MSK 012 Transistor ADSR

North Coast Synthesis Ltd.
Matthew Skala

August 31, 2021

Contents

General notes

This manual documents the MSK 012 Transistor ADSR, which is an envelope generator for use in a Eurorack modular synthesizer. It implements a traditional attack-decay-sustain-release envelope without using any integrated circuits, performing all the logic and other active functions with discrete transistors and diodes.

Specifications

The module's peak current requirement in ordinary use is 10mA on the +12V supply and 5mA on the −12V supply; most of the time it will draw much less. Unusual loads on the output, including so-called "passive" modules, may cause the MSK 012 to draw more than this amount of current. It does not require +5V power.

The input impedance is nominally 100kΩ. The maximum output impedance is about 3.5kΩ (low side) or 1kΩ (high side). Shorting the input or output to any fixed voltage at or between the power rails should be harmless to the module; patching the MSK 012's output into the output of some other module on the same power system should be harmless to the MSK 012, though doing that is not recommended because it is possible the other module may be harmed.

Attack time is adjustable from about 40μs to 1.9s in three ranges; decay time from about 6μs to 6.8s; and release time from about 8μs to 9.5s. All these values were measured on a prototype and may vary a little with component tolerances. The attack curve is concave downward and decay and release are concave upward.

The peak voltage of the envelope is fixed at about 8V; sustain voltage ranges between 0V and the peak. The input uses a Schmitt trigger (not dependent on fast edges), turning on at about 2V and off at about 1V. The input is treated as a gate, not a trigger: when it goes low, the module immediately goes into the release phase (whether it was previously in attack, decay, or sustain) and when it goes high, the module immediately goes into the attack phase (regardless of whether the release had completed).

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 suffer or cause any damage. Some other kinds of misconnection may possibly be dangerous to the module or the power supply.

Front panel controls and connections

Here's a summary of the items on the front panel of the module. All four knobs are logarithmic ("audio") taper.

A knob attack time
D knob decay time
S knob sustain level
R knob release time
range switch sets the general range of the three timing knobs, up for slow and down for fast (labelled as such), or in the unlabelled centre position for very fast
G input gate control voltage
E output envelope control voltage

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

This module is built on a single PCB 4.20″×1.25″, or 106.68mm×31.75mm, which mounts perpendicular to the front panel. With about another 1mm of gap between the PCB and panel, the total depth requirement is 33mm.

Component substitutions

This circuit should work with most general-purpose bipolar transistors; I specify 2N5088 and SS8550D (formerly, PN200A). These are high-gain types. I use these same transistors in other North Coast projects. The output buffer transistors (Q4 and Q5) are the ones where the gain matters most; be careful substituting those.

You can substitute other capacitor values to change the overall speed of the envelopes. However, I would not recommend using electrolytics because of their higher leakage, which could be a problem for the long envelope times where they'd be most likely used; and although substituting a smaller value for C3 to make the fastest envelopes even faster will work up to a point, I don't recommend it. Most people who think they want extremely fast envelopes are wrong. The problems they think they can hear that they think are caused by the envelope not being fast enough are more often caused by the envelope being *already much too fast*, and will not be solved by making it even faster. Also, I encountered some stability problems with this module's circuit on the breadboard with a value of 0.01μF for C3, specifically that the module would go into oscillation when the *decay* (not attack) was set to its absolute minimum time. I couldn't reproduce that with a module built on a PCB, so I think it was caused by stray impedances on the breadboard, and the final design value for C3 is 0.033μF anyway (to make the longer times in the fastest range more convenient). But it suggests that there are some limits to how far into the ultrasonic it's reasonable to push this module's timing.

Modification for ±15V power

To modify the circuit to run on ±15V power while keeping roughly the same output voltage range, make sure all components are rated for the increased voltage, and try changing these resistors.

- R7 (upper limit of sustain level): change from 20kΩ to 33kΩ.
- R16 (sensitivity for end-of-attack detector): change from 27kΩ to 18kΩ.

For a higher output peak on ±15V power, R7 will need to decrease and R16 increase; for a lower peak, the other way around. These values were determined by simulation, not testing a real instance of the circuit, so some breadboarding and experimentation may be necessary.

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 https://northcoastsynthesis.com/. Your support of my business is what makes it possible for me to continue releasing module designs for free. Even if you only use the free plans and cannot buy the commercial products I sell, any assistance you can offer to increasing the profile of North Coast would be much appreciated. For instance, you might post photos of your completed DIY build on your social media. The latest version of this document and the associated source files can be found at the North Coast Web site.

Email should be sent to mskala@northcoastsynthesis.com.

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 still 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.	
2	C1, C2	470pF	radial ceramic, 0.2″ lead spacing
1	C3	0.033μF	film, 0.2″ lead spacing
2	C8, C9	0.1μF	axial ceramic
1	C5	0.22μF	film, 0.2″ lead spacing
1	C4	4.7μF	film, 0.2″ lead spacing
2	C6, C7	10μF	radial aluminum electrolytic, 0.1″ lead spacing
15	D1–D15	1N4148	or 1N914; switching diode
2	D17, D18	1N5818	or SB130; Schottky rectifier
1	D16	LTL2R3KRD-EM	hi-efficiency red LED, Lite-On
2	H3, H4	M3x6	M3 machine screw, 6mm body length
2	H1, H2		nylon washer for M3 machine screw
1	J3		male Eurorack power header, 2 × 5 pins at 0.1″
2	J1, J2	MJ–3536	switched mono 3.5mm right angle jack, CUI
5	Q1, Q2, Q4, Q8, Q9	2N5088	NPN general purpose amplifier, TO-92 EBC
4	Q3, Q5–Q7	SS8550D	or PN200A; PNP high gain, TO-92 EBC
1	R21	1kΩ	
2	R2, R26	3.6kΩ	
1	R27	7.5kΩ	
3	R7, R17, R18	20kΩ	
1	R16	27kΩ	
7	R5, R6, R9, R12, R23–R25	39kΩ	
1	R10	100kΩ	conductive plastic panel pot, BI Technologies P160KNP series, audio taper
2	R8, R13	100kΩ	
5	R14, R19, R20, R22, R29	270kΩ	
1	R3	680kΩ	
3	R4, R11, R15	1MΩ	conductive plastic panel pot, BI Technologies P160KNP series, audio taper
2	R1, R28	1MΩ	
1	SW1	100SP3T1B1M1QEH	E-Switch 100-series SPDT on-off-on toggle

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

Also needed: solder and related supplies, PCB, panel, knobs, wire, heat-shrink tube, Eurorack power cable, etc.

Building the module

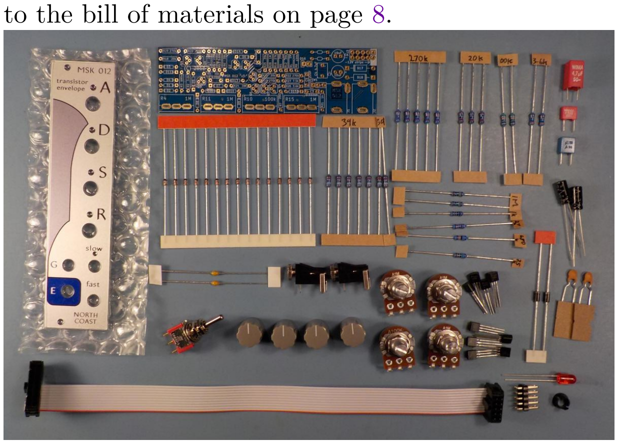
In order to achieve 6HP width, this module is built with a couple of components (the switch and LED) mounted off the board and connected by "flying" hookup wires. The board is crowded, especially where the wires attach, so it is recommended to install the components in the order described here: first the small on-board components, then the wired-in panel components, then the larger on-board components, and finally the rest of the panel components. If you leave the wired components until too late, it can be difficult to fit the wires into the board.

Although I'm describing a separate step for each component value, and that's how I built mine so as to have plenty of photo opportunities, if you are reasonably confident about your skills you may find it easier to put most components in place first and then solder them in a single step. Except for the overall sequence of putting in the wires before the board gets too crowded, the order does not matter much.

The board shown in my photos here is a version 3 board, which is the version shipped in North Coast kits; but some earlier boards were used in prototypes and early runs of pre-assembled modules. If you disassemble a pre-assembled module, there may be minor differences in the silkscreen, in particular showing component values that were changed in development.

Preliminaries

Count out the right number of everything according to the bill of materials on page 8.



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.

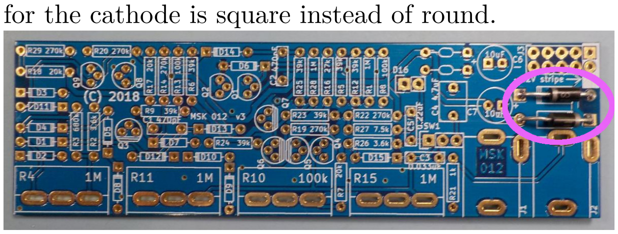
- Some photos in these instructions were taken with the older grey knobs, and some dealers may still have kits containing grey knobs in their stock, but newer kits will have blue knobs.
- Do not overtighten the setscrews when attaching the knobs! The screw should be tight enough to hold the knob onto 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.

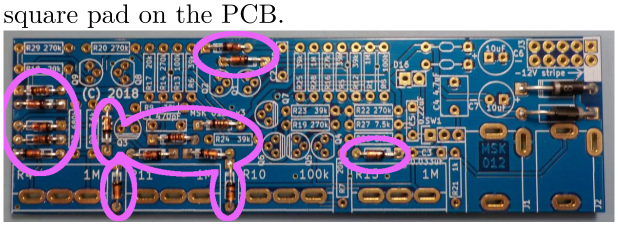
- Because it's a standard M3 thread, in a pinch it's possible to substitute a plain M3 machine screw such as are commonly used with Eurorack cases, although one of those would obviously look less nice.

Diodes

Install the two Schottky diodes D17 and D18. These protect the module against reverse connection of the power supply. They are polarized and must be installed in the correct direction; otherwise they will prevent the module from operating. One end of each diode will be marked, usually with a stripe of grey paint around the black plastic body of the diode. That end is the *cathode*. The diode outline on the PCB silkscreen is marked with a similar stripe showing the direction of the cathode, and the solder pad for the cathode is square instead of round.

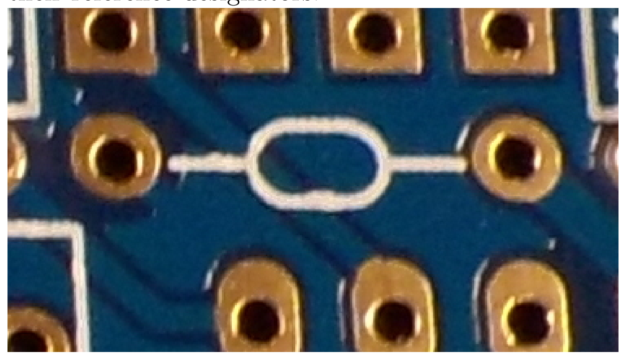


Install the 15 1N4148 or 1N914 switching diodes D1 to D15. These switch currents around, and provide controlled voltage drops, to sequence the different stages of the envelope. As with the Schottky diodes, these are polarized, with the cathode indicated by a stripe on the diode body (usually a black stripe on the orange-pink glass) and a stripe and square pad on the PCB.

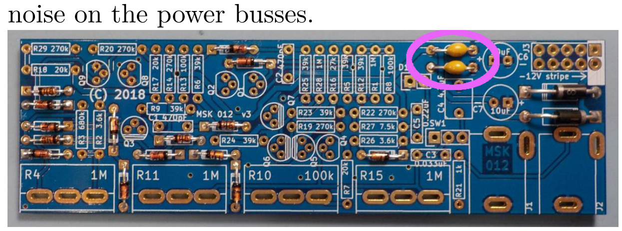


Decoupling capacitors

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



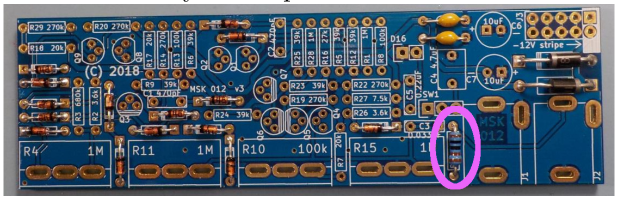
Install these capacitors where the symbol appears. They are not polarized and may be installed in either orientation. They act as filters for high-frequency noise on the power busses.



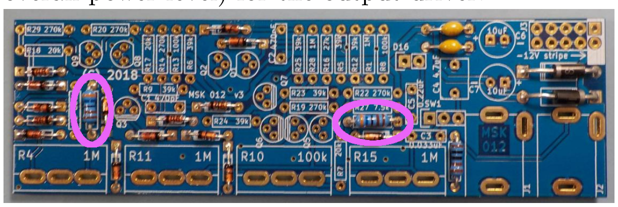
Fixed resistors

Resistors are never polarized. I like to install mine in a consistent direction for cosmetic reasons, but this is electrically unnecessary. In this module, metal film 1% resistors are recommended for all fixed-value resistors. These will usually have blue bodies and four colour bands designating the value, plus a fifth band for the tolerance, brown in the case of 1%. These are the resistors normally shipped in the North Coast kits, but we may occasionally ship better-tolerance resistors (such as 0.5%) 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.

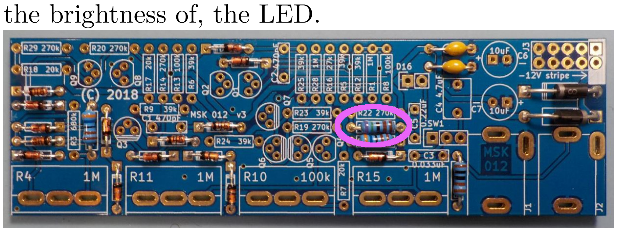
Install the 1.0kΩ (brown-black-black-brown) resistor R21. This is a current-limiting resistor for the envelope output, to protect both the MSK 012 and the external circuit in case the output is connected to an excessively low impedance.



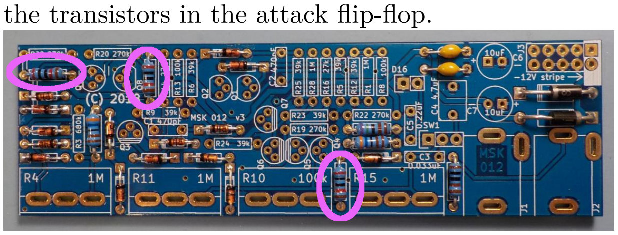
Install the two 3.6kΩ (orange-blue-black-brown) resistors R2 and R26. The resistor R2 limits the maximum current into the capacitor during attack, and the resistor R26 is the emitter resistor (which sets overall power level) for the output driver.



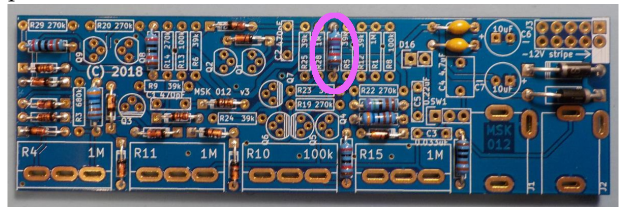
Install the 7.5kΩ (violet-green-black-brown) resistor R27. This sets the current through, and therefore the brightness of, the LED.



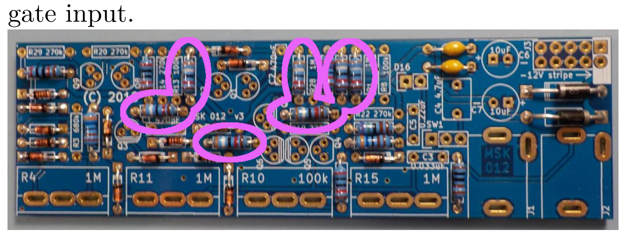
Install the three 20kΩ (red-black-black-red) resistors R7, R17, and R18. The first of these, R7, sets the high end of the adjustment range for sustain voltage; R17 and R18 are collector resistors, controlling the current drawn in the "on" and "off" states, for the transistors in the attack flip-flop.



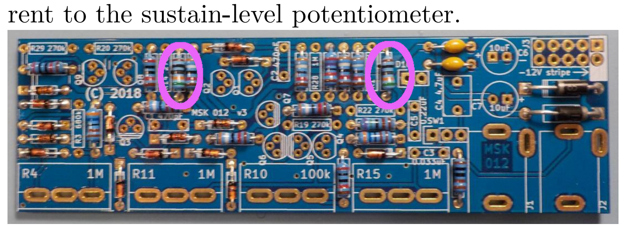
Install the 27kΩ (red-violet-black-red) resistor R16. Do not confuse this one with the five 270kΩ resistors to come later, which have orange bands in the fourth colour code position. This resistor controls the trigger level for the output Schmitt trigger, and thus the peak voltage attained during the attack phase.



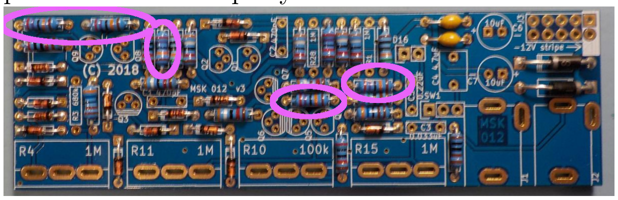
Install the seven 39kΩ (orange-white-black-red) resistors R5, R6, R9, R12, R23, R24, and R25. Four of these (R5, R6, R23, and R24) are collector resistors for the transistors in the input and output Schmitt triggers. Two (R9 and R25) control the pulse length for the pulse outputs of those Schmitt triggers. The remaining one (R12) controls the sensitivity of the gate input.



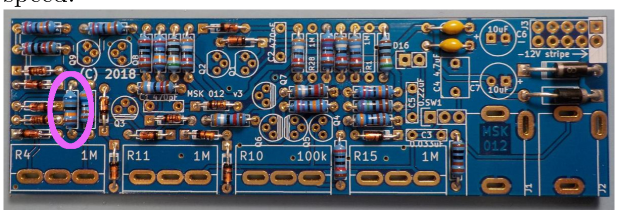
Install the two 100kΩ resistors (brown-black-black-orange; photo shows 0.5% resistors with green tolerance bands) R8 and R13. The first sets the module's input impedance and the second provides current to the sustain-level potentiometer.



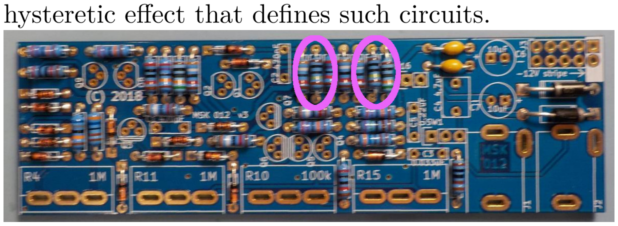
Install the five 270kΩ (red-violet-black-orange) resistors R14, R19, R20, R22, and R29. These are used in multiple locations throughout the module to provide weak default voltage or current levels, which will be overridden by stronger signals at the appropriate points in the envelope cycle.



Install the 680kΩ (blue-grey-black-orange) resistor R3. This sets the overall range of the attack speed.



Install the two 1.0MΩ (brown-black-black-yellow) resistors R1 and R28. These act as weak positive feedback paths in the Schmitt triggers, creating the hysteretic effect that defines such circuits.

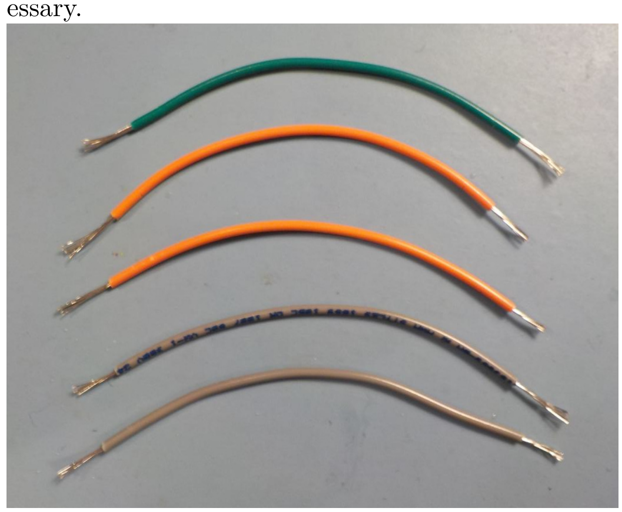


Wired panel components

First, some notes on wire. You will need five pieces of hookup wire, each about 3 inches (7.5 centimetres) long, preferably in at least three colours. North Coast kits are specified to come with three pieces each 6 inches long, in three different colours; cut each in half and you will have the requisite five pieces plus one spare. Do not use wires much longer or shorter than specified or you may find that either they will not reach from the board to the panel properly, or you have a tangled mess of excess wire which will not fit in the space the module must occupy. The wires in a kit should be in three different colours but not necessarily exactly the colours shown in the photos here.

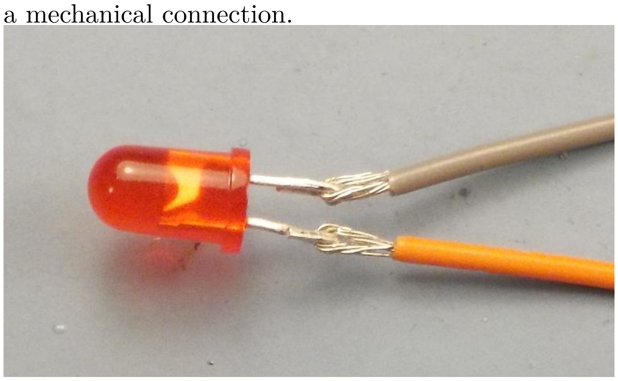
We recommend (and ship in the kits) pre-tinned stranded copper wire. If you use plain copper, you will need to tin it, and to do so carefully so as not to make the overall wire diameter too thick to fit through the holes on the PCB. Stranded wire should not be soldered with water-soluble (often misleadingly called "organic") flux because it is not realistically possible to remove all traces of it from between the strands. However, using some added non-water-soluble flux from a flux pen on these connections is a good idea to help make the connections with as little excess heat as possible, especially on the LED, which is heat-sensitive.

Prepare five pieces of hookup wire, each 3″ (7.5cm) long. Strip the ends, and tin them if necessary.



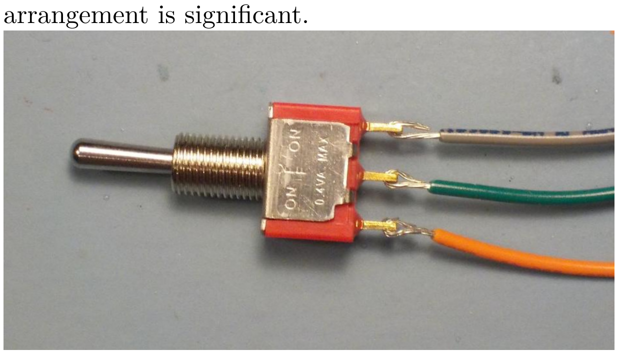
The LED, D16, is a polarized component. One of its two legs is the cathode, identified by a shorter leg, a flattened side on the plastic package, and a much larger metal terminal visible inside the plastic. Since the cathode will connect to the 0V plane, I suggest using a black, grey, or darker coloured wire for it.

Cut the LED legs to about $\frac{1}{2}''$ (13mm) length. Form them into hooks, trying to place as little stress on the place the wires enter the LED's plastic body as possible. Take two pieces of hookup wire, form one end of each into a hook, and link them with the hooked legs of the LED. Be sure you know which colour you have used for the cathode (grey is shown in the photo). Then pinch the hooks closed to form a mechanical connection.



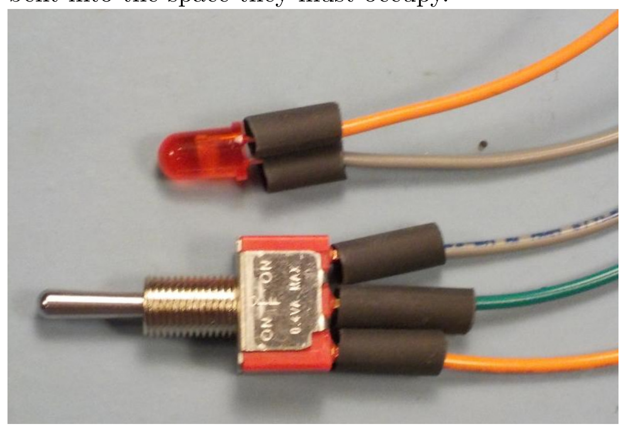
Carefully solder the LED connections, using as little excess heat as possible. If you are concerned about this point, you might use a clip-on heat sink on the LED legs to reduce the amount of heat conducted into the LED body. Dabbing the connections with a non-water-soluble flux pen before soldering may also be a good idea. After soldering, if your flux was not a no-clean type, you should remove the flux with solvent; it should be safe to dip the entire LED assembly at this point into any reasonable flux-cleaning solvent.

Take the remaining three pieces of hookup wire and form one end of each into a hook. Hook them through the holes on the terminals on the SPDT toggle switch SW1, then pinch them closed to form mechanical connections. There is no specific colour code recommended, but it is recommended to use three different colours for the three terminals because their arrangement is significant.



Solder the three switch terminals. The switch is less heat-sensitive than the LED, but some added no-clean flux is still recommended. The switch cannot safely be dipped in solvent, so if you need to clean these connections after soldering, you will need to use a brush.

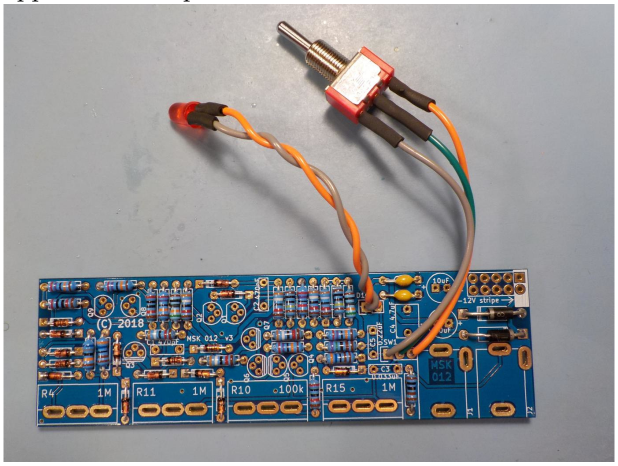
Cut five small pieces of heat-shrink tubing and place them over the connections on the LED and switch. North Coast kits come with significantly more heat-shrink tubing than is required here and you should have some left over; do not attempt to use it all or you will make the wires too stiff to be bent into the space they must occupy.



Carefully shrink the heat-shrink tubing. The best way to do this is with a hot-air gun made specifically for electronics assembly, on a relatively low temperature setting (cooler than would be appropriate for hot-air soldering). The stream of hot gasses above a candle flame will work, with a fair bit of caution. A hair dryer may possibly work. Heat from both sides to shrink the tubing evenly around the solder joints.

Do not have containers of, or brushes or rags contaminated with, flux cleaner or similar products open nearby while working with any open flame. Do not overheat the heat-shrink tubing so that it melts completely, chars, or catches fire; do not overheat other plastic parts so that they melt. After shrinking, the heat-shrink tubing will be very soft and fragile until it fully cools; do not touch it until then.

Put the free ends of the hookup wires into the holes for the D16 and SW1 footprints on the PCB. The cathode of D16 (as described above) should connect to pin 1 of the D16 footprint, denoted by a square pad and further away from the end of the board where the power header and Schottky diodes are. The switch has a groove or keyway on its bushing, to mate with a tab on the anti-rotation ring. The terminal nearest that groove should connect to pin 1 of the SW1 footprint, again denoted by a square solder pad and furthest away from the power end of the board, with the other two terminals in sequence from there. Twisting the wires together for a neater appearance is optional.



Solder the wires to the board.

Transistors

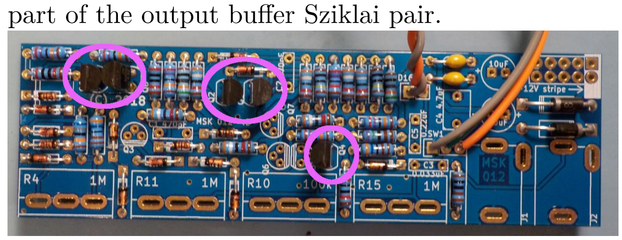
The MSK 012 contains two different types of transistors packaged in TO-92 packages. Each looks like a little black pill of epoxy plastic with one flat side and three metal legs; they can be distinguished by etched or printed numbers on the flat side, and it is important to sort them carefully and install only the proper component type in each footprint.

There is not enough space on the boards to print a part number for every TO-92 component, but there are two different silkscreen symbols used to help with recognition. The SS8550D or PN200A transistors are shown on the board with extra silkscreen lines along the flat edge, as in the left photo. The 2N5088 transistors are shown by a plain outline without extra lines, as in the right photo.

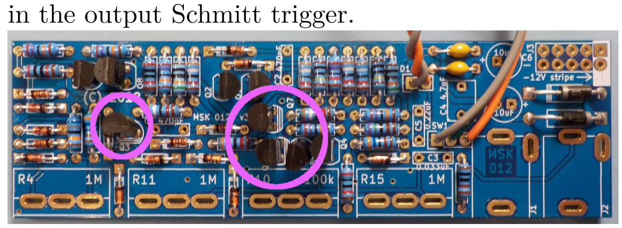
All TO-92 components in this project are polarized and must be installed in the correct orientation to work; that orientation is shown by the silkscreen symbols. Install each component so that its flat side points in the same direction as the flat side shown on the silkscreen. The three legs of the component must be carefully bent into the same triangular pattern (left and right forward, middle backward) as the holes on the board, and then the component pressed into place. There should be a gap of about three millimetres between the board and the component body; do not attempt to seat the component flush on the board because of the risk of breaking off the legs where they enter the body.

The solder pads for these components are smaller and closer together than for any other through-hole components in the project, and the components themselves tend to be relatively heat-sensitive. Solder them carefully, avoiding creating any solder bridges between adjacent pads. Do not use excessive time and heat trying to get the solder to flow through the board and fillet on both sides, especially not on pads connected to the ground plane; two-sided fillets are nice if they happen naturally, but it is good enough for solder to completely cover the pad on one side.

Install the five 2N5088 NPN transistors Q1, Q2, Q4, Q8, and Q9. Most of these act as switches in the input Schmitt trigger and attack flip-flop; Q4 forms part of the output buffer Sziklai pair.

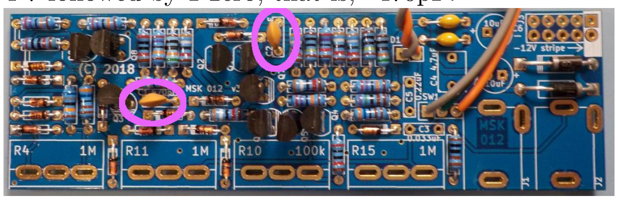


Install the four SS8550D or PN200A PNP transistors Q3, Q5, Q6, and Q7. The first, Q3, sinks current away from the envelope capacitor during the release and sustain phases; Q5 forms part of the output buffer Sziklai pair; and Q6 and Q7 act as switches in the output Schmitt trigger.

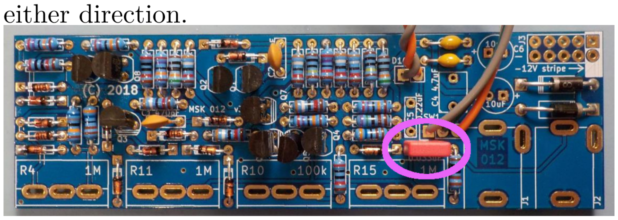


Capacitors

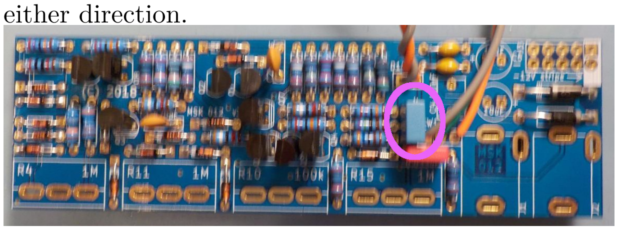
Install the two 470pF ceramic capacitors C1 and C2. These capacitors form pulses from the switching transitions of the input and output Schmitt triggers. They are not polarized and may be installed in either direction. They will probably be marked "471," which indicates the number of picofarads using something like the resistor code: significant digits 4 7 followed by 1 zero, that is, "470pF."



Install the 0.033μF film capacitor C3. This capacitor stores the envelope voltage in the fastest speed range, also taking a small part in the other two speed ranges. It is not polarized and may be installed in either direction.



Install the 0.22μF film capacitor C5. This capacitor stores the envelope voltage in the middle speed range. It is not polarized and may be installed in either direction.



Install the 4.7μF film capacitor C4. This capacitor stores the envelope voltage in the slow speed range. It is not polarized and may be installed in either direction.

Install the two 10μF electrolytic capacitors C6 and C7. These filter lower-frequency interference on the power lines. They are polarized components, and may explode if connected backwards. As such, there are multiple clues to help you install them in the right direction. The negative leg of each capacitor will be marked in some way, usually with a printed stripe and minus signs on the plastic wrapping of the capacitor body. The negative leg of the capacitor will usually also be shorter, though that is less reliable than the body markings. On the PCB, the positive and negative pads are marked with positive and negative signs in the silkscreen, and the solder pads themselves are round for negative and square for positive.

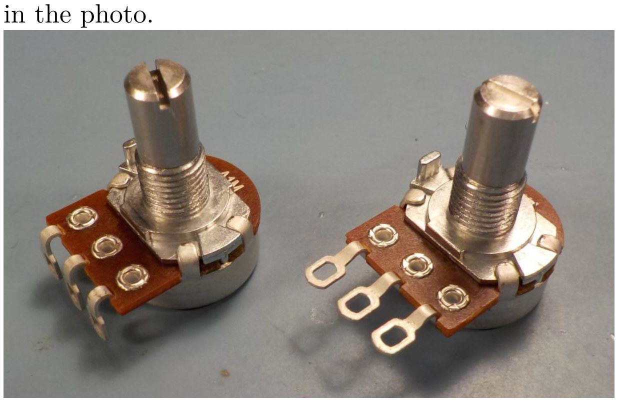
Power header

Install the 10-pin dual-row Eurorack power header J3. It is not polarized in the horizontal plane. However, if it has shorter legs on one side, then those are the ones that should go through the PCB (leaving the longer legs sticking up to mate with the connector on the power cable), and if it has tin plating on one end of the pins and gold on the other, then the tin side should be the one soldered through the board. Secure the header carefully to the board, possibly with tape, before soldering it. It is easy to accidentally solder it at an angle, which is a difficult error to fix and may cause trouble when you later attach the power cable.

Note that Eurorack power connections are polarized even if the connectors are not. The cables are usually grey ribbon type with a red stripe along one side indicating pin 1, which carries −12V power. For most modules including the MSK 012, the red stripe should be at the *bottom* when the module is mounted vertically in a case. On the MSK 012, the correct location of the −12V supply is also marked with the text "−12V stripe," arrows, and a white area on both sides of the PCB silkscreen. This module is also protected (by the Schottky diodes you just installed) from damage in case of a reversed power connection; if you connect the power backwards and nothing else is wrong, then the module will not power up but will be fine once you connect the power correctly. However, many other modules are not so protected, and it is dangerous to get into the habit of depending on protection diodes. Destroying a module by connecting power backwards is almost a rite of passage for Eurorack users.

Panel components

Sourcing good-quality panel potentiometers with all the right features for a given project is difficult, and for the MSK 012, the closest I was able to come was to find a type with right-angle bent terminals, where the physical design of the module requires straight PCB terminals. So, if you are using the potentiometers supplied in a North Coast kit or close equivalents to them, your potentiometers will probably resemble the one at left in the photo, with its terminals bent downward. Carefully bend the terminals with pliers so that they lie in roughly the same plane as the fibreglass panel of the potentiometer, as shown at right in the photo.

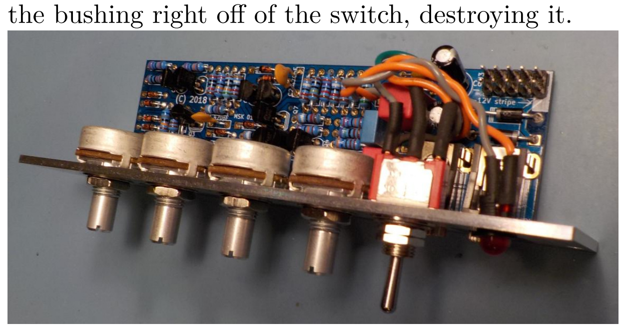


Consult the exploded assembly diagram on page 34 for details of how the following parts fit together.

Install the four potentiometers (R4, R10, R11, and R15) in the panel with their supplied hardware, but only *very loosely* at this point; they should be just tight enough to keep the anti-rotation tabs in the corresponding holes, with enough play for subsequent steps to position them exactly. Take careful note of which one of these is R10, the 100kΩ unit; it must go in the hole labelled “S,” third from the top end of the panel. Snap the LED mounting clip (small black plastic object) into the panel hole above the words “NORTH COAST.” Install the two panel jack sockets (J1 and J2) in their respective holes.

Fit the PCB onto the protruding legs of the potentiometers and jack sockets. It may be necessary to adjust the bends a little to get all legs to fit. Try to get the PCB at right angles to the panel as nearly as possible. Then tighten the nuts on the potentiometers, and solder all these components.

Push the switch and LED into their corresponding holes in the panel, with the LED fitting inside its mounting clip. The keyway on the switch must face toward the top end of the panel so that the tabs on the anti-rotation ring can fit into the switch keyway and the small panel hole. The switch will probably come with two nuts. When tightening them, do not overtighten; overenthusiastic use of a wrench can pull the bushing right off of the switch, destroying it.



Attach the knobs to the potentiometers. Twist each shaft to its limits in each direction to ascertain how the slot in the shaft corresponds to where you want the knob pointer, then slide the knob onto the shaft in the correct orientation and tighten the setscrew with a small flat screwdriver. Do not fasten the knobs pushed all the way down onto the shafts, or they will rub against the bushings and be hard to turn; instead, back them off about a millimetre from the bottom.

There is a rectangular white area on the back of the board reserved for adding a serial number, signature, quality control marking, or similar. Use a fine-tipped permanent marker to write whatever you want there.

Your module is complete.


MSK 012
transistor
envelope
A
D
S
R

Testing

The MSK 012 needs no trimming or adjustment; unless there are defective components or build errors, it should work as soon as it's built. But here are some notes on testing and troubleshooting.

Short-circuit test

With no power applied to the module, check for short circuits between the three power connections on the 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 definitely 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 it will probably read infinite, but at least 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.

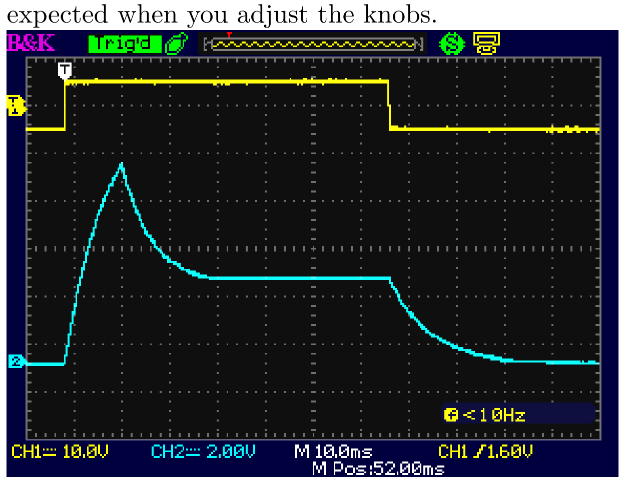
Optional: Although we test all cables before we sell them, bad cables have been known to exist, so it might be worth plugging the Eurorack power cable into the module and repeating these continuity tests across the cable's corresponding contacts (using bits of narrow-gauge wire to get into the contacts on the cable if necessary, or probing the pins of the power connector on the back side of the circuit board) to make sure there are no shorts in the cable crimping. Doing this test *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.

Envelope shape

If you have access to an oscilloscope and pulse-wave LFO: apply power to the module, feed a pulse wave of about 10Hz into the input, set the range switch to the middle (fastest) position, and look at the output with the oscilloscope. It should look like some sort of reasonable ADSR envelope, and it should change as expected when you adjust the knobs.



It is possible, and normal, for the "zero" voltage between pulses to appear a fraction of a volt less than zero, especially when there is no load on the output except a high-impedance scope probe. It is also possible that the maximum sustain level (S knob turned fully clockwise) may be a little higher than the attack peak, resulting in an unusual-looking envelope shape.

Troubleshooting

Some common problems to look for if something goes wrong:

- Solder bridges, especially between closely-spaced joints on TO-92 transistors.
- Other solder problems, such as solder not sticking to the board (can be caused by too-high iron temperatures, or the wrong flux).
- Polarized components installed backwards.
- Components swapped around, such as the 100kΩ panel pot installed in the wrong one of the four spots (the other three requiring similar-looking 1MΩ pots), or swapping the 27kΩ resistor with one of its 270kΩ cousins.
- User error (misunderstanding the intended operation of the module), such as missing the sustain phase when the sustain level is set fully counterclockwise, or using short triggers on the input with slow envelope settings.

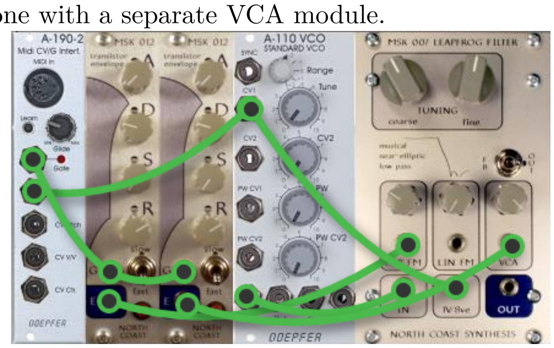
During development an issue was noted between this module and the Befaco Rampage. The Rampage's gate output apparently never went quite to zero volts, remaining at a high enough positive voltage in between gates that the MSK 012 never recognized the end of the gate. The final released design of the MSK 012 has had its gate thresholds raised to help prevent such issues, and that seemed to fix the problem; but in general, do be aware that it cannot work miracles in the case of faulty input, and the transistor Schmitt trigger at the input does require some nonzero amount of current-sinking (not just an open circuit) in the low state. The Rampage's outputs have been known to cause problems with other modules before, because in the low state they connect to ground only through an LED, presenting a high impedance at voltages below about one volt.*

*https://www.muffwiggler.com/forum/viewtopic.php?t=187044

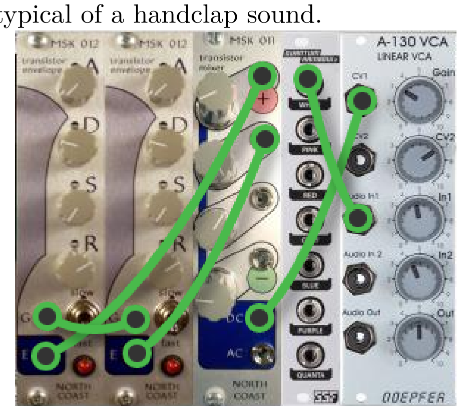
Patch ideas

Here's the most traditional use for any ADSR generator: applying an amplitude envelope to oscillator output with a VCA.

For a more deluxe version of the traditional subtractive patch, use separate ADSR envelopes (both driven by the gate from the MIDI interface) to control amplitude envelope and cutoff on the filter. This example uses the built-in VCA of the North Coast Leapfrog filter, but the same thing could easily be done with a separate VCA module.



This is a sort of quick and dirty handclap patch; maybe also a finger-snap, brushed drum, or a few other kinds of percussion, depending on the envelope settings. The key to all these kinds of sounds is getting the right envelope shape, and it may be more complicated than just a basic ADSR. Here two MSK 012 modules have their outputs added by a DC mixer, with the result controlling a VCA with white noise as the audio. Normally the two ADSR envelopes would be set on different time ranges, one "fast" and the other "faster." One has minimum attack and a little bit of decay; the other has a little bit of attack and minimum decay; both have zero sustain. The sum of the two envelopes gives a double peak typical of a handclap sound.



Chaining two envelopes, the first one set for slow attack and release and maximum sustain level, creates a gate delay. The second envelope won't start until the attack of the first reaches about 2V, and won't end until the first envelope's decay drops to about 1V. If you want a well-behaved gate voltage for something else, the second envelope can be set to very fast range, minimum attack and release, maximum sustain level, to get a nice square-sided 8V pulse output. But if you just want a delayed envelope, the second ADSR generator can be set to create the desired shape.

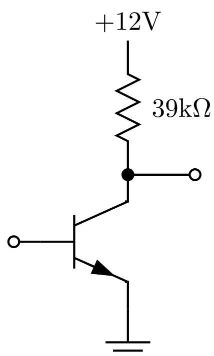
Because of the Schmitt trigger on the gate input, which activates on voltage levels without requiring a sharp edge, the MSK 012 can be used to condition signals for other modules with pickier inputs. Here, the Z3000's built-in frequency counter/tuner requires sharp-edged signals and cannot get a reading on the smooth waveform of the Hikari Sine. Feeding the sine wave through an MSK 012 (fastest speed range, minimum attack and release times, maximum sustain level) converts it into a 0-8V pulse wave, which the Z3000 can reliably measure.

Circuit explanation

The design concept for the MSK 012 was to explore generating a classic ADSR envelope with logic as simple as possible, and in particular, a minimum number of transistors. Many ADSR envelope generators would use op amps for computing the necessary time-varying voltages, or even just throw in a microcontroller and do everything in software, and either of those expedients might keep the *parts* count down by using parts containing dozens or thousands of transistors; but how far can we get with a very small *transistor* count?

Inverters and multivibrators

One simple circuit is used repeatedly in this module in varying forms. It looks something like this.

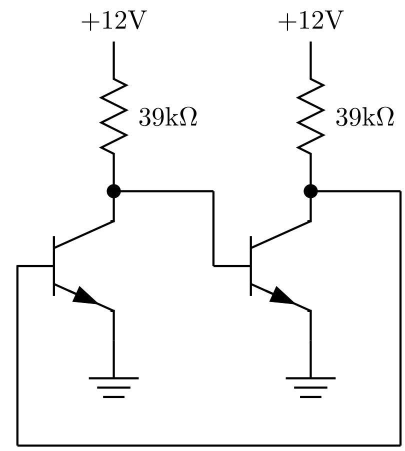


This circuit is the simplest possible logic gate in a system of *resistor-transistor logic* (RTL) where logic "1" is represented by the power supply voltage through a few tens of kiloohms of resistance, and logic "0" is represented by a low impedance with a low voltage drop to ground.

Suppose we apply logic "1" to the input. Current flows into the base of the transistor (a few hundred microamps), the transistor switches on, in saturation mode because its base current multiplied by its gain works out to far more than the current available through the resistor, and so the output (which is just the collector of the transistor) goes to the saturation voltage of the transistor, which will be something like 0.3V. The transistor is capable of absorbing a signficant amount of additional current through its collector without much increasing its voltage drop, so the output of the circuit has a low impedance. The output is thus a perfect RTL "0."

Now suppose instead we apply a logic "0" to the input. The base of the transistor is held at about 0.3V, which is below its turn-on voltage (typically about 0.7V), so the transistor does not pass any significant current, and the power supply voltage is exposed to the circuit output through the 39kΩ resistor, therefore constituting an RTL "1." In either case, the circuit fed with one logic state at the input generates the other at the output; it is an RTL *inverter*.

Now consider two inverters connected in a loop.



If the input to the first inverter is logic "0," its output will be logic "1," which causes the output of the second inverter to be logic "0," feeding back into and reinforcing the existing state of the first inverter's input. That is a stable state for the system. If the input to the first inverter were "1," that is a second distinct stable state: first inverter's output goes to "0," second inverter's output goes to "1," and that reinforces the original "1." Once it is in one of these states the circuit will reliably remain there; if it is temporarily somewhere else (for instance, during startup) it will quickly fall into one of the two stable states and then stay fixed. A circuit of this type is called a *bistable multivibrator*; bistable because it has two stable states, multivibrator from an archaic naming convention for switching circuits coined in the early 20th Century.

In order to make this circuit really useful, we need to modify it in some way (usually by adding some more resistors) to allow an external input to temporarily overcome the feedback loop and force the circuit into a chosen state. Then when the external input is no longer strong enough to force it into one state, the multivibrator will hold its previously-set state until something else changes to switch it to the other state. Its basic usefulness, then, is as a one-bit memory that retains a logic state. The MSK 012 uses three such one-bit memories to keep track of the envelope cycle.

Input Schmitt trigger

The gate control voltage coming into the module first goes through the input Schmitt trigger, shown in Figure 1. This section conditions it into a well-behaved logic level; although the gate voltage might be slowly varying and anywhere between the power voltages (±12V), we need a straightforward logic “1” or “0” with sharp transitions to drive the rest of the module.

The input Schmitt trigger is a basic multivibrator as described in the previous section, made up of two RTL inverters, with a resistor network driving the input of the first one. For sufficiently high input voltages (that is, over about 2V) the input forces the multivibrator into the logic “1” state. For sufficiently low input voltages (below about 1V), the multivibrator is forced into the “0” state. In between those thresholds, the voltage divider formed by R8 and R12 would have an output voltage sufficiently near Q1’s switching voltage that the feedback through R1 is sufficient to keep the multivibrator in its current state, whichever that is. The name *Schmitt trigger* refers to this general kind of switching circuit where there are two input thresholds and after switching, the input must move some amount to hit the other one before it can switch again.

The current state of the input Schmitt trigger appears as the RTL signal labelled B on the schematic, but there is also an edge detector formed by C1, R9, and D3. When the gate input goes above 2V, causing B to become high, current flows through C1 and D3 to create a pulse of current into the signal labelled B+. That happens only on the low-to-high transition, for a few microseconds. The current charges the capacitor, so the pulse dies away, and then when B switches in the opposite direction, the diode blocks the current so there is no negative pulse, and the capacitor discharges through R9. The pulse output B+ is used elsewhere in the module to switch the attack flip-flop.

Note D6, from ground to the base of Q1. This diode is meant to protect the transistor from breakdown of its base-emitter junction in the case of a large negative voltage at the module input. At the rated worst case input of −12V, the network of R1, R8, and R12 could bring the transistor base to about −3.3V, which exceeds the 2N5088’s rated limit of −3.0V. The diode limits the base voltage to no lower than −0.6V.

Output Schmitt trigger

There is another Schmitt trigger connected to the module output and shown in Figure 2.

This one is used to detect the end of the attack phase, and it is built with PNP transistors (essentially, all the polarities reversed from those of the input Schmitt trigger) in order to better handle switching at a voltage near the positive supply. The resistor network at the input of this section is similar to that of the input Schmitt trigger, with the addition of R22, which drains some current into the negative supply to prevent the current through R16 and R19 from artificially raising the module’s output voltage.

When the module’s output reaches about 8V, marking the end of the attack, this Schmitt trigger switches into the “1” state. The plain state of the output Schmitt trigger is not used elsewhere in the module, but there is another edge detector (C2, R25, D14) which generates a pulse into the signal marked A+ when the end-of-attack event happens. The Schmitt trigger will turn off again (ready to detect another attack) after the output voltage drops a couple of volts.

Attack flip-flop

Both input and output Schmitt triggers feed into the attack flip-flip, which keeps track of whether the module is currently in the attack phase, as shown in Figure 3.

This is a basic bistable multivibrator with 270kΩ resistors between the stages, weakening the signals so that although they remain strong enough to hold the state without external input, they can be overridden by pulses from the Schmitt triggers. At the start of the external gate signal, there is a pulse on B+ from the input Schmitt trigger, setting the flip-flop into the “1” state (start of attack). When the output voltage reaches about 8V, there is a pulse on A+ from the output Schmitt trigger, resetting the flip-flop to the “0” state. The current state is available as the signal

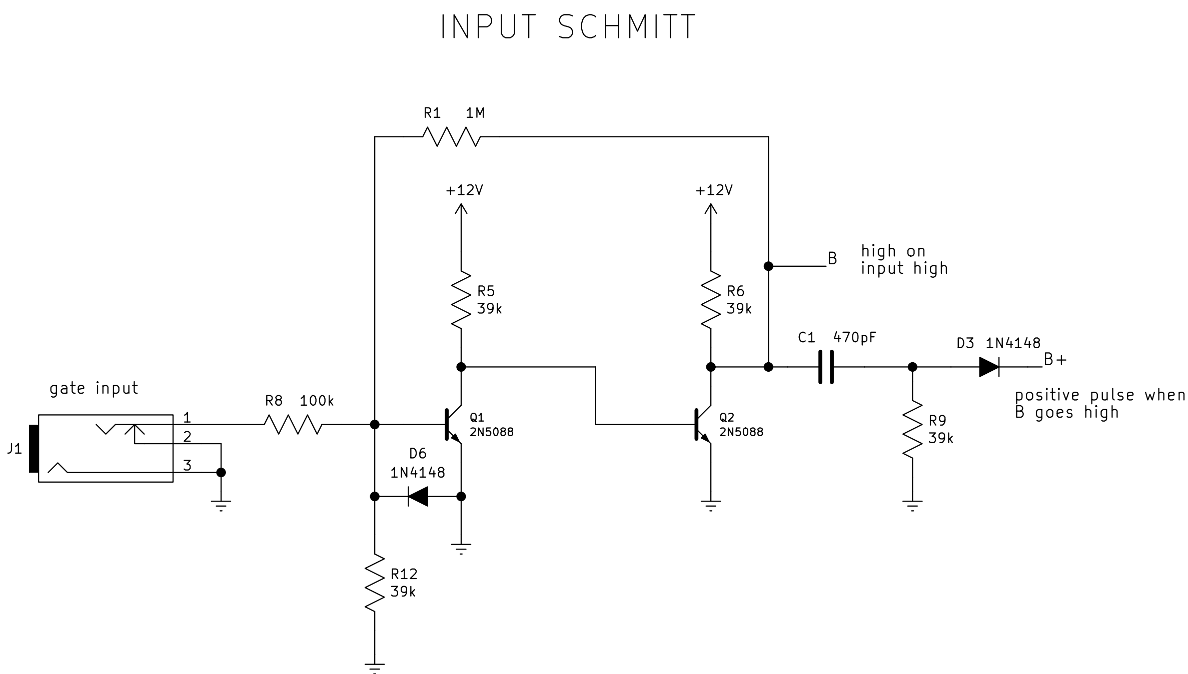


Figure 1: Input Schmitt trigger

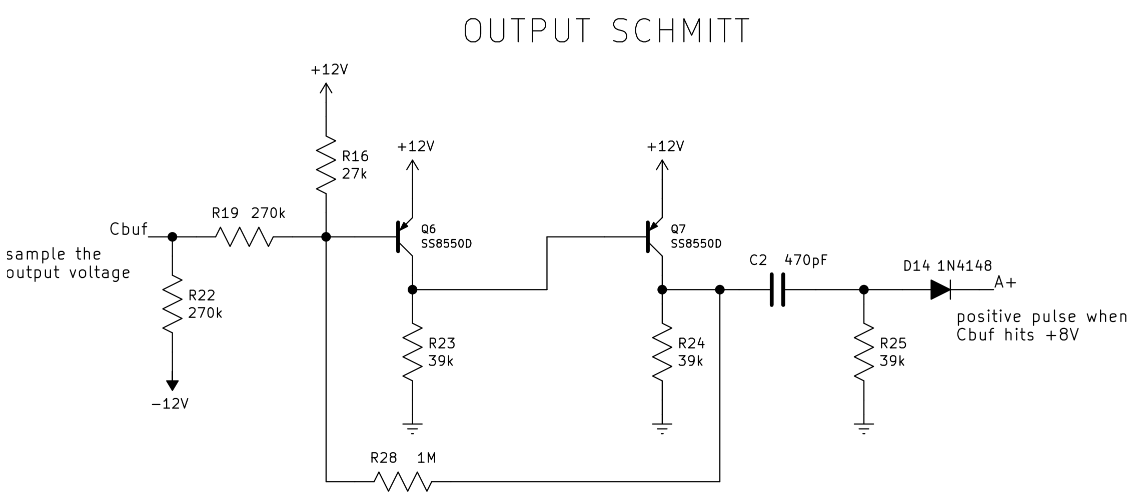


Figure 2: Output Schmitt trigger

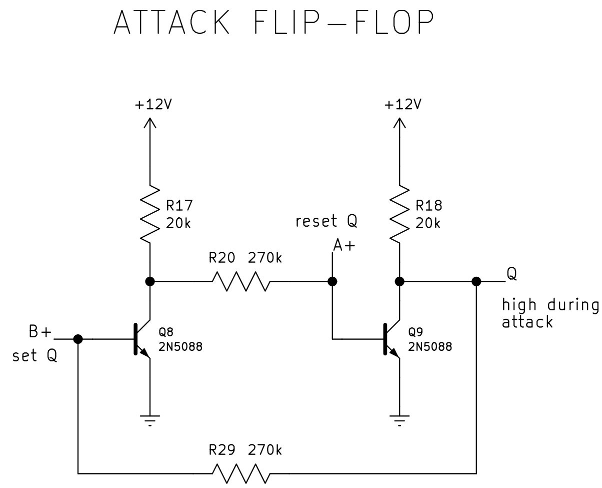


Figure 3: Attack flip-flop

marked Q.

Note that the collector resistors (R17 and R18) in this flip-flop are a little smaller than elsewhere in the module, at 20kΩ instead of 39kΩ. The value of R18 is reduced in order to be sure it can provide enough current to the ADSR driver, which consumes this signal; and the value of R17 is made to match it so that the overall static current consumption of this section will not change with the state of the flip-flop, reducing the possibility for audio-rate spikes to be generated in the power system.

ADSR driver

The section called the ADSR driver (Figure 4) does most of the work of shaping the envelope.

First, note the connection labelled C, which goes to the capacitor in the output buffer section. This section's output is in the form of a bidirectional current that charges or discharges the capacitor at different rates. Its input is two logic signals: Q from the attack flip-flop, which is high from the start of the gate until the module output reaches about 8V, and B from the input Schmitt trigger, which is high throughout the time the input gate is high.

The ADSR driver's operation is best understood by looking at one state at a time. First, suppose the input gate is low and has been for a long time. Both B and Q are low, and the envelope capacitor is at or near 0V. With B low, current from R2, R3, and R4 flows through D1 into B, and the anode of D1 remains low enough that D2, D4, and D5 are reverse-biased or at least not sufficiently forward-biased to overcome their forward voltages and start conducting. Current flows from +12V to −12V through R7, some part of R10, D7, and R14, and doing the math, this puts the base of Q3 a little above ground, which means its emitter will be as well, and then D4, D5, and D8 are reverse-biased, blocking D11 out of the circuit. The only remaining path for current to or from the output is through R15 and D13 to the near-ground B signal; that keeps the envelope capacitor near ground. Technically, its minimum voltage will be about +1.0V, that being one diode drop for D13 and one saturated transistor voltage for Q2 in the input Schmitt trigger, but this is compensated by an offset in the output buffer to bring the actual output of the module near zero.

Now, suppose the input gate goes high. The B signal goes high, which triggers a pulse on the B+ signal, setting the attack flip-flop, and so Q goes high as well. With B high, D13 is reverse-biased, taking R15 out of the circuit. With Q high, the base of Q3 is kept high, which keeps D4, D5, and D8 reverse-biased and takes R11 out of the circuit. Then the envelope capacitor is exposed to current from the resistor network of R2, R3, and R4 through D2. The capacitor starts to charge at a rate controlled by the setting of R4; this is the attack phase of the envelope.

When the envelope capacitor has charged enough for the output of the module to exceed about 8V, the output Schmitt trigger fires, generating a pulse on A+ which switches the attack flip-flop and causes Q to go low. Then the components connected to the base of Q3 come into play.

The potentiometer R10 is adjusted to the sustain voltage, with a proper offset for diode and transistor drops elsewhere in the circuit. Its low end is kept at about −1.2V by forward-biased D9 and D10, serving as primitive voltage regulators with current through R13 keeping them forward-biased. The high end of R10 is separated from the positive supply by R7, keeping the maximum sustain level at about the 8V maximum of the attack phase. (From breadboard experiments: setting a sustain voltage higher than the top of the attack doesn't break anything, but results in strange and probably undesirable envelope shapes.) The sustain voltage goes through D7 (which serves to block it from loading the Q signal when Q is high) to the base of Q3. With Q blocked by reverse-biased D11 and D12, Q3 functions as an emitter follower, holding its emitter at the sustain level.

Current flows from the envelope capacitor through R11 and D8 into Q3. The envelope capacitor discharges in a nice exponential-decay curve at a rate

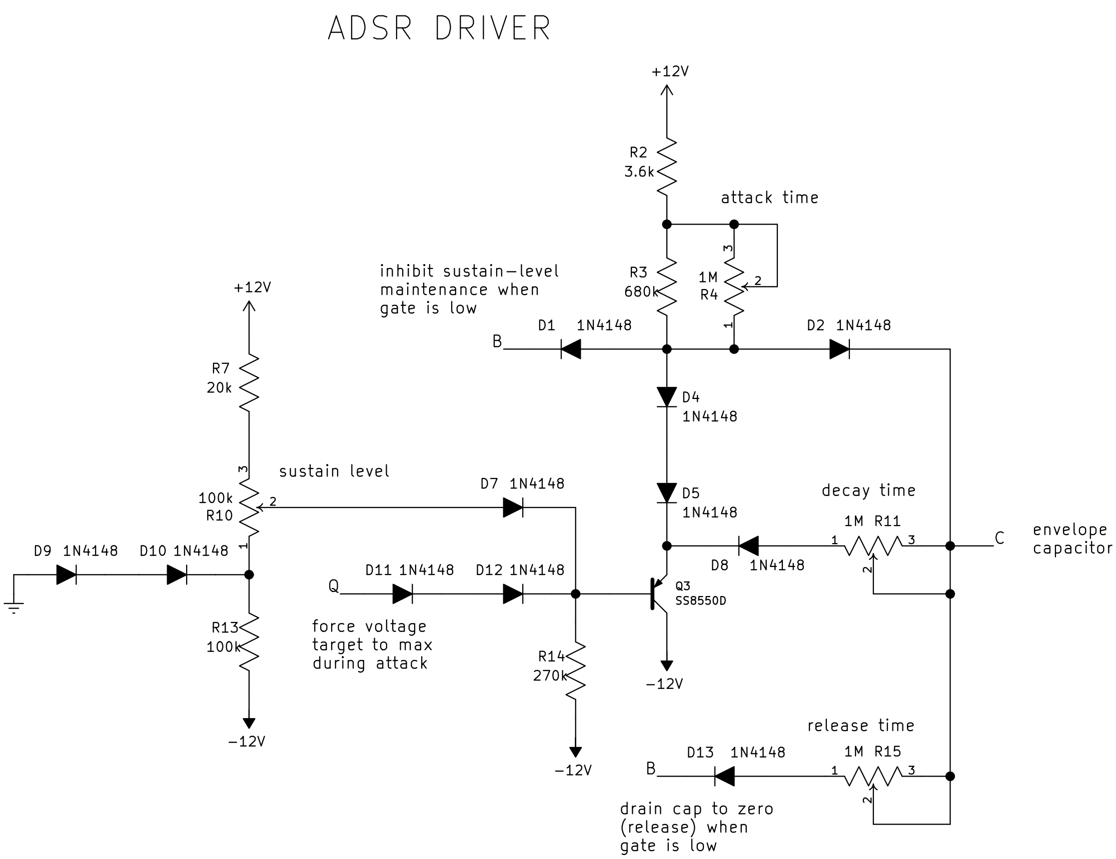


Figure 4: ADSR driver

set by R11 until it hits the sustain voltage. Current from the attack-rate network is bypassed through D4 and D5 to also drain into Q3; note that D4 and D5 maintain a voltage of two diode drops, whereas the voltage needed across D2, R11, and D8 to turn on the diodes will inevitably be more than that due to the voltage drop across R11 during decay, so current from the attack network does not go into the capacitor to gum things up there; only the decay potentiometer is in play during the decay phase.

However, suppose after the decay phase the gate remains high for a long time. There may be a small amount of leakage from the envelope capacitor, causing its voltage to droop. *Then*, as the voltage on the envelope capacitor drops below the intended sustain level, D2 will start to conduct, stealing current from D4 and D5. Note that the anode of D4 is kept two diode drops above the emitter of Q3, so this effect will kick in to prevent the capacitor going below one diode drop from the emitter of Q3; that is, exactly the sustain level. Current through R11 drops the voltage to the sustain level during the decay, but current from the attack circuit prevents it from dropping any further in case of a long sustain, and the current available corresponding to the attack speed is always enough to compensate for current lost from the capacitor elsewhere in the module.

When the gate input eventually goes low again, we are back where we started. The B signal goes low, stealing current from the attack network and preventing it from holding the capacitor up at the sustain level. The decay circuit is blocked off by the emitter of Q3 being at the sustain level, thus unable to increase the capacitor voltage or decrease it any further than that level. The only remaining path for current to or from the envelope capacitor is through D13 and R15, draining it at the release rate until it reaches its minimum level.

Output buffer

The output buffer, shown in Figure 5, integrates the current output of the ADSR driver into a voltage and buffers it, both for the output Schmitt trigger and the output of the whole module.

The "envelope capacitor" may really be two; it consists of the 0.033μF capacitor C3, possibly in parallel with C4 or C5. The three-position switch allows choosing to use one of those, or neither of them, for total nominal capacitance of 0.033μF, 0.253μF, or 4.733μF (though tolerances mean that the larger numbers are not precise to the stated number of digits).

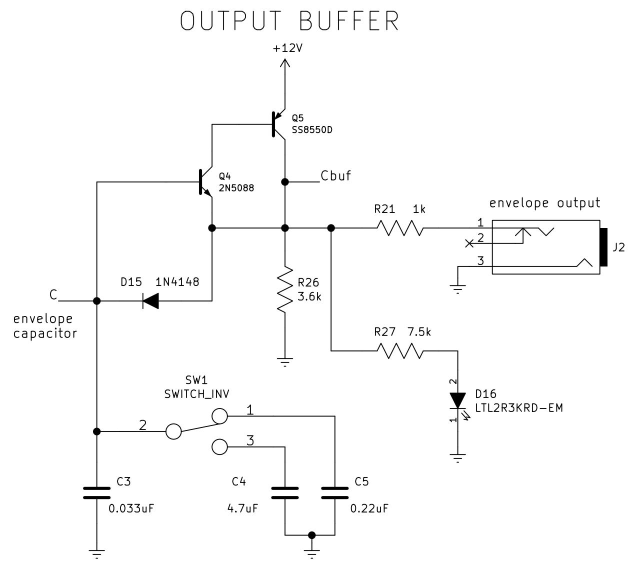


Figure 5: Output buffer

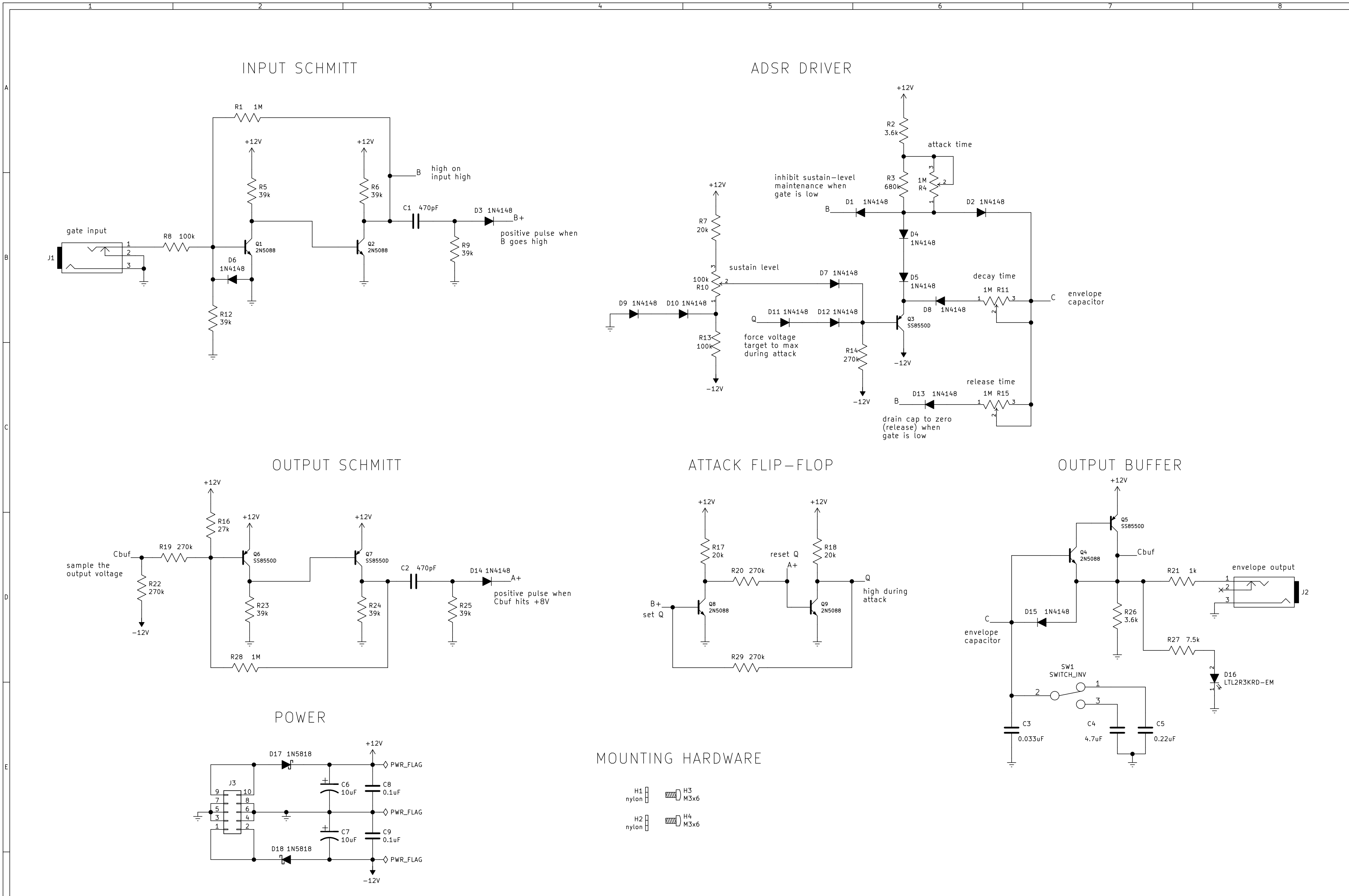
Transistors Q4 and Q5 (one PNP, one NPN) form a *Sziklai pair*, which functions much like a single NPN transistor of very high gain (equal to the gain of its two transistors multiplied together). The extra gain is needed because even with the high-gain transistors I specify, a single transistor would draw too much current from the envelope capacitor and disrupt the envelope shape at low speeds, when biased to produce the desired amount of output current. The diode D15 protects Q4 from reverse bias, much like the similar diode at the module input, in the case of someone shorting the module output into the positive power supply or similar while the output is being driven low. The Sziklai pair combined with R26 is an emitter follower that simply generates the desired output voltage, removing the offset from the capacitor voltage.

From the emitter of the Sziklai pair, the output voltage goes to the output Schmitt trigger to detect the end of the attack phase; through a current-limiting resistor R21 to the output jack of the module; and through another current-limiting resistor R27 to supply the LED D16. Note the high value of R27, which is not an error; the specified LED is a high-efficiency type being operated significantly below its maximum rated current so as not to be annoyingly bright. The target is about 825μA when the envelope hits 8V.

Mechanical drawings

On the following pages you will find:

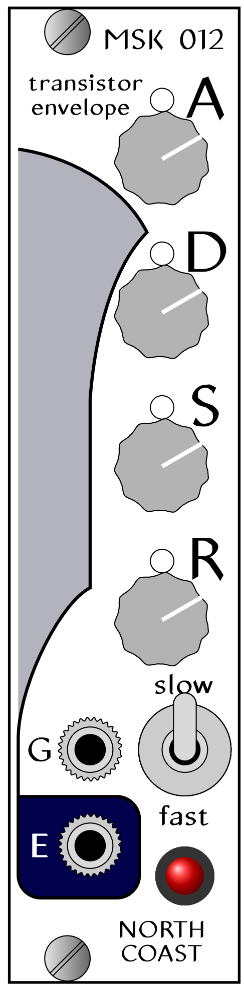
- the schematic diagram for the module;
- a mock-up of what the completed module looks like from the front panel;
- the top-side silk screen art showing component placement;
- the bottom-side silk screen art showing component placement (*note this drawing is mirrored, and shows what you actually see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

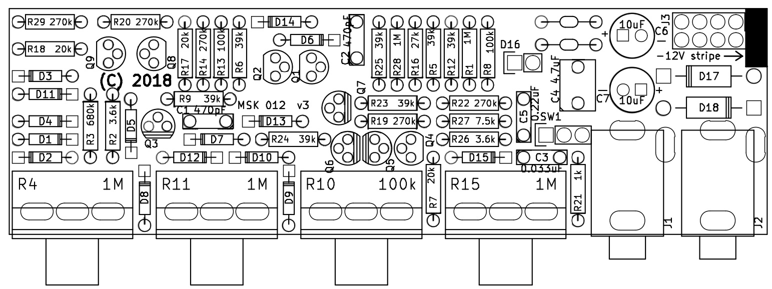


MSK 012 Transistor ADSR

$Id: qadsr.sch 6210 2018-09-02 00:38:21Z mskala $

1/1


MSK 012
transistor envelope
A
D
S
R
slow
G
fast
E
NORTH COAST



MSK 012 Transistor ADSR

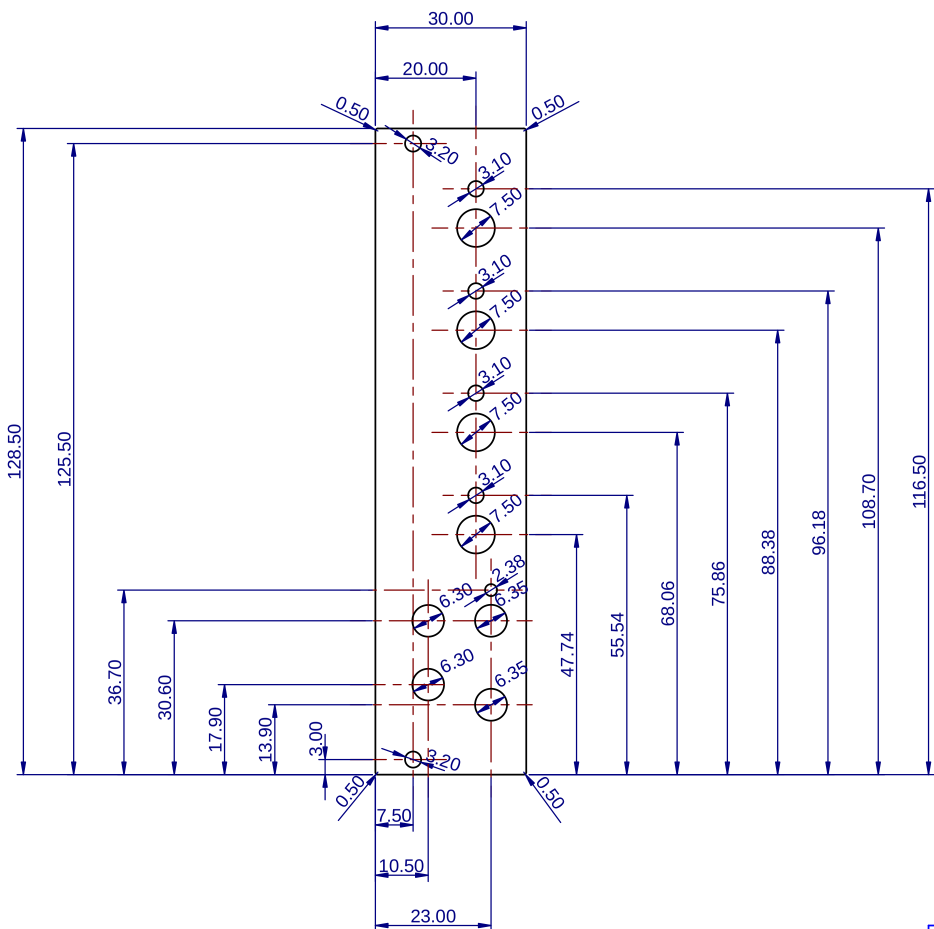
$Id: qadsr.kicad_pcb 6073 2018-06-06 20:26:37Z mskala $

1/2

MSK 012 Transistor ADSR

$Id: qadsr.kicad_pcb 6073 2018-06-06 20:26:37Z mskala $

1/2

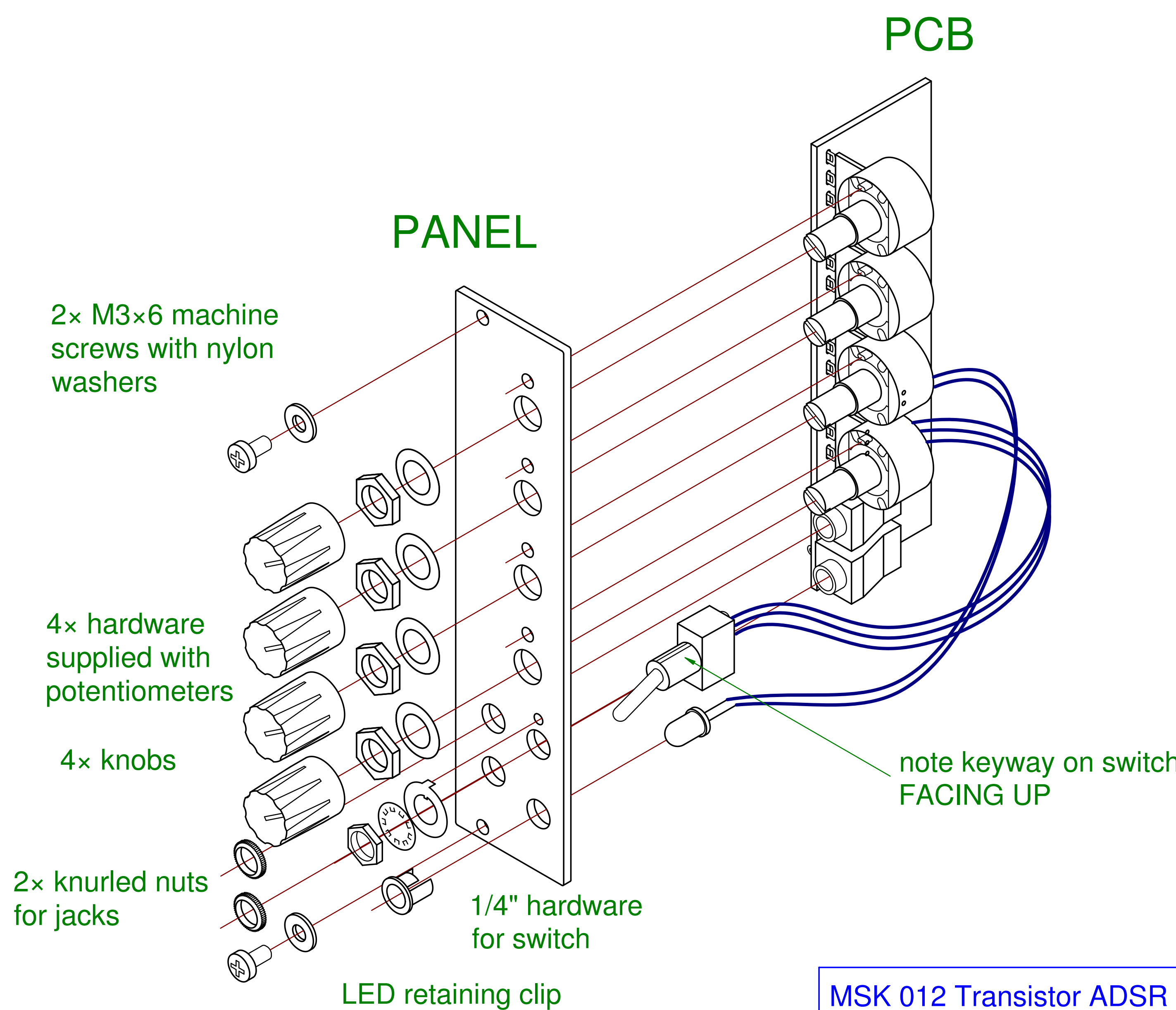


NOTES

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

2: Material is aluminum, 2.00mm thick. Overall dimensions are 128.50mm by 30.00mm, per Doepfer spec for a 6HP Eurorack synthesizer module.

MSK 012 Transistor ADSR panel mechanical	
$Id: panel-mechanical.dxf 5947 2018-03-07 01:22:55Z mskala $	1/1



MSK 012 Transistor ADSR assembly diagram

$Id: panel-mechanical.dxf 5947 2018-03-07 01:22:55Z mskala $ | 1/1