BMC017. 2LFOSH
Last updated 12-3-2013

I. Features

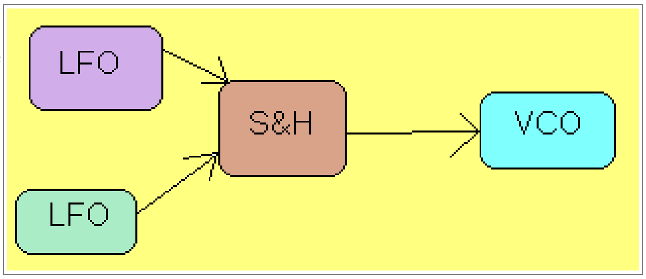
The 2LFOSH module is a combination of three different modules on one board, designed to be easy to be easy to build and have simple, bare-bones features, making it ideal for people new to building DIY synthesizers, and those with small systems who need multiple uses from each module. It requires no calibration.

The module is composed of two low frequency oscillators (LFOs) and a sample and hold (SH) circuit. The LFOs each have a switch to change between LFO ranges and audio ranges. Technically, the correct term is "manually controlled oscillator" when it is operating at audio range, but LFO is still used commonly. Each LFO has just two outputs, a square wave and a triangle wave. There is also a mixed triangle wave output which is the inverted sum of the two triangle wave outputs. Each LFO has an LED indicator, to visually show frequency.

When design started, the original focus was on an easy to build sample and hold module. A sample and hold has two inputs, one input for the voltage to be sampled and one input for a trigger to tell it when to sample the input. It is common to see an LFO used to trigger the samples, and a noise generator used as the voltage input. Often the noise generator is built into the same module as the sample and hold. I decided to build a second LFO into the module instead of a noise source because in my experience, it would be more useful when not using it as a voltage source for the SH than a noise source; but it would still provide voltages that were unpredictable enough for the human ear to pick up on when run at audio range with the SH trigger run at a lower frequency.

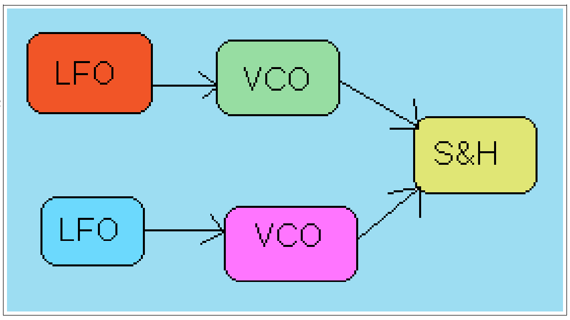
Sample Patches

1.This patch consists of one LFO triggering the SH while the other provides voltage for it to sample. The output of the SH is modulating the frequency of a VCO.



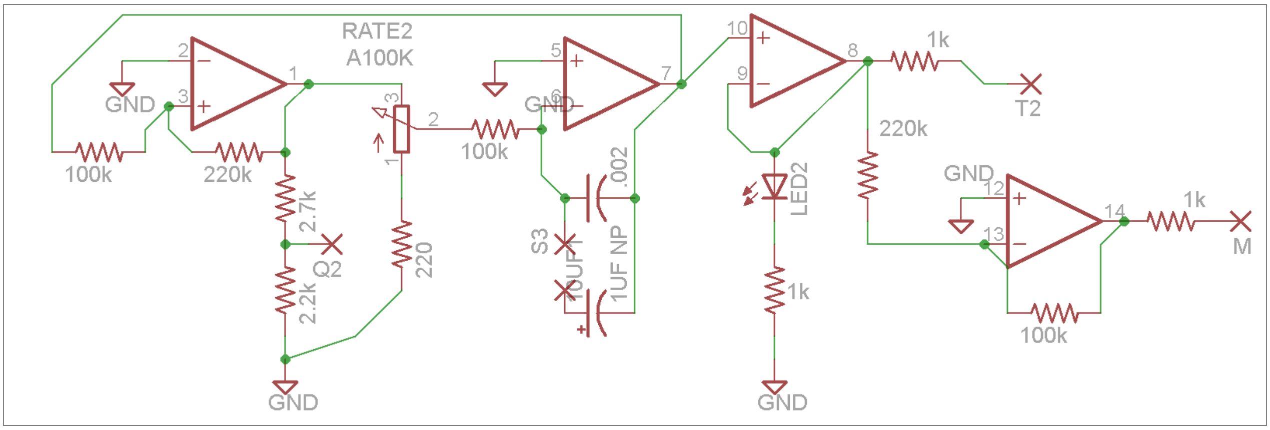
Listen here

2.This patch consists of each LFO modulating the frequency of a VCO. One VCO is triggering the SH while the other provides voltage for it to sample. We are listening to the output of the SH.



Listen here

II Schematics

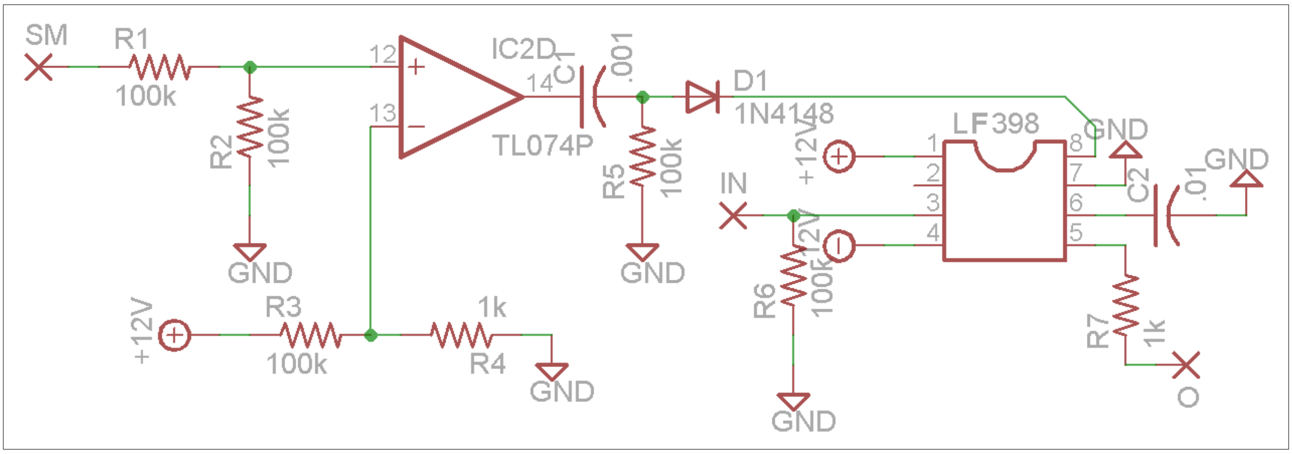


This schematic shows one of the LFOs and the mixer, not pictured is the 220K mixing resistor coming from the other LFO. This LFO uses the time honored Schmitt Trigger/Integrator topology.

The op-amp on the far left is an inverting schmitt trigger, whenever the voltage input to it from the 100K resistor goes beyond +/-5V the output of the schmitt trigger goes either positive or negative depending on polarity of the input. This output goes first to the square wave output through a voltage divider (the schmitt trigger outputs +/-12V, these resistors attenuate it down to +/-5V). It then goes through a pot resisting the flow of current into the integrator, the 220 ohm resistor between the pot and ground sets the minimum frequency.

The integrator is the middle op-amp, current from the schmitt trigger charges or discharges the capacitor in the op-amps feedback loop. The switch puts a second capacitor in parallel with the normal capacitor to put the module in low frequency mode. The output of the integrator goes to the input of the schmitt trigger and to a buffer. The buffer's output goes to an LED indicator and the triangle output and the mixer.

NOTE FOR +/-15V BUILDERS: The parts values presented in the schematic are for a +/-12V system. When building for +/-15V a few of the components associated with the schmitt trigger need to be replaced. These values are untested, you may need to experiment with different values to get the performance you want. The 220K in it's feedback path should be increased to a 330K. The 2.7K/2.2K voltage divider should be replaced with a 2.2K/1K voltage divider, and the 100K resistor between the pot's wiper and the integrator should be increased to 150K. In the Construction section, these parts are marked on the board.



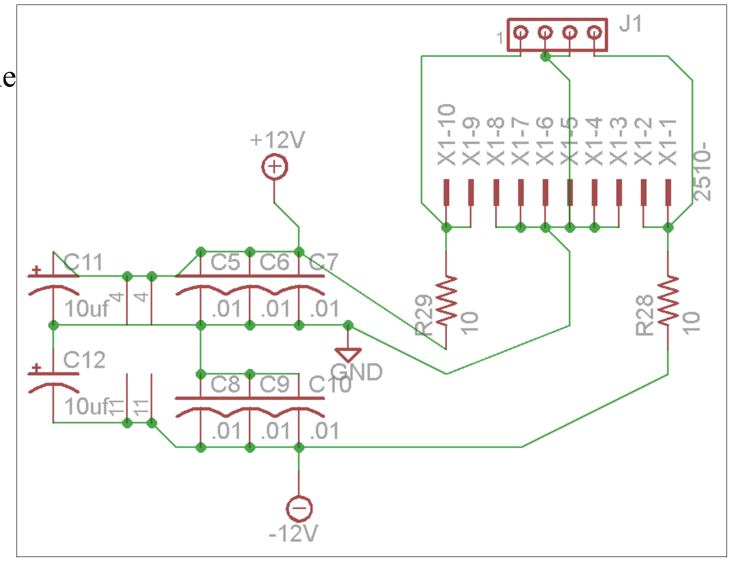
Sample and Hold

On the far left is the sample trigger input. It goes through a voltage divider and into a comparator. In testing without the voltage divider there were some accidental double triggers. The comparator goes positive whenever the input voltage goes slightly above ground. The output of the comparator goes through a .001uf capacitor and 100k resistor network to limit the output pulse length. The 1N4148 diode only allows the positive output of the comparator to reach the LM398. The LM398 is a monolithic sample and hold integrated circuit.

On pin of 3 of the LM398 we see the voltage input wirepad with a 100K resistor to ground, keeping the input voltage at ground when no input voltage is present. Pin 6 is connected to the hold capacitor which should be a poly type .01uf. Pin 5 is the output of the LM398 which goes through a 1K output resistor.

Power Supply

Here we see the two footprints for the power conencters. The external power is filtered by a 10 ohm resistor and 10uf capacitor on each power rail. We then see the power connections for the two op-amps, and decoupling capacitors at the power pins for all of the integrated circuits.



III. Construction

Parts List

Semiconductors

Value	Qty	Notes
TL074	2	
LF398	1	Accidentally marked "LM398" on PCB
1N4148	1	
LED	2	3mm

Resistors for +/-12v

Value	Qty	Notes
10 ohm	2	7.5mm lead spacing
220 ohm	2	" "
1K ohm	7	" "
2.2K	2	" "
2.7K	2	" "
100K	10	" "
220K	4	" "
A100K pot	2	16mm PC mount Pins

Resistors for +/-15V

Value	Qty	Notes
10 ohm	2	7.5mm lead spacing
220 ohm	2	" "
1K ohm	9	" "
2.2K	2	" "
100K	8	" "
150K	2	" "
330K	2	" "
A100K pot	2	16mm PC mount Pins

Capacitors

Value	Qty	Notes
.001 uf	1	5mm lead spacing Poly type
.0022 uf	2	" "
.01 uf	1	" "
.01 uf	6	2.5mm cheap ceramic decoupling cap. Value non-critical
1uf	2	Non-Polarized or Bi-Polar electrolytic 2.5mm lead spacing

10uf	2	Electrolytic 2.5mm lead spacing

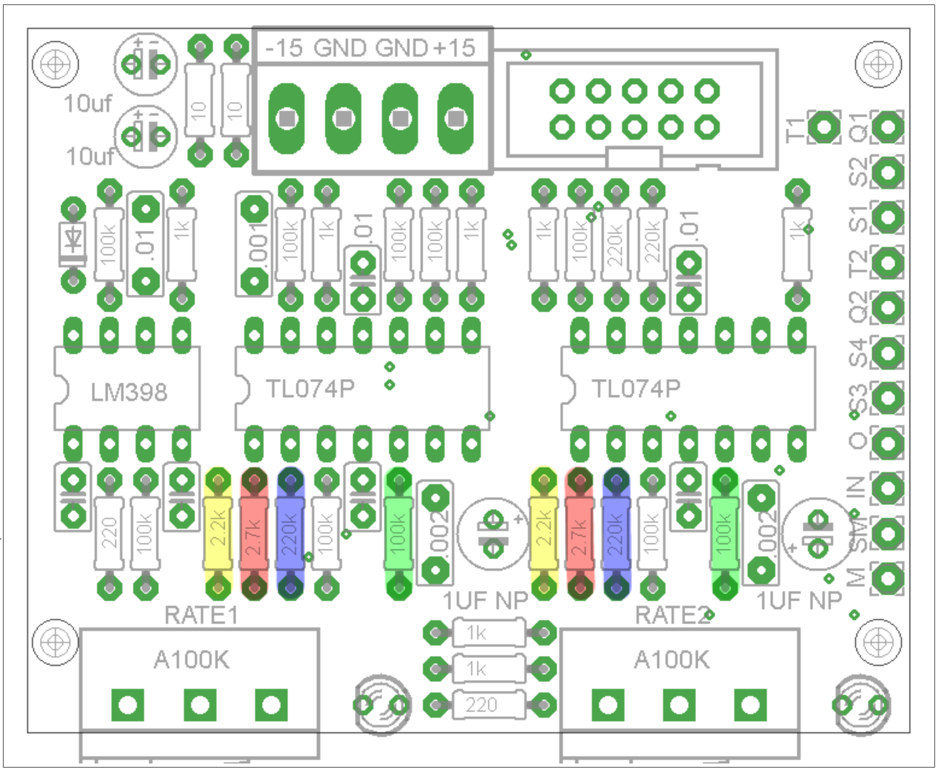
Other

Value	Qty	Notes
14 pin socket	2	DIP socket
8 pin socket	1	DIP socket
Power connecter	1	either MOTM or Eurorack type
SPDT Switch	2	or SPST, panel mount
Jack	8	1/4" or 1/8" mono, 2 should be switching type.
Knobs	2	

The Board

Here we see the printed circuit board. It is 62mm x 50mm and the spacing of the mounting holes is 57mm x 40mm. Pots are at 1 5/16" pitch.

The highlighted resistors are the ones needed to replace for +/- 15V. The Yellow resistor should be a 1K, the red should be 2.2K, the blue should be 330K and the green should be 150K.



Installing LEDs Sideways

The PCB indicates that the LEDs should be mounted parallel to the board, do not do this. Leds should be pointing in the same direction as the pots. The leads of the LED should be bent at a 90 degree angle, the easiest way to install them is in four steps:

1. Place the LED on the edge of the board facing out with it's leads going over it's pads on the PCB. Make sure the bottom lip of the LED is flush with the board.
2.Clip the leads 2 or 3 mm past the pads on the PCB.
3.Bend the LED leads 90 degrees 2 or 3mm from the edge.
4.The LED should pop into place easily.

Here we see a diagram for wiring. The text represents what wiring pad each connection should be made to. On the top left is a legend showing which connection on the jack is the tip and which is the switch for those unfamiliar with 1/4" jacks. If using 1/8" jacks the switch is usually the lug near the sleeve lug and the tip is the lug further from the sleeve lug.