

BMC15 Quad Mix/VCA

Last edited September 14, 2024

This documentation covers PCB version 4.1 (w/SMD footprint for 13700) and version 3.1 (w/through hole footprint for 13700).

1.Parts List

2.Board Layout/Wiring Instructions

3.Set up.

4.Schematics.

5.Using Rev 3.1 board with Rev 4.1 parts list

This module contains 4 VCA circuits normalled into a 4 channel mixer. VCAs can be used individually and their outputs are automatically removed from the mixer. Each VCA has an attenuator for CV, when no CV is input a +5V supply is normalled in, so it can be used as a 4 channel mixer without having to use external voltage sources.

PCB Changes in REV 4.1:

1. Changed resistor value in CV reject circuit for easier calibration.

[REVISION 4 Documentation here.](#)

1. Parts List

Semiconductors:

Name	Quantity	Notes
LM13700	2	SOIC Package for Rev 4.1 board, DIP package for Rev 3.1
TL074	2	
TL072	1	
2N3906	4	
7805	1	For Rev 4.1 use 78L05 TO-92 package, for Rev 3.1 use TO-220 package
1n4148	4	

Resistors:

Name	Quantity	Notes
10 ohm	6	5mm package
470 ohm	4	On Rev 3.1 board some are marked as 10 ohm, see page 8
1K ohm	4	
10K ohm	4	
47K	1	
100K	30	On Rev 3.1 board some are marked as 4.7K, see page 8
330K	4	
B100k trimpot	8	
B100k Pot	5	16mm PCB Mounted

Capacitors

Name	Quantity	Notes
10pf	4	ceramic 2.5mm lead spacing
.1uf	10	" "
10uf	2	electrolytic

Connecters

Name	Quantity	Notes
DIP8	1	Socket
DIP14	2	Socket
DIP16	2	Socket (Rev 3.1 board only)
Power Connector	1	MOTM or Eurorack
Switching jacks	13	

2.Board Layout/Wiring Instructions:

Each channel has 5 wires to be connected.

I should be connected to the tip of the input jack

T should be connected to the tip of the CV jack

S should be connected to the switch of the CV Jack

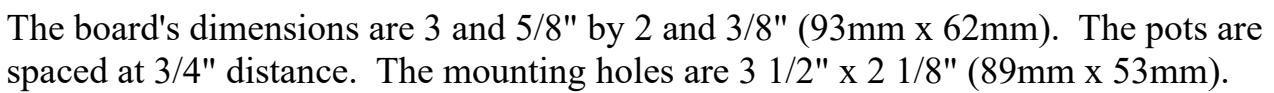
O should be connected to the tip of the output jack

M should be connected to the switch of the output jack

MIX should be connected to the tip of the mix output jack

To reduce gain to less than 1, replace yellow highlighted resistors with 47K

Pin 1 of the LM13700 is marked in red.



The board's dimensions are 3 and 5/8" by 2 and 3/8" (93mm x 62mm). The pots are spaced at 3/4" distance. The mounting holes are 3 1/2" x 2 1/8" (89mm x 53mm).

[illegible]

Each channel has two trimpots that need to be adjusted.

1. Input a square wave from a VCO into the CV jack of a channel and nothing into the input jack. Turn the knob for that channel all the way up. Adjust the "Reject" pot for that channel until you hear the square wave the least.

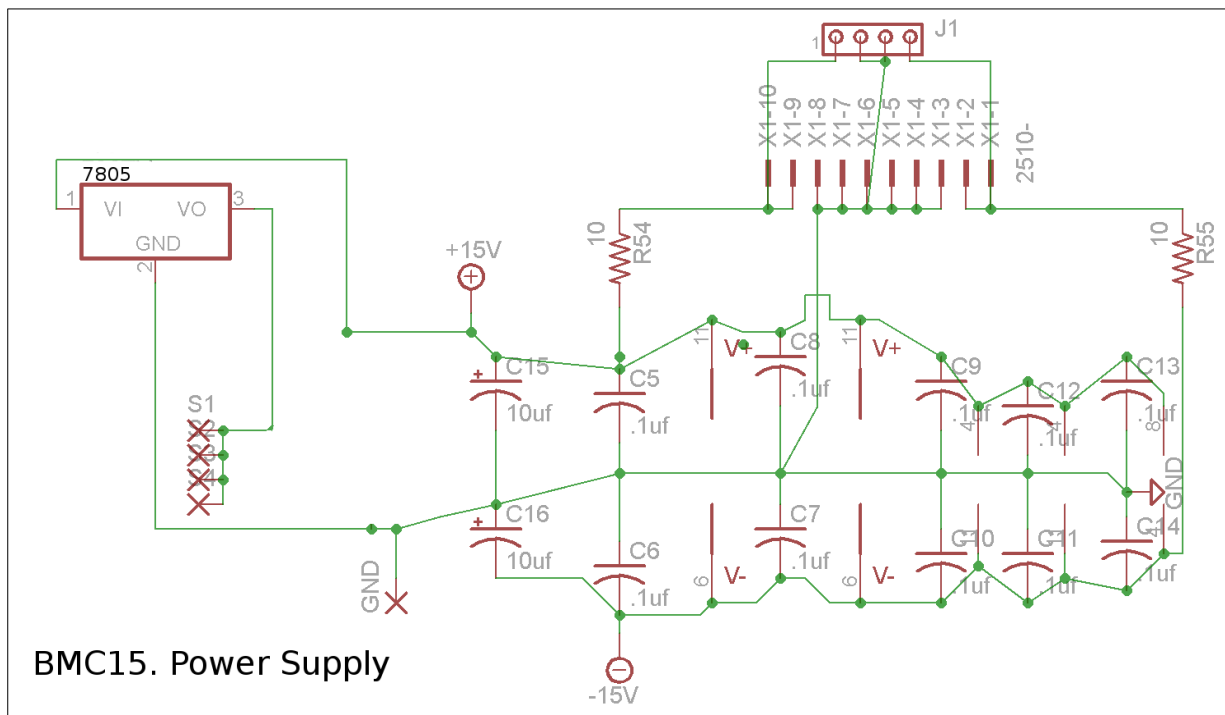
2. With nothing plugged into the CV jack and a square wave plugged into the input, turn the knob for the channel all the way down. Then adjust the "Offset" trimpot for that channel until you hear nothing.

The diagram shows a multi-stage electronic circuit. At the top left, a 10V supply is connected to a 100k resistor, which is in series with a 10 ohm resistor. This is followed by a 100K resistor and a 470 ohm resistor. A 100k resistor is also connected to a 10 ohm resistor. The circuit includes two TL074P op-amp stages. The first stage has its non-inverting input (+) connected to a 100k resistor and its inverting input (-) connected to a 100k resistor and a 10pf capacitor. The output of the first stage is connected to a 10k resistor and a 100k resistor. The second stage has its non-inverting input (+) connected to a 100k resistor and its inverting input (-) connected to a 100k resistor and a 10pf capacitor. The output of the second stage is connected to a 1k resistor and a 100k resistor. A 2N3906 transistor is connected to the output of the second stage. The circuit is powered by a 10V supply and ground. The output is labeled 'Out' and '1'.

**BMC015.
REV 4.1
VCA CELL**

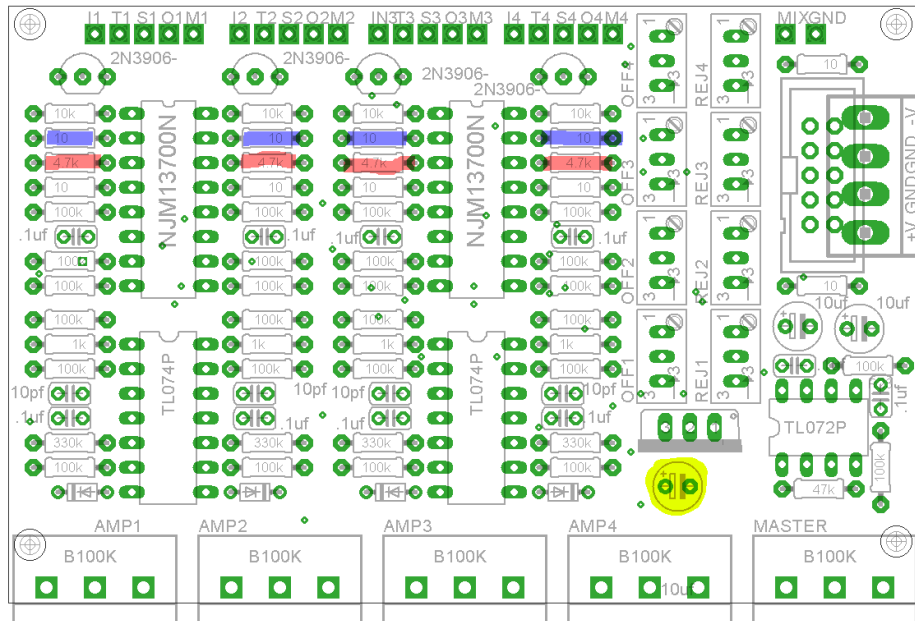
http://www.birtofasynth.com/Thomas_Henry/Pages/VCA-1.html

The mixer section just consists of an inverting gain stage that mixes the outputs, an attenuating potentiometer and a unity gain inverting stage.



Finally we see the power supply. On the top are the power connectors. The 10 ohm resistors are used to filter out power supply noise. On the left is a 7805 voltage regulator, this is used to provide a steady 5v reference for normalizing to the CV attenuators when no input is present. The rest of the schematic just shows the power rails connecting to the ICs supply pins and decoupling caps.

5.Using 3.1PCB with Rev 4.1 values.



Refer to the image above to show which parts to replace

Blue 10 ohm resistors should be replaced with 470 ohm resistors

RED 470K ohms become 100K ohms

YELLOW 10uf capacitor doesn't need to be stuffed. It was for filtering signal from the 7805, but it's not necessary

Below is an image of the 3.1 board with traces to aid in troubleshooting

