more thing to try is swapping the two TL074 chips. Occasionally, if the input offset on these chips is near the boundary of its acceptable specification, it can make adjustment difficult, and since the two chips will have different random input offsets, swapping them can resolve the issue.

Troubleshooting

It would require several books to convey all the skills and knowledge useful in troubleshooting even a simple electronic circuit like this one, but here are some possible symptoms and some suggestions on diagnosis and treatment.

No response from the module at all: no signal on the outputs even in resonance mode, or with strong signals on the inputs. Most likely a power problem, such as a power cable plugged in wrong or a short circuit. It might even be a problem in the power supply and not the module itself.

No oscillation with resonance turned up to maximum: it is normal that the module may not oscillate over the entire range of the tuning knob, and for the amplitude to vary a bit over the range where it does oscillate, but it should cover most if not all of the range. Be sure that the output offset adjustment, described above, has been done; excessive DC offset may inhibit oscillation. However, if you want absolutely the most oscillation possible, you may find that tweaking the trimmers a little bit away from zero-offset may increase the amplitude a little bit. This kind of experimentation is best done with an oscilloscope on the output as you adjust the trimmers. It's easy to gain an intuitive feeling for the oscillation/offset tradeoff when working live on the oscilloscope screen.

Module responds, but not as expected: first attempt to localize the problem. Do both inputs work? Do all three outputs work? If you can find two tests where one works and the other doesen't, then the problem is probably related to whatever part of the module was involved in the failing test and was not involved in both tests. Unfortunately, the multiple feedback loops in the state-variable filter mean that a large part of the module is involved in pretty much every function; but the general principle of narrowing things down remains at the heart of troubleshooting.

A specific problem noted on one prototype during testing: it's possible for contamination to be washed into the board-to-board connector during cleaning, and this can make the connection flaky, with possible hard-to-debug results including DC offsets that won't go to zero even with the trimmers at the ends of their ranges. If you've washed your boards with solvent, and especially if you haven't been careful to keep it out of the female connector, then try gently unplugging and re-plugging the board-to-bard connection a few times.

Shorts on board-to-board connector: in a couple of assembled builds I found a problem where pins of the board-to-board connector were shorting to the ground plane. I think the solder mask may have been slightly misaligned even though it passed quality control, allowing solder to short to an exposed bit of ground plane; or, possibly, too much soldering heat had burned through the mask to create a hole. Anyway, the main symptom was one or all of the output jacks giving DC voltages that didn't change with input CV and front-panel controls. To further diagnose it, check for continuity from each pin of the connctor to ground. Three of the pins are supposed to be connected to ground, but if more seem to be, it's probably a short. Doing the test with the boards separated will allow determining which board has the short. On my boards, once I knew the issue I was able to look closely and see the solder bridging from pin to ground plane nearby, and cutting the extra away with a sharp knife was enough to fix the problem.

Bad amplitude (including *no* amplitude) or DC offset in oscillation, combined with funny response to large-amplitude signals on the rectifier input: this combination of issues suggests that one of the Zener diodes was swapped with the nearly identical-looking switching diode. If one of these issues occurs without the other, it suggests that maybe all the correct diodes were installed, but at least one of them is backwards.

General quality issues: many problems can be diagnosed just by looking closely at your work, preferably with at least one night's sleep between when you assembled the module and when you examine it. Look for bad solder joints that fail to connect; solder bridges between nearby connections (especially on the discrete transistors in the exponential converter); components missing; components exchanged (especially resistors with similar colour codes, such as 1kΩ swapped with 10kΩ); polarized components such as diodes mounted backwards; and so on.

General tips for debugging DIP ICs: make sure for, for each IC, that

- it really is the type of IC it's supposed to be, not something else (beware of cheap ICs you buy from Chinese sellers on eBay, though the ones in this project are common enough in more reputable channels that you probably wouldn't have attempted that anyway);
- it is plugged in snugly;
- all the legs of the chip go nicely into the corresponding holes in the socket, with none bent outside or folded up under the chip;
- it is plugged in *at all* (forgetting to do so is a surprisingly common mistake!);
- it is plugged in the right way around, with the Pin 1 indentation or notch matching the clues on the board (if this is wrong, the chip is probably destroyed and will need to be replaced);
- there are no solder bridges on the chip socket, unsoldered pins, debris clogging the socket holes, or similar;
- its decoupling capacitors (the small ceramic ones) are installed and there is nothing wrong with their solder joints; and
- in the case of the two TL074 chips, try swapping them and see if that makes any difference.

Patch ideas

Here's a classic subtractive patch. The pitch CV from the MIDI interface drives both the VCO and the main CV input of the Coiler VCF. Since the tracking is not exactly 1V/octave, timbre will change depending on pitch. The gate CV goes to an MSK 012 envelope generator, which controls the output VCA. Using the low-pass output from the Coiler gives a classic analog subtractive sound, but either of the other outputs could be used to produce different effects; or the oscillator could be patched into the rectifier input of the Coiler instead of or in addition to the main audio input jack.

This low-pass gate patch gives an organic sound. The tuning knob is turned to its minimum and the envelope output just goes straight to frequency CV. Between notes, the envelope voltage is low enough to tune the filter cutoff below the audio, cutting off the signal.

Deluxe subtractive voice with stereo effect. There are two oscillators (in this case, Intellijel Dixie II modules), which could be tuned to near-unison or a musical interval. One is patched to the regular audio input of the Coiler, and one to the rectifier. There are separate ADSR envelopes for filter cutoff and amplitude envelope. The amplitude envelope is applied independently to the BP and LP outputs of the Coiler, using both channels of the VCA, and then those two signals go to an MSK 008 octave switch operating as a "mid-side decoder"; that is, it computes the sum and difference. The two channel outputs of the octave switch are the left and right channels of a stereo signal. This patch puts the low-pass signal more or less in the centre of the stereo field, with higher frequencies from the band-pass output and some phase effects resulting from the interaction of the two, spread out on the sides.

The Coiler can be used as an oscillator just by turning the resonance knob all the way up. It will not track 1V/octave, but for drones with manual control, and sequencing by ear instead of with a MIDI interface, that makes little difference. Sine waves appear on all three outputs, with the purest one on LP and the most distorted on HP. Adding a self-patch from the BP output to the frequency CV, as shown, creates some extra distortion in the waveform; with some adjustment of the CV attenuation knob, it's possible to get a reasonable sawtooth wave out of this patch.

Patch a sine wave LFO into the rectifier input, tune the filter well above the LFO frequency, and the result is a "bouncing ball" CV on the low-pass output.

With the tuning set low and audio patched into the rectifier, the Coiler's low-pass output functions as a simple envelope follower. In this patch, it's used as part of a compressor. The envelope follower goes through an attenuator/inverter to control an exponential VCA (should be exponential for good results). I have shown the Coiler rectifier and VCA audio inputs patched together, and for standard compression the audio would be applied that way, to both of them. For "sidechain," that is, one signal controlling the compression of another, just feed the main signal into the VCA and the controlling signal into the Coiler.

Circuit explanation

The MSK 009 Coiler VCF is a conventional two-pole state-variable design with a twist, or rather a couple of coils. The general theory of similar filters with capacitor-based integrators can be found in any textbook on active filters and I do not propose to go through it in great detail here; instead, I will give an intuitive description of the operating principle, using no more than elementary calculus, and then note some details specific to the Coiler.

Two-pole state-variable intuition

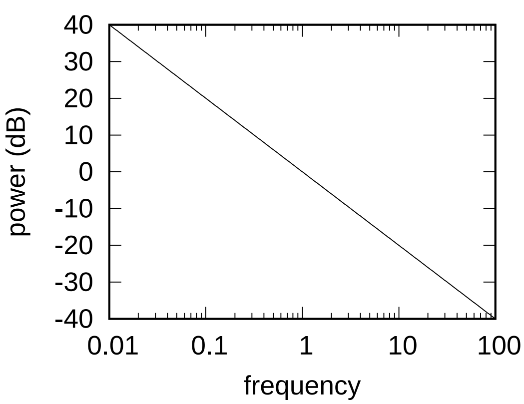
The expression $\sin fx$ describes a sine wave of frequency f radians per unit of time x. Let's take the integral of that.

$$\int \sin fx \, dx = -\frac{1}{f} \cos fx$$

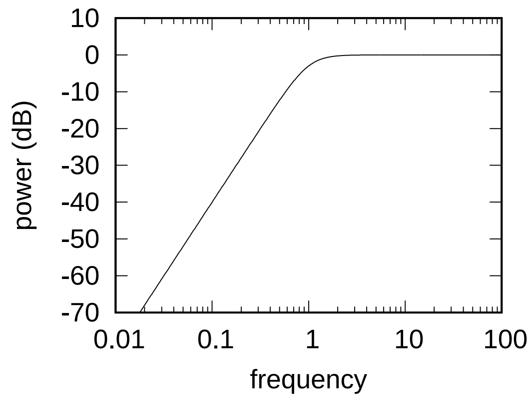
The integral of a sine wave is just another sine wave at the same frequency, shifted by 90°, and scaled by the inverse of the frequency. If we take another integral, it gets shifted another 90° for a total of 180°.

$$\iint \sin fx \, dx = -\frac{1}{f^2} \sin fx$$

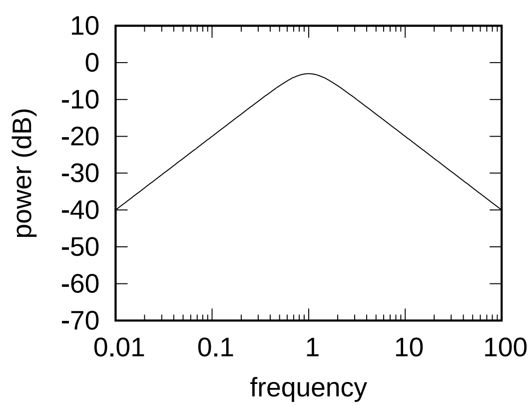
The frequency response of a voltage integrator circuit, plotted on a Bode plot, is just a downward slope of 6dB/octave or 20dB/decade. Note that it hits unity gain (0dB) at normalized frequency 1, where the $1/f$ terms in the integrals become unity.



Now, consider the frequency response of a two-pole Butterworth high-pass filter. There is some curvature in the graph around the cutoff, but significantly above the cutoff frequency, it is flat (0dB/octave), and significantly below the cutoff, it rises at 12dB/octave.

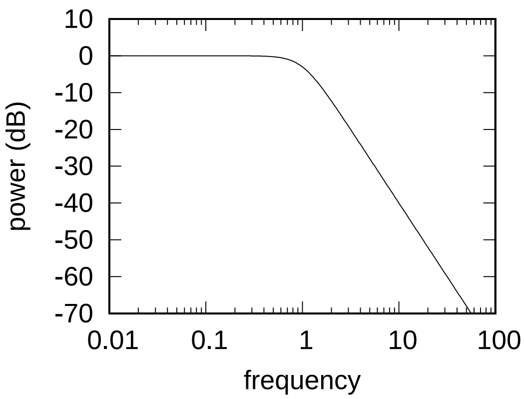


What happens if we take a circuit with a high-pass response, and integrate its output? The integrator response is falling at 6dB/octave. When applied to the 12dB/octave rise below the cutoff, that leaves a 6dB/octave rise. When applied to the flat response above cutoff, the integrator changes the response to falling at 6dB/octave. Overall, we have a band-pass response with 6dB/octave slopes on both sides.



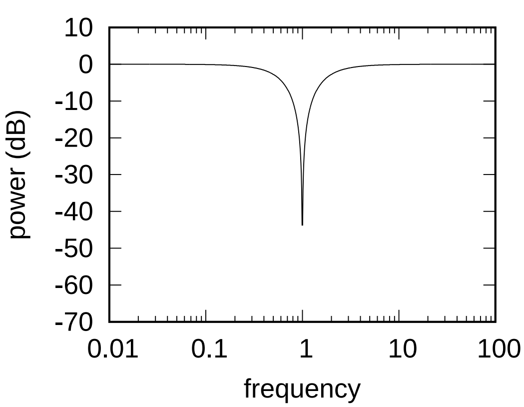
It may not show clearly at this scale, but the peak of the band-pass response does not reach 0dB but only −3dB, due to the curvature of the high-pass response in the vicinity of the cutoff frequency.

Now suppose we attach another integrator to the circuit. The lower slope, rising at 6dB/octave, is cancelled out, and the upper slope, falling at 6dB/octave, is steepened to 12dB/octave. The result is a two-pole low-pass response.



So a low-pass response is just the integral of a band-pass response, which is the integral of a high-pass. These relations provide the start of a recipe for a multimode filter: just start with a high-pass response and attach two integrators. Then we have a filter with three outputs, high-, band-, and low-pass. But where will the high-pass response come from?

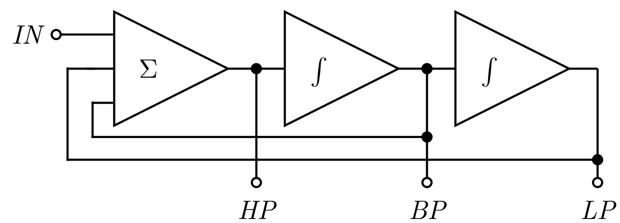
Consider the sum of the low-pass and high-pass responses. Remember that taking two integrals of a sine wave entails a 180° phase shift; a voltage inversion. At the cutoff frequency, the low-pass and high-pass responses will be equal, but with opposite phase, so they cancel out to zero. Above or below the cutoff frequency, one or the other of the low- and high-pass responses will be near 0dB while the other is much lower, and the sum will be close to 0dB. So the sum of the low- and high-pass responses is a notch response.



Adding in the band-pass response as well can fill in that notch; but because the peak of the bandpass is 3dB down, it's necessary to put in a bit of gain; specifically, boosting the voltage by $\sqrt{2}$, which doubles the power (3dB gain). With that modification, the sum of the three filter outputs recovers the input (at least, if we ignore phase).

$$IN = HP + \sqrt{2}BP + LP$$

We can solve for the high-pass response and find that it is the sum of (possibly scaled and inverted) copies of the input, band-pass, and low-pass responses. And that is the missing piece for the multimode filter. The filter core will consist of two integrators in series, driven by a mixer that combines the input and the two integrator outputs, with scaling and inversion.



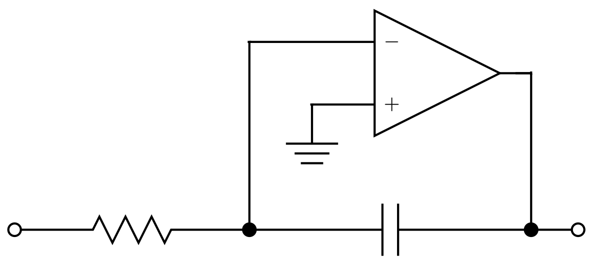
There are many variations on the basic two-pole state-variable filter. Some provide multiple inputs instead of multiple outputs, by mixing in the different inputs at different points along the chain of integrators; and the details of the scaling and inversion vary depending on the desired phase relationships among the different responses. In the Coiler in particular, the mixer that creates the high-pass output inverts all three of its inputs because that makes for a more convenient op amp circuit, and the voltage gain on the band-pass feedback path is 2 at minimum resonance instead of $\sqrt{2}$.

Note that everything in the theoretical description is scaled to the frequency at which the integrators have unity gain. If we add some gain or attenuation at each integrator, the same at both of them, then we shift that frequency without changing anything else; and so the cutoff frequency of the filter changes.

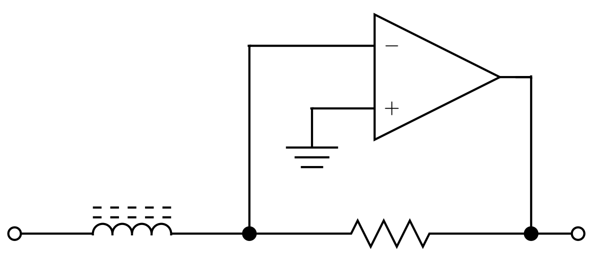
Integrators

The two-pole state-variable topology provides a framework for generating many filter designs just by swapping in different integrator, mixer, and VCA circuits. There are many such designs on the market and most of them distinguish themselves from others by using different VCAs; no VCA design gives perfect voltage multiplication, especially in situations like overdrive, and the differences in the VCAs' imperfections are supposed to give different sounds. The MSK 009 instead makes its main innovation in the integrator circuit.

Here is the topology of a standard capacitor-based voltage integrator. Bearing in mind that the op amp negative input is a virtual ground, the current through the resistor is directly proportional to the input voltage. Then the same current must flow through the capacitor (given the effectively infinite input impedance of the op amp), and the voltage across the capacitor is (by nature of a capacitor) the time integral of the current. Voltage out for the circuit is negative time integral of voltage in, scaled by component values.

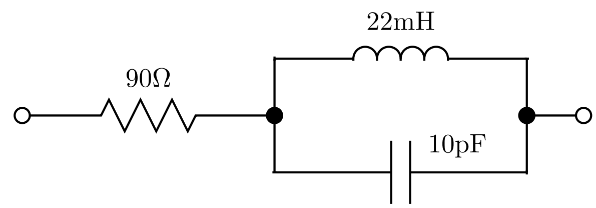


The capacitor-based integrator goes from voltage input to voltage output, with a 90° phase shift corresponding to integration, by internally using a current in phase with the input. But we can also design a similar circuit that will use a current in phase with the output, as seen below.



The current through the inductor is, by nature of an inductor, the time integral of the voltage across the inductor. Then the currents must balance at the op amp negative input, so the same current flows through the resistor, which converts it back to a voltage. With ideal components, this inductor-based voltage integrator is exactly equivalent to the capacitor version.

But we do not have ideal components, especially not in the case of inductors. Small to medium capacitors are capable of approaching ideal behaviour at audio frequencies, but inductors seldom come close, especially not if they are of practical physical size and cost. In particular, the EPCOS ferrite bobbin inductors used in the Coiler VCF behave very much as the following circuit made from ideal components would behave.



This equivalent circuit was found by a combination of measuring real-life samples of the components, and fitting simulation results to the curves in the data sheet.

When one of these non-ideal inductors is balanced against a plain resistor in the inductive op amp integrator circuit, the behaviour depends on the frequency. At very high frequencies (far above audio), the 10pF capacitance dominates. The inductor behaves more like a capacitor, and the circuit is more like a differentiator than an integrator. At sufficiently low frequencies, the inductance and capacitance are not significant and the coil's behaviour is mostly controlled by its 90Ω resistance. Then the integrator circuit behaves like a plain inverting amplifier. With the component values used in the Coiler VCF, the point at which the inductors start becoming more like resistors is at frequencies below about 1kHz; well into the audio range.

So to balance the component that is like an inductor at higher frequencies and like a resistor at lower frequencies, we will use something that is like a resistor at higher frequencies and like a capacitor at lower frequencies, namely a series circuit of a resistor and a near-ideal capacitor. Over the audio spectrum, the integrator shifts smoothly between capacitor-based integration (resistor on input, capacitor in feedback loop) and inductor-based integration (inductor on input, resistor in feedback loop).

See Figure 3. The component L1 is labelled a 22mH inductor, but bear in mind that because of the non-ideal behaviour of real-life components, it only provides a near-pure inductance over a limited range of frequencies. At the bottom of the audio range, it is more like a resistor. The coil's capacitance can be ignored for most of this discussion; that only comes into play at frequencies that should never be present in this circuit. The coil is at the input to the op amp U2D, whose feedback loop contains a 6800pF capac-

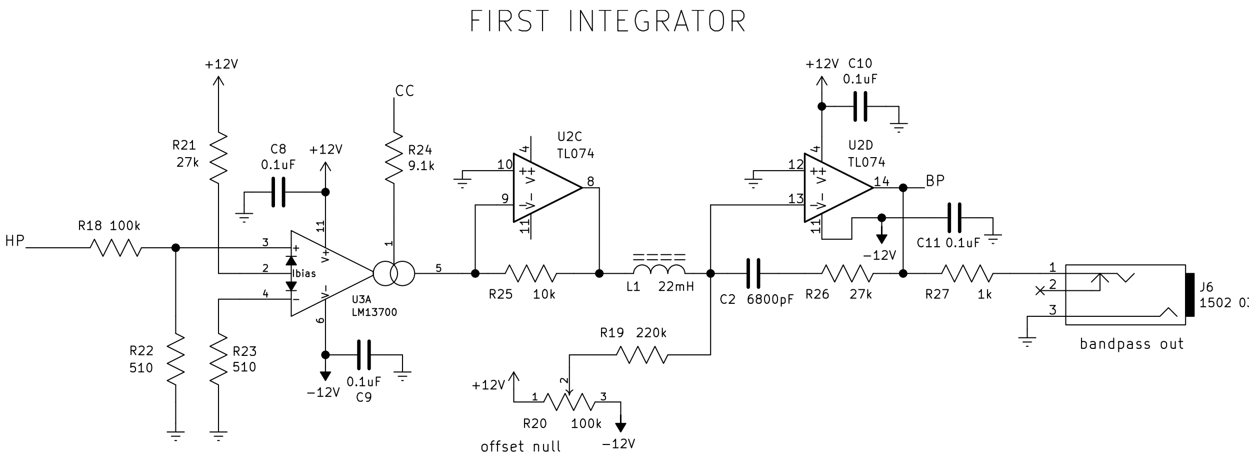


Figure 3: One integrator in the filter core.

itor and 27kΩ resistor. At high audio frequencies, when the inductor is really providing inductance, the capacitor has low impedance and the feedback loop looks like a resistor; then we have the inductor input, resistor feedback style of op amp integrator. At low frequencies, when the inductor looks more like a resistor, the capacitor will have significant impedance and the circuit functions as the resistor input, capacitor feedback style of op amp integrator.

The resistance and capacitance values were chosen by simulation and prototyping to give phase response as flat as possible. However, this response is not completely flat, not least because the model of the coil's behaviour over frequency is only approximate. As frequency varies over the audio range, the phase response of the integrators, and therefore the overall sound of the filter, will vary.

The other components in Figure 3 are basically all support for the integrator components L1, C2, R26, and U2A. Immediately before the coil, the op amp U2C and feedback resistor R25 function to convert the current from the OTA into a voltage. This section is needed because the OTA, U3A, produces its output signal as a *current* level instead of a voltage. Traditional capacitor-based integrators actually take current input anyway, and use an input resistor to convert voltage to current; when using such an integrator with an OTA, it is usual to leave out the input resistor and run the OTA directly into the op amp's virtual ground. Keeping the OTA output at a fixed voltage helps mimize distortion. But becaus the Coiler's coils require real voltage input, not current, it's necessary to convert the current to a voltage, and U2C and its feedback resistor do that while allowing the OTA output to stay at 0V.

Immediately before the current to voltage converter comes the operational transconductance amplifier U3A, with its support components. This is the variable-gain element used for tuning the filter. It is a fairly conventional design. Note the linearizing diode current supplied by R21. This part is often left out of LM13700-based designs, but I like to include it both because it improves distortion performance and because it makes the gain of the amplifier more predictable, requiring less trimming. The gain of the amplifier is linearly controlled by a control current (labelled "CC" on the diagram) from the exponential converter to be described later. The 9.1kΩ resistor on that input helps both with keeping the gain of the two integrators the same, and with limiting the control current to a safe level. The exponential converter is not capable of driving its output voltage above 0V, so the maximum possible current through R24 is about 1.2mA, not enough to damage the LM13700.

The offset null trimmer R20, with its scale-setting resistor R19, helps to compensate for input offsets in the op amps and OTAs (most significantly, the OTAs). There are seven IC amplifiers in the large feedback loop: one in the input mixer and three in each of two integrators. In principle, every one of those amplifiers has an input offset, and trimming all of them would require seven adjustments. That is too many adjustments to be practical in a commercial module.* Instead of trimming every single amplifier,

*I learned my lesson with the Leapfrog, which has 13 trimmers.

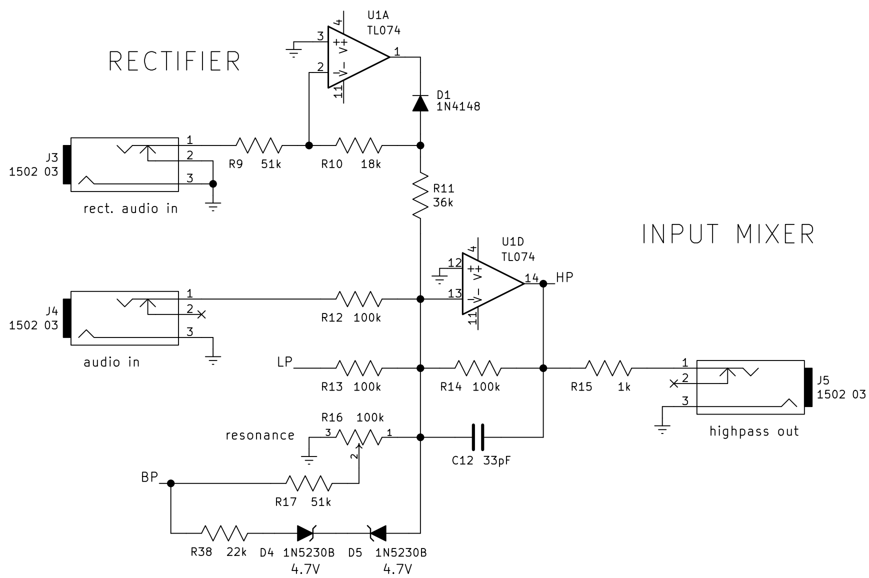


Figure 4: Input mixer and rectifier.

there is one trimmer on each integrator, and the idea is to average them out. The exact consequences of adding or subtracting an offset at this point in the circuit are complicated; I used simulation to determine that R20 primarily affects the externally visible offset on the HP and LP outputs, and R30 primarily the BP and LP. Not having precise trimming of the OTA offsets leads to a more than trivial amount of control feedthrough, but I'm treating that as just part of the unique sound of this module. When the control signal is coming from an envelope, some envelope feedthrough into audio may help listeners perceive the envelope as "snappy."

The second integrator has just the same circuit as the first, and works the same way.

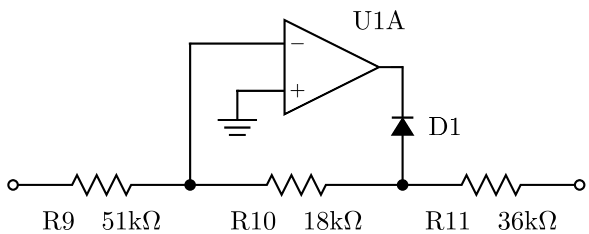
Input mixer and rectifier

The input mixer and rectifier sections are shown in Figure 4. The input mixer is straightforward: it is a standard inverting op-amp summer, with inputs from the module input J4 and the low-pass and band-ass integrator outputs. The capacitor C12 is for stability at high frequencies; since the capacitance of the coils can make the integrators stop being integrators and switch over to doing differentiation once the frequency gets high enough, there is the danger that the entire module could go into parasitic oscillation at ultrasonic frequencies should there be enough feedback around the loop at such frequencies. The op amps themselves have limited frequency range, but maybe not limited enough; the 33pF capacitor kills the gain in the ultrasonic range and makes sure parasitic oscillation will be impossible.

The feedback pass from the band-pass output is responsible for the "resonance" control and (deliberate) oscillation of the module. Note the pin numbers on the panel pot R16: when this knob is turned to its user-visible "minimum" (full counterclockwise), the gain on the BP feedback path is actually *maximized* at about 2. This feedback path inhibits oscillation. Increasing resonance reduces the gain on it, ultimately as far as zero, at which point the module should oscillate reliably.

If the only feedback is through the low-pass feedback path, then the oscillations will build steadily until they drive the amplifiers into clipping. The output amplitude in that case is somewhat unpredictable (because it depends on internal offsets we can only partially trim) and is usually more than desired (±5V being a good target for maximum output in Eurorack). The clipping components R38, D4, and D5 help keep it under control. When oscillation reaches about ±5V, the diodes start to conduct, increasing the gain on the feedback path and inhibiting further increase in the oscillation amplitude. This specific placement of the diodes (on the BP path, feeding into the op amp virtual ground) was chosen largely by trial and error; I tried putting limiting circuits at several other points but found they had the most reliable and pleasant-sounding effects here.

The rectifier section, shown again below, is an unusual one. Note that unlike most op-amp precision full-wave rectifiers, this uses only one op amp and one diode. It's a near variation of one I have seen attributed to "Tompkins, W.J., and J.G. Webster (eds.), *Interfacing Sensors to the IBM PC.* Prentice-Hall, 1988" but I have not actually read that reference.



Suppose the input voltage is positive. In order to keep its negative input at 0V, the op amp U1A must draw the same current through R10 that flows through R9, and that means the voltage at the diode anode is $-18/51$ times the input voltage. Bearing in mind that the input mixer's summing node is kept at 0V with a 100kΩ input impedance set by R14, the feedback resistor on U1D, the gain from the diode anode to the input mixer's output (module HP output) is $-100/36$, and that combined with the rectifier's $-18/51$ gain is unity, to within component tolerances.

But suppose the input voltage is negative. Nothing can drive the U1A negative input to zero. The only current sources available are the virtual ground at the U1D negative input, which can only bring U1A's negative input *halfway* to zero through the resistor network; and the output of U1A, which is on the far side of a reverse-biased diode. Then the U1A negative input remains below the positive input of 0V, U1A output heads for the positive rail, but will have no effect, and the input voltage has a straight shot through the series combination of R9, R10, and R11 (totalling 105kΩ) into the 100kΩ *inverting* impedance of the mixer's summing node. Gain from the rectifier input to the mixer's output is close to *negative* unity.

On positive input the gain is positive unity, and on negative input the gain is negative unity. Positive and negative input voltages both will translate to positive output voltages; this is a full-wave rectifier.

The full-wave rectifier certainly isn't perfect. There will be some difference in gain between the positive and negative half-cycles due to the use of standard components close to, rather than exactly, the right ratios. The op amp is being used with a large differential input range, driven into saturation on negative input, which is not ideal. But note that because of the voltage division involved its input cannot go below about -6V for the rated ± 12V input range, and so the phase inversion problem with FET-input op amps like the TL074 is not an issue here. The biggest problem with this one-diode, one-op-amp full-wave circuit is that it only works correctly when driving a fixed input impedance. That is the case here because we have the well-behaved summing node of the input mixer to drive, but the circuit would require some sort of buffering if it were providing a rectified output directly to the outside world. Actually, the story of this circuit is that the rest of the module required seven op amps, they come in packages of four, and I didn't want to leave one unused. The single-op-amp full-wave rectifier circuit nicely used up available resources while providing a musically useful extra feature.

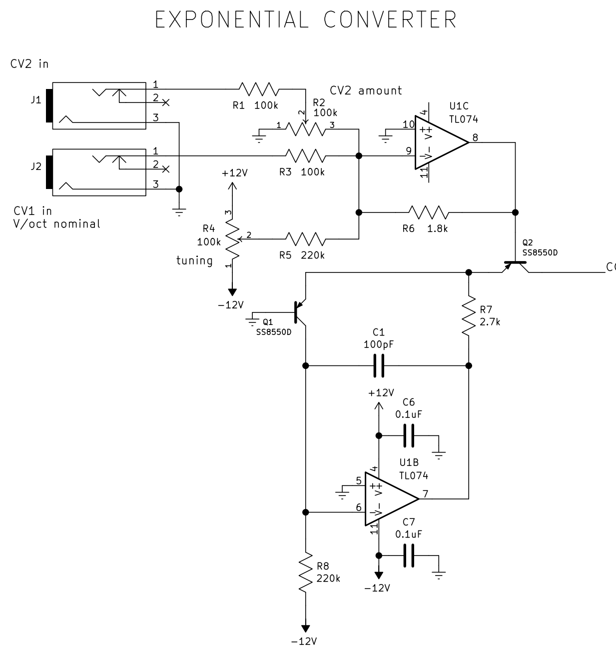


Figure 5: Exponential converter and control voltage processor.

Exponential converter

The exponential converter is shown in Figure 5. It's a fairly conventional design, with partial temperature compensation.

The 1.8kΩ feedback resistor R6 sets the gain for U1C, on the CV1 input with its 100kΩ input resistor, to -18mV per volt of control voltage input. That is (approximately) the right amount to apply to the base of a silicon transistor to double the collector current; so this gives approximately 1V/octave response. The CV2 and tuning-knob inputs are straightforward, one having an attenuator and the other applying an offset to the summing node of the front-end amplifier.

The second op amp, U1B, is the "servo" op amp for the temperature compensation. Its function is to keep the emitters of both transistors at whatever voltage is necessary (probably around +0.7V) for such a transistor to pass about 54μA when its base is at 0V. The transistor Q1, called the reference transistor, has its base permanently connected to 0V. The current through this transistor is set by R8; constant current because the high side of R8, and Q1's collector, are kept at 0V by op amp feedback. Then the op amp drives the emitters of the transistors as necessary to maintain the right emitter-base voltage for that current level. This implies that if the control voltage from U1C happens to be zero, the output current through Q2 to the "CC" connection (which feeds the integrators) will also be 54μA. For each additional volt on the V/oct input, the control voltage applied to Q2 goes down 18mV, increasing the emitter-base voltage by 18mV and doubling the current.

The reason for this complexity is that the desired emitter-base voltage for a fixed current level can vary quite a bit with temperature. If we used a single transistor and a fixed assumption for the offset of the emitter-base voltage, then the filter's frequency would change with any small temperature variation, making tuning difficult. Here, we make some effort to keep Q1 and Q2 at the same temperature and then any change in the required emitter-base voltage for a fixed current will be detected and applied by the servo op amp; the front-end op amp is only controlling the *difference* between the emitter-base voltages of Q1 and Q2. This takes care of the main temperature effect on tuning.

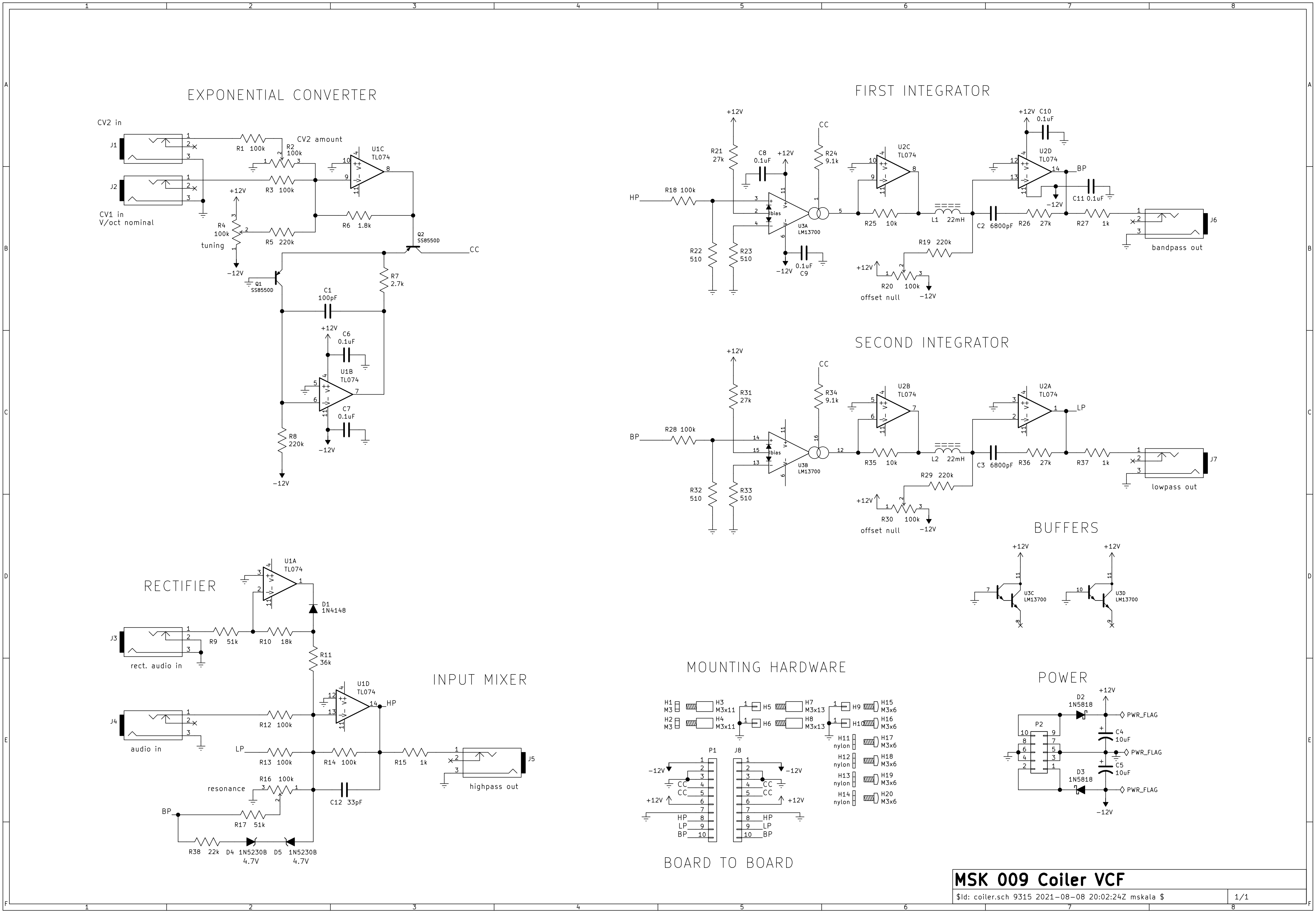
There is another temperature effect in the fact that the "18mV" quantity representing how much differential voltage to apply for a doubling of current, is itself temperature-dependent. In the Coiler, there is no effort made to compensate that voltage. There are enough other inaccuracies and unpredictable things in the design that it seems not to be worthwhile; users requiring precise tracking will be using other filters, such as the MSK 007 Leapfrog. But when this kind of exponential converter is used in other applications, it is common practice to apply temperature compensation in the front-end amplifier as well, often by replacing the feedback resistor with a carefully chosen temperature-sensitive resistor or network.

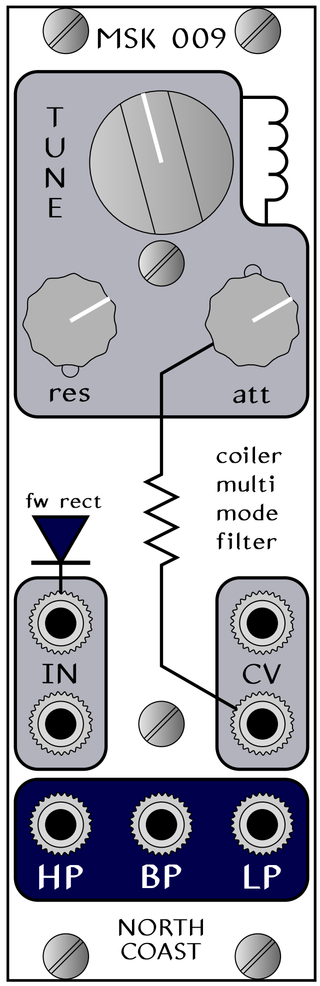
Two other components worth mentioning in this section are C1 and R7, both of which are intended to keep the op amp U1B stable. Q1 has a lot of voltage gain between its emitter and collector. The current through the transistor, and therefore the voltage at the collector given the current-to-voltage characteristic of R8, is an exponential function of the emitter-base voltage. If we connected the emitter directly to the output of U1B, then U1B would be in negative feedback with significant voltage gain around the feedback loop, and that is likely to push it into parasitic oscillation. Op amps in general are not rated for stability with feedback loop gain greater than unity. So the resistor R7 is included to cut down the gain around the loop. With that resistance in series with the emitter-base drop, it takes a much larger voltage change on the op amp output to produce a significant change in current. The capacitor C1 also serves to improve stability by putting in a phase shift at high frequencies. The servo op amp should not respond faster than the highest modulation frequency of the filter, so this capacitor prevents it from responding, and possibly oscillating, in the ultrasonic range.

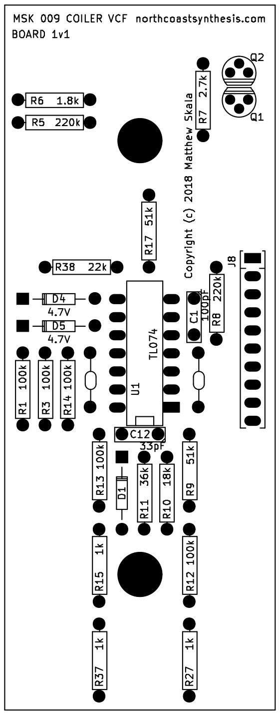
Mechanical drawings

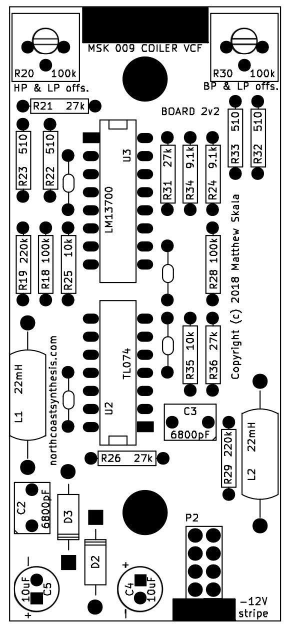
On the following pages you will find:

- the schematic diagram for the module;
- a mock-up of what the completed module looks like from the front panel;
- the top-side silk screen art showing component placement;
- the bottom-side silk screen art showing component placement (*note this drawing is mirrored, and shows what you actually see looking at the board, not the X-ray view used in other Kicad output*);
- a drawing of the front panel, with the hole locations and other information for manufacturing it; and
- an exploded isometric drawing showing how the boards and hardware fit together.


EXPONENTIAL CONVERTER
FIRST INTEGRATOR
SECOND INTEGRATOR
BUFFERS
RECTIFIER
INPUT MIXER
MOUNTING HARDWARE
POWER
BOARD TO BOARD
CV2 in
CV1 in
V/oct nominal
CV2 amount
tuning
rect. audio in
audio in
resonance
highpass out
bandpass out
lowpass out
offset null
MSK 009 Coiler VCF
$Id: coiler.sch 9315 2021-08-08 20:02:24Z mskala $
1/1

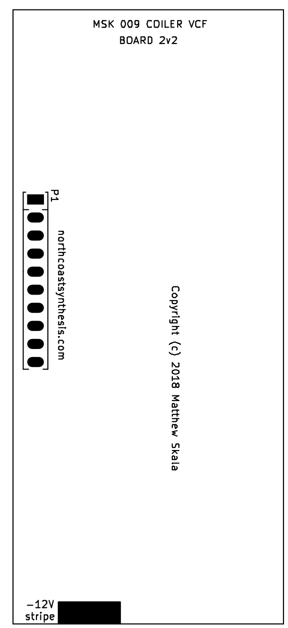

MSK 009
TUNE
res
att
coiler
multi
mode
filter
fw rect
IN
CV
HP
BP
LP
NORTH
COAST

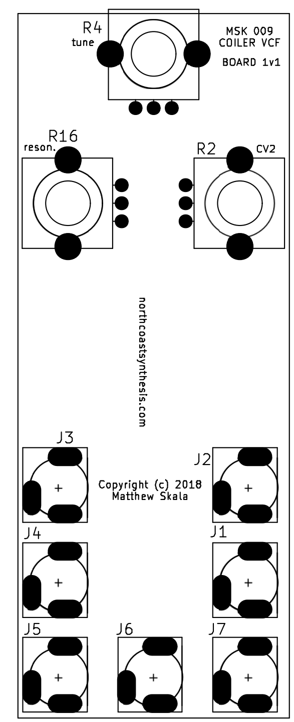




MSK 009 Coiler VCF

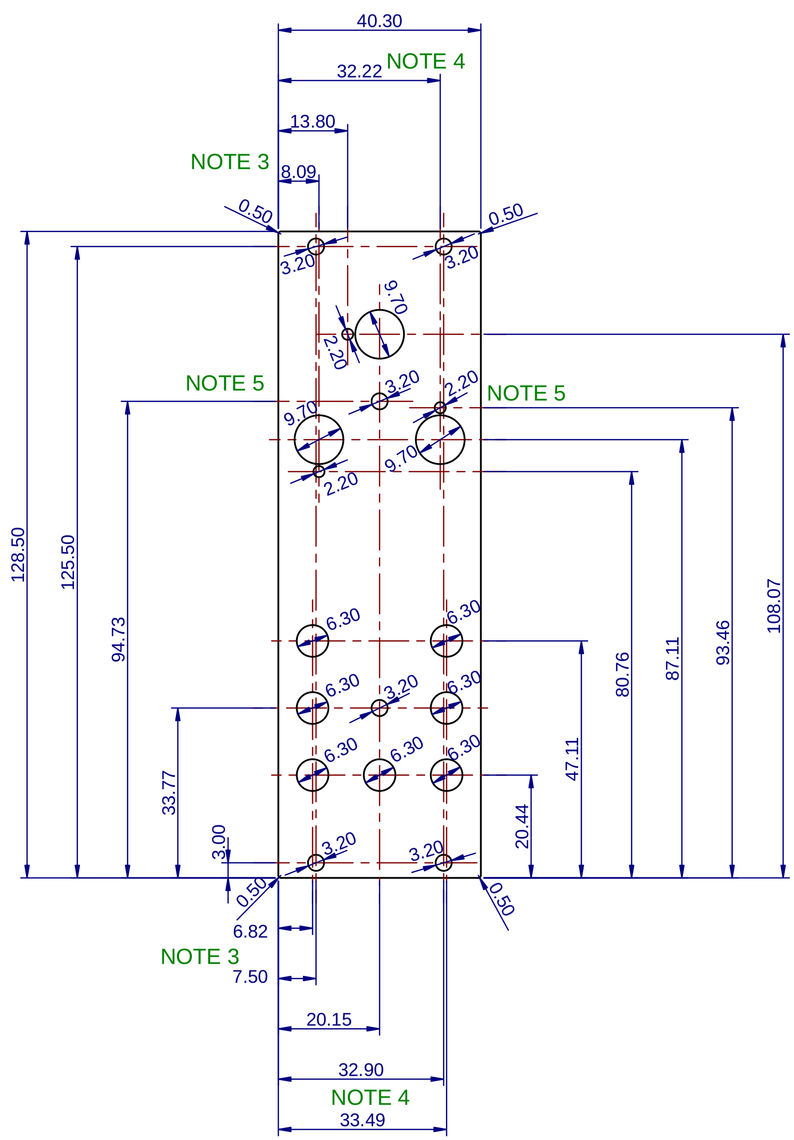
"$Id: coiler.kicad_pcb 6464 2018-11-26 20:40:37Z mskala $" | 1/2





MSK 009 Coiler VCF

"$Id: coiler.kicad_pcb 6464 2018-11-26 20:40:37Z mskala $" | 1/2



NOTES

1: All dimensions are millimetres with tolerance +/- 0.10mm. Do not scale from drawing.

2: Material is aluminum, 2.00mm thick. Overall dimensions are 128.50mm by 40.30mm, per Doepfer spec for an 8HP Eurorack synthesizer module.

3: Do not confuse these similar horizontal dimensions. Three 6.30mm holes are 6.82mm from the left edge. Two 3.20mm holes are 7.50mm from the left edge. One each 9.70mm and 2.20mm holes are 8.09mm from the left edge.

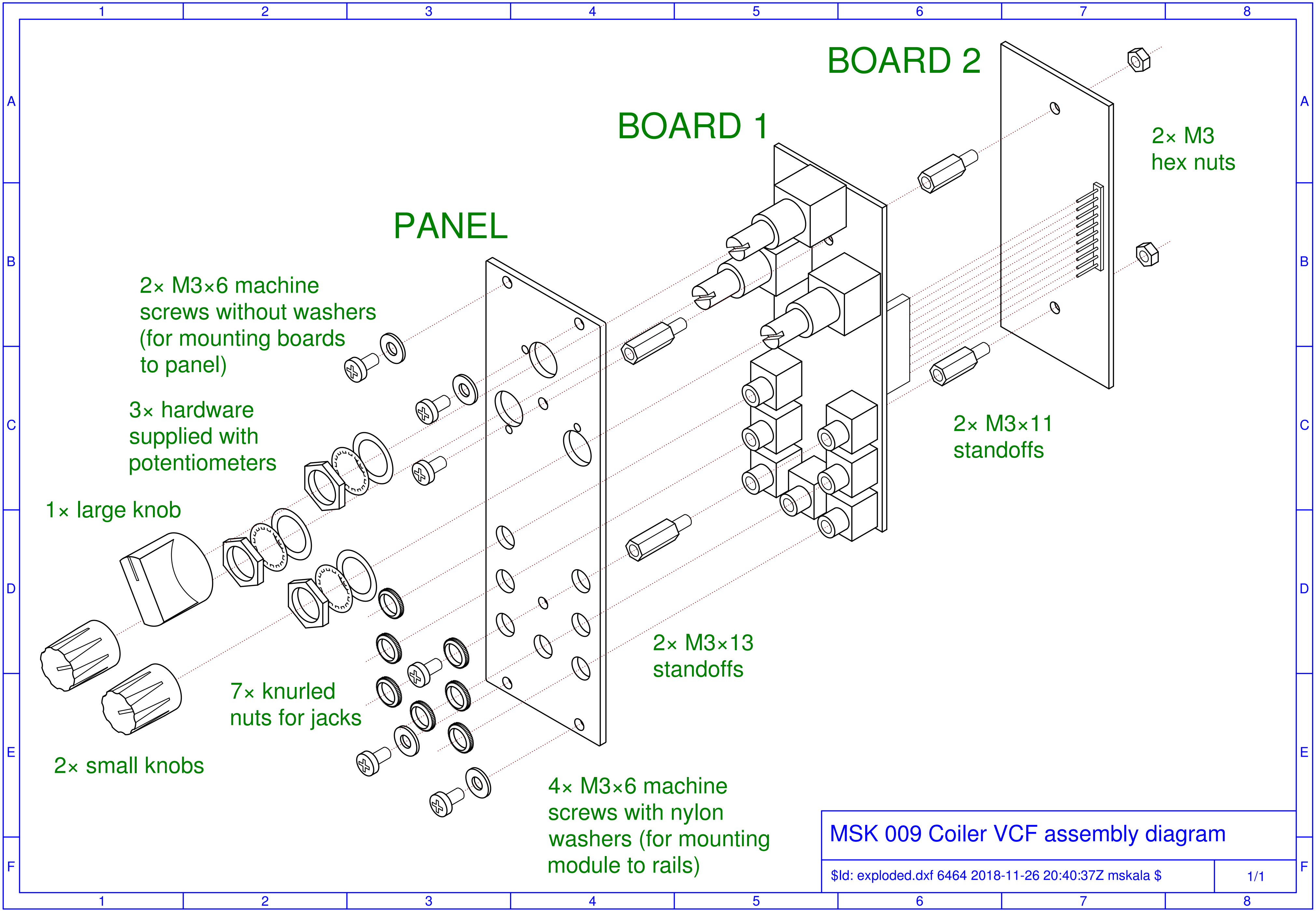
4: Do not confuse these similar horizontal dimensions. One each 9.70mm and 2.20mm holes are 32.22mm from the left edge. Two 3.20mm holes are 32.90mm from the left edge. Three 6.30mm holes are 33.49mm from the left edge.

5: Do not confuse these similar vertical dimensions. The 2.20mm hole is 93.46mm from the bottom edge. The 3.20mm hole is 94.73mm from the bottom edge.

MSK 009 Coiler VCF panel mechanical

$Id: panel-mechanical.dxf 6372 2018-10-25 00:48:48Z mskala $

1/1


BOARD 2
BOARD 1
PANEL
2× M3 hex nuts
2× M3×6 machine screws without washers (for mounting boards to panel)
3× hardware supplied with potentiometers
1× large knob
2× M3×11 standoffs
2× M3×13 standoffs
7× knurled nuts for jacks
2× small knobs
4× M3×6 machine screws with nylon washers (for mounting module to rails)
MSK 009 Coiler VCF assembly diagram
$Id: exploded.dxf 6464 2018-11-26 20:40:37Z mskala $
1/1