BMC018. Analog Drum REVISION 2

Last updated February 26 2022

REVISION NOTES

1. LM13700 DIP package replaced with LM13700 SOIC package
2. 1K LED current limiting resistors replaced with 10K LED current limiting resistors

Previous revision documentation here.

I. Features

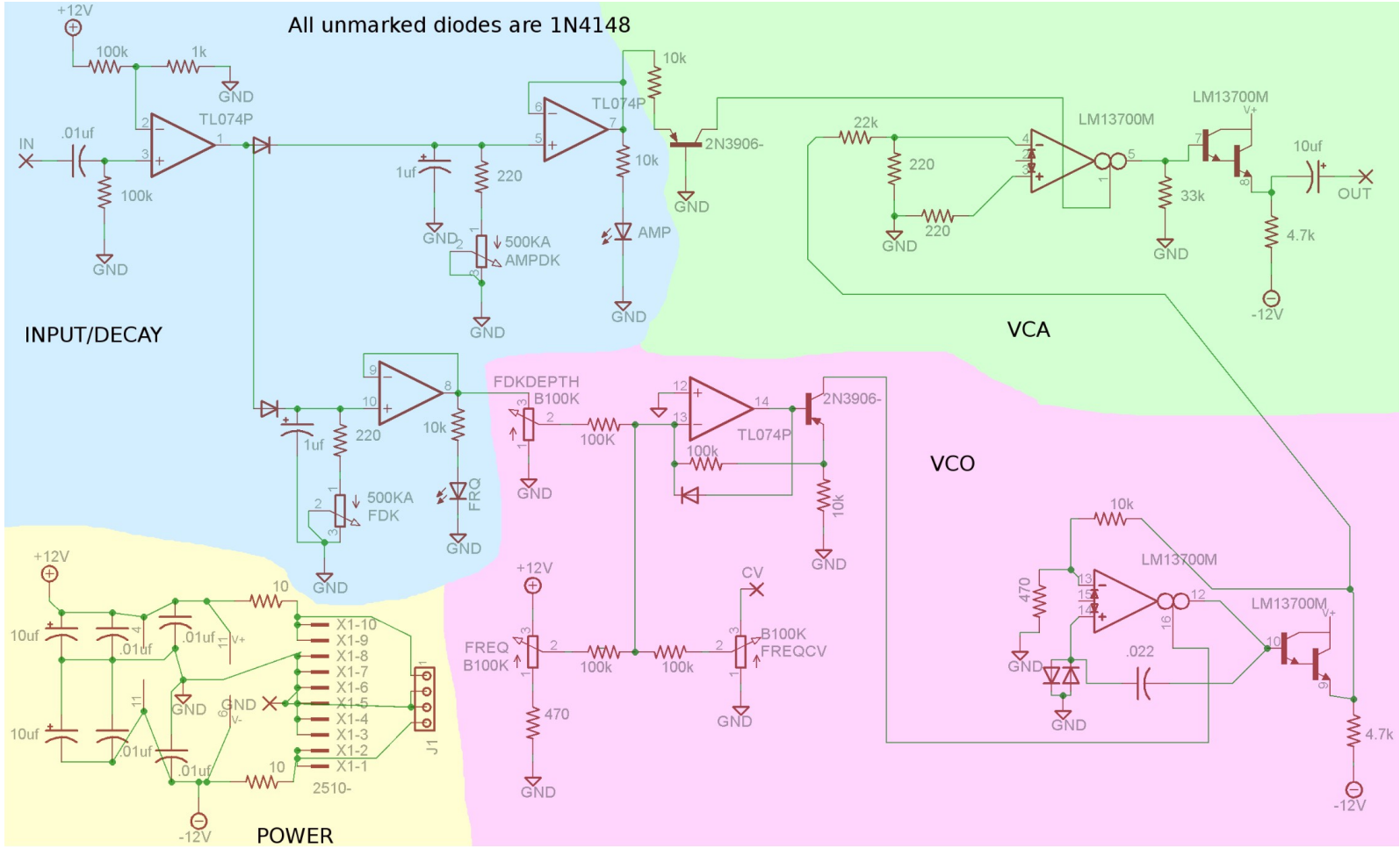
This module is an analog drum sound. It was designed to be built using a minimum of parts and no difficult to obtain parts. I initially was only trying to build a "disco tom" type sound, but I'm more impressed with it's bass-drum like sounds. It has the following controls:

1.Amplitude Decay
2.Frequency Modulation Decay
3.Depth of Frequency Decay
4.Baseline frequency
5.External Frequency CV Depth

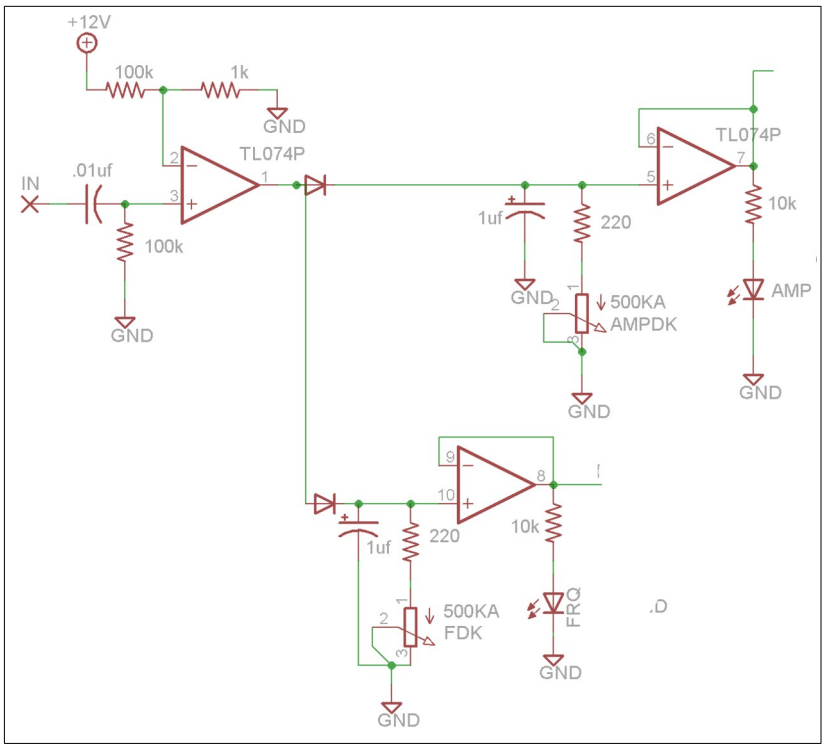
It has just two inputs, one for a trigger and one for external CV. The output is normalized to the CV input to allow for self-modulation.

II. Schematics.

On the next page is the master schematic. There are highlighted sections of the master schematic that indicate what sub-circuit of the module the parts correspond to, these sub-circuits are then described on the following pages.


All unmarked diodes are 1N4148
INPUT/DECAY
VCA
VCO
POWER
+12V
100k
1k
GND
TL074P
IN
.01uf
100k
1uf
220
500KA
AMPDK
AMP
10k
2N3906-
22k
LM13700M
33k
10uf
OUT
4.7k
-12V
FDKDEPTH
B100K
500KA
FDK
FRQ
FREQ
FREQCV
CV
470
.022
10
10uf
.01uf
X1-10
X1-9
X1-8
X1-7
X1-6
X1-5
X1-4
X1-3
X1-2
X1-1
J1
2510-

INPUT/DECAY

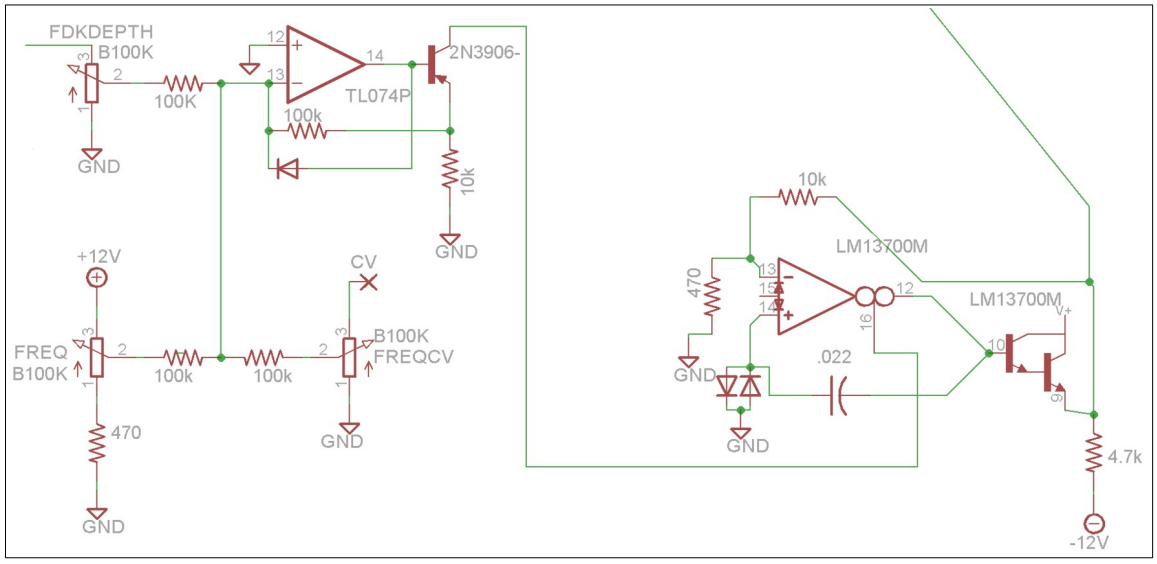


On the far left we see the wirepad marked "IN." A trigger or gate signal should be input here. The .01uf capacitor and 100K resistor to ground form a pulse shortener, making the pulse length of the input signal irrelevant. This is then input to an op-amp wired as a comparator. The output of the comparator goes to the two decay sections through a pair of 1n4148 diodes.

When the output goes high, current flows through these diodes quickly charging the 1uf capacitors. The 220 ohm resistors and 500KA pots in parallel with the capacitors provide a path for the capacitors to discharge the voltage. The higher the resistance, the more slowly the capacitors discharge.

Each capacitor is also connected to an op-amp wired as a buffer, which is outputing the capacitor's voltage onto the next stage, as well as lighting up an LED indicator. The output of the Frequency Decay section goes to the VCO, and the output of the Amplitude Decay section goes to the VCA.

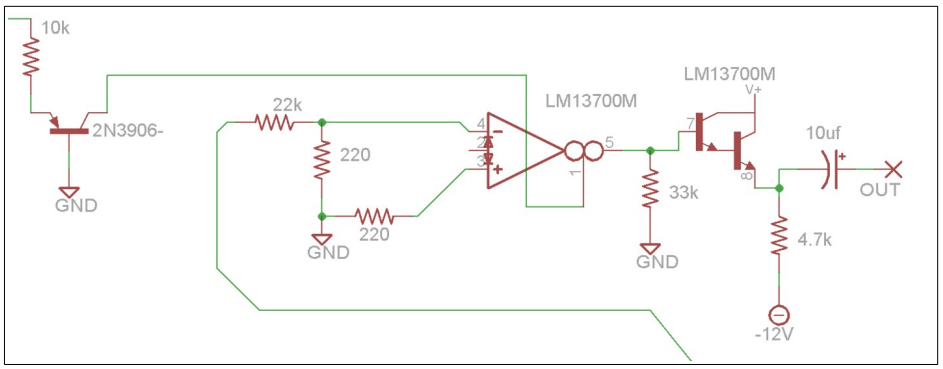
VCO



On the far left we see the input from the decay circuit. This voltage is summed with the voltages from the external frequency control and the baseline frequency control. These voltages are summed together on the negative input of the op amp wired in conjunction with a 2N3906 to form a linear voltage controlled current source.

This current source is controlling the current of one half of an LM13700 OTA wired as a VCO. The design for the VCO is taken from the LM13700 datasheet. The OTA is forming an Integrator/Schmitt Trigger type oscillator by itself.

IF BUILDING FOR +/-15V replace the 10K resistor in the feedback path of the OTA with a 15K. This is untested, if it works for you, or you like it more with a different value, let me know!

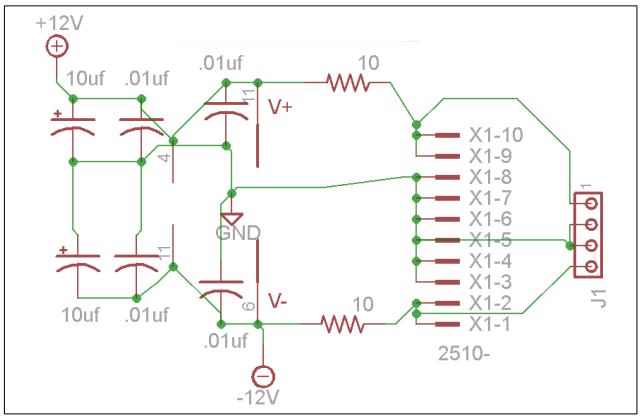


VCA

On the far left we see the output of the decay section for amplitude. A single 2n3906 in series with a 10k resistor forms the current control. This VCA is incredibly simple, the signal is input to the inverting input through a 22K and 220 ohm resistor voltage divider used to limit the signal on the input. The output of the VCA goes through the onboard buffer and then a 10uf capacitor to decouple the DC bias.

Power Connections.

Here we see the two power connecters for MOTM and Eurorack style systems. The supply is filtered by a 10 ohm resistor and 10uf capacitor, and then .01uf decoupling capacitors are placed near the power supply pins of the two ICs.



Parts List

Semiconductors

Value	Qty	Notes
TL074	1	DIP Package
LM13700M	1	SOT Package
2N3906	2	
1N4148	5	
LED	2	3mm

Resistors

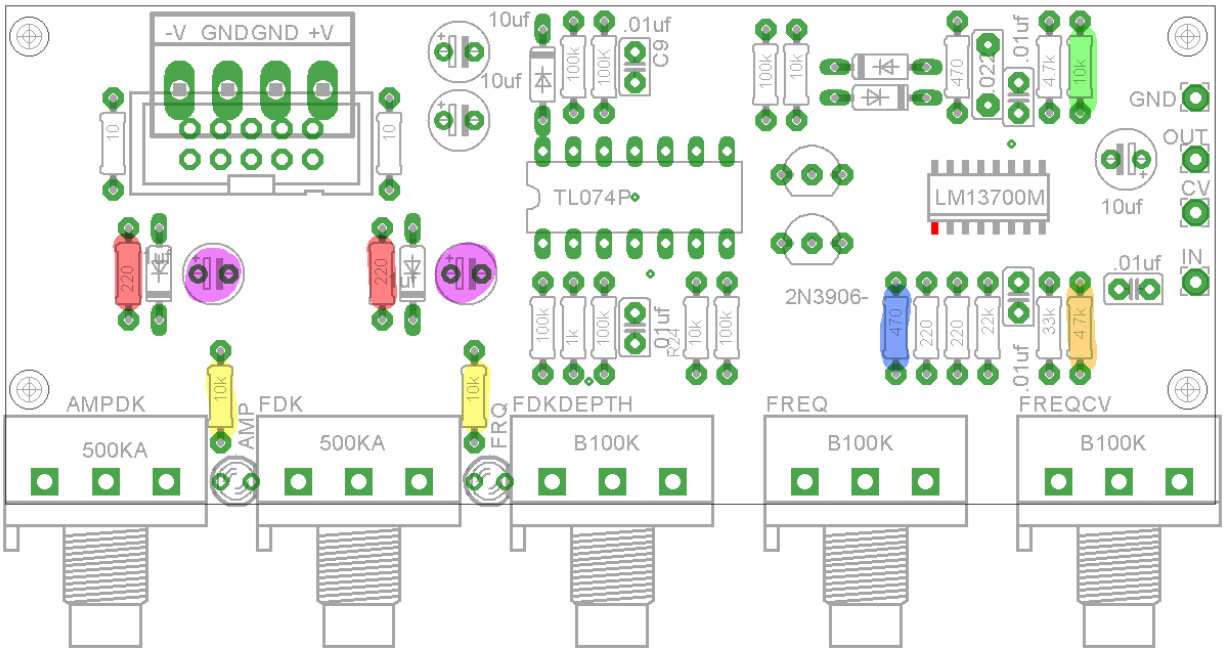
Value	Qty	Notes
10 ohm	2	7.5mm lead spacing
220 ohm	4	" "
470 ohm	2	" "
1K ohm	1	" "
4.7K	2	" "
10K	5	" "
22K	1	" "
33K	1	" "
100K	6	" "
500KA Potentiometer	2	16mm pot, PCB mounted
100KB Potentiometer	3	" "

Capacitors

Value	Qty	Notes
.01uf	5	cheap ceramic 2.54mm
0.022uf	1	Poly 5mm lead spacing
1uf	2	Electrolytic
10uf	3	" "

Other

Value	Qty	Notes
Power Connecter	1	Either Eurorack or MOTM
14pin DIP Socket	1	
Jacks	3	Either 1/4" or 1/8"
Knobs	5	



The Board

The PCB is 100mm x 41mm. The pots are spaced 21mm apart. The mounting holes are spaced 91mm x 29mm. Pin 1 of the 13700 is highlighted in red

I've highlighted a few parts that a builder may want to modify.

The yellow resistors control the brightness of the LEDs. 10K is a value that won't be overly bright for most values. To increase the brightness lower the value of the resistors and to make the LEDs darker, increase the resistor value.

The red resistors control the minimum decay time. To increase the minimum time, increase the value of the resistor. If you have a problem where when turning one of the decay pot times down all the way where the decay controlled by the other pot is also shortened, you should increase the value of these resistors.

The purple 1uf capacitors control maximum decay time. To get longer decays this can be increased to a 10uf or beyond, to get finer control over short decay times, a smaller cap can be used.

The blue resistor controls the minimum frequency on the frequency control pot. Increase it to increase the minimum frequency.

The orange resistor biases the output buffer, if your output waveforms have unwanted clipping, this can be lowered to a 2K or 1K which should give a cleaner waveform.

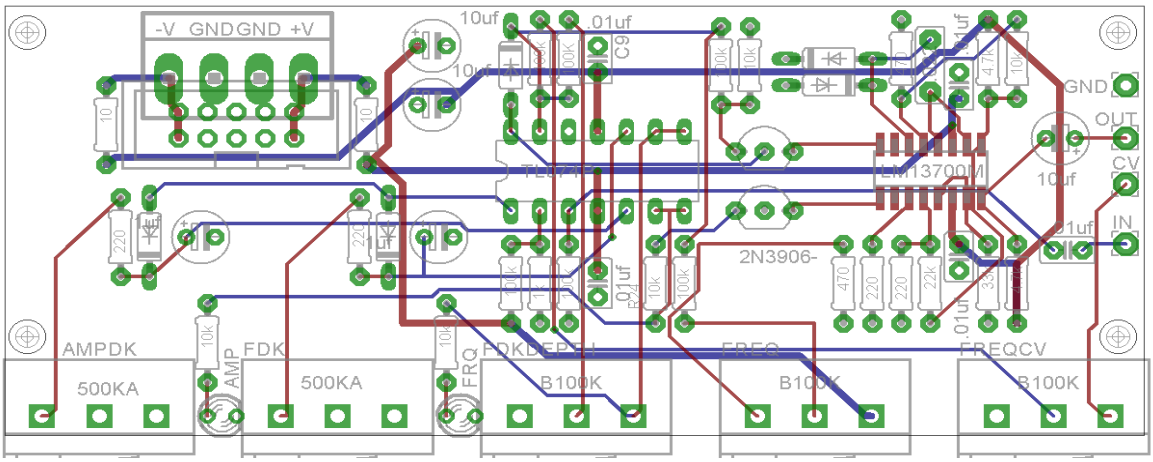
The green resistor is the 10K that should be replaced with a 15K for +/-15V

WIRING

The PCB has four wirepads, they should be connected as follows:
IN → Tip of the Trigger/Gate Input Jack
CV → Tip of the CV Input Jack
OUT → Tip of the Output jack, and optionally the switch of the CV jack.
GND → The sleeve of any one jack if using a metal panel. If using a non-conductive panel, then

connect the sleeves of the jacks together.

Below is an image of the PCB with traces included to help when troubleshooting. The ground plane is not depicted, so assume pads with no connection are connected to ground:



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.