MB-808

The TR-808 Clone with the MIDIbox Sequencer

ASSEMBLY MANUAL

INTRODUCTION

The MB-808 is a reproduction of the Roland TR-808 drum machine. As of 2010, there are three revisions to the MB-808 circuit board. The third revision is quite mature – I think you'll find it something you can be proud to build. Just ask my wife how much of my life I put into it!

(Reference MIDIbox info and license)

While the analog voice section of the MB-808 is derived from the Roland TR-808 design, the sequencer is taken from the MIDIbox project.

(Reference 808 web site and forum)

OPTIONS

NOTE: There have been three revisions of MB-808 pcbs and each has shipped with a different set of parts.

As a minimum, you should order everything on the Mouser parts list and on the Digikey parts list. In addition to that, everyone will need a mounting panel, knobs, and 3mm leds (SmashTV sells 100 packs for a very good price if you don't mind the 4-5 week delivery time), MIDI connectors, heatsinks for the three regulators, fuse(s) for the mains input, standoff washers and screws, plus some kind of case, and second batch people will need to order the lighted switches (see forum for sources, including group buy).

Front panel notes: There are two options for mounting the pcb to the front panel - 1) use the .375 standoffs with washers to clear the .400" height of the RK09L potentiometers, or 2) use the washers and nuts on the potentiometers to mount the panel. When you use the standoffs, you won't have to worry about covering up the potentiometer washers and nuts, but if you use the washers and nuts, we'll be able to complete the graphics (no screws getting in the way), plus we won't have to worry about a possible shock hazard with the first batch of boards. The number of standoffs listed on the BoM has been incorrect before rev 4.4 - there are actually 13 mounting holes in the pcb, so if you use option one, you'll need 13 standoffs with washers and screws. If you use the potentiometer mounting method, you'll still need four standoffs to support the momentary switches at the bottom of the board.
Potentiometers and encoders - the 9mm potentiometers on the BoM are a different type than the ones included in the kit. If you're interested in having them all be the same, don't order the ones on the BoM and check out this topic on the forum for discussion. Also, the 24 detent encoder listed is 5mm shorter than the rest of the pots. While the 18 pulse per full rotation encoder is the correct height, you might consider a 12 pulse per rotation version since there are 12 voices in the basic MB-808 setup. I'm having a distributer give me a quote on the correct size encoders. For those of you who want to look for them yourselves, here are the part numbers (all Bourns):

- PEC11-4225F-N0012 - 24 detents with 12 pulses per 360 degrees (2 clicks to send one pulse)
- PEC11-4125F-N0018 - 18 detents with 18 pulses per 360 degrees
- PEC11-4225F-N0024 - 24 detents with 24 pulses per 360 degrees

"RT" switches at top of panel are optional and may not be needed if the optional mixer/effects add-on panel is used.
If you're interested in using anything in the aux section, you'll have to buy the appropriate sized potentiometers and/or DPDT switches. For each potentiometer, you'll need a 3-pin header, and for each switch, you'll need two 3-pin headers - you can just add another 40 pin breakaway header for this.
If you are planning to use an LCD, you'll have to add in two 10k trim pots and one BC337 transistor, along with the LCD itself and a ribbon cable. Looks like you should also get a 16 pin DIL (two row) header.
If you are going to build any of the options (909 voice add-on panel, mixer/effects add-on panel, LCD meter bridge) you have two options. Now that we have 2A regulators, (thanks to SmashTV for the link to the +5V 2A regulators - it was easy to find the other two once I knew where to look!) you can run those options from the psu on the main board. You also have the option to leave off most of the power supply components and mount them on a separate power distribution board. (See minimal BoM for what is needed for that.) Also, it is probably a good idea to upgrade the transformer for the extra add-

ons - as a matter of fact, I went ahead and modified the Digikey BoM to list the 25VA transformer - if you know that you will not add any of the options, the original 62053 will be fine. If you already purchased the 62053 transformer, but want to add options, you can supplement it with a 62050. Wiring options - pin headers are included for most connections (but not the aux section). You can solder connector wires directly onto the PCB, but it's easier to assemble and disassemble if you use the headers. As of B0M version 4.8, I have included the crimp pins and housings that you'll need to attach to the connecting wires. One of the 3-pin and ten of the 2-pin connectors are used to access the instrument triggers - if you don't want to access the triggers, you can delete those headers and crimp pins from the BoM.

PLEASE NOTE: I am not an Excel expert by any means. Also, I have not reconciled the costs between the spreadsheet and the Mouser site - don't go by the costs in the spreadsheet - let your Mouser Shopping Cart be the final answer.

STAGE 1 OF ASSEMBLY

OVERVIEW

We're going to go across the pcb twice, the first time to install resistors, capacitors, transistors and diodes, the second to install trimmers, potentiometers and install ICs.

For stage 1, I suggest using organic core solder if you can. Organic core solder has a stronger flux which will help you create better solder joints. The problem is that if you don't wash the flux off your board, it will remain activated and continue to etch your parts. Over time this will cause failures. In addition, you don't want to use organic core solder on anything where the flux can get into a part and not be washed out. (Do not use organic core solder on potentiometers or stranded wire, or any other part that you can't properly clean.)

After we've installed all the parts that are easily washed, we'll switch to rosin core or no-clean solder and attach the remaining parts and test each voice as we go along in stage 2 of assembly.

This is the pre-release version of the assembly document. I will include all bills of materials and all individual voice circuit pictures here eventually. In the meantime, please refer to the Bills of Materials web page at:

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/

and the individual circuit pictures at:

http://www.eight-oh-eight.org/Individual_Circuits/Pics/

PREPARATION AND PARTS

1.) *Carefully* open the box, I would not recommend using a box cutter or any unusually brutal scissors. All you need to do is cut the tape along the seams on the front and sides and the box will open.

2.) Don't throw anything away! Not until you've located everything on this list:

1 Tube of IC's containing 14 JRC4558's and 1 CA3080
1 Bag containing 5 1MB dual pots and 1 500KB dual pot
1 Bag containing 4 DPDT switches
1 Bag containing 16 100KA pots, 1 50KB pot and 1 50KA pot
1 Bag containing 6 1N60 Diodes, 1 2SK30A Transistor and 1 2SC828 Transistor
1 Bag containing 33 Lighted switches (With batch #1 and #3 kits...)
1 Bag containing 50 2SC945 Transistors (assorted brands)
1 Tube containing a PIC18F4620 preburned by SmashTV
1 PCB saran wrapped to preserve freshness

3.) Ordering the remaining parts: Refer to the spreadsheets at http://www.eight-oh-eight.org/parts and discussion posted in the Partslist and Datasheets thread

Check the parts list and errata on the http://www.eight-oh-eight.org web site.

Parts
http://www.eight-oh-eight.org/parts
http://www.eight-oh-eight.org/phpbb2/viewtopic.php?t=42

Errata
http://www.eight-oh-eight.org/phpbb2/viewtopic.php?t=171

POWER SUPPLY

WARNING: You will be dealing with AC mains voltages in this part of the build, if you are not careful you can injure yourself, or worse! Keep a clean bench and *PAY ATTENTION* to what you are doing!

2	1K	Resistor	R382, R383
6	100n	Polyester Film Capacitor	C9, C23, C24, C25, C26, C112
22	10u/25	Electrolytic Capacitor	C211, C212, C213, C214, C215, C216, C222, C223, C224, C225, C226, C227, C228, C229, C230, C231, C232, C233, C234, C235, C236, C237
2	2200u/16	Electrolytic Capacitor	C21, C22
2	2200u/35	Electrolytic Capacitor	C8, C10
9	1N4004	Power Diode	D29, D32, D33, D34, D35, D36, D37, D38, D41
2	3MM	LED	LED1, LED2
1	4 Pin	.156" Spaced Header	SV1
2	Shurter	PCB Mounted Fuse	F1, F3
1	L78S05CV	+5 Volt Regulator	IC37
1	L78S15CV	+15 Volt Regulator	IC2
1	L7915CV	-15 Volt Regulator	IC36
1	2 pin	Keyed Header	+5V
1	3 pin	Keyed Header	+/-15V

1 Solder in all diodes except D41.

2.) Solder in the two 1K resistors and LED's 1 and 2.

2.) Solder in all 100n capacitors except c24. Any poly or electrolytic cap will work here and a voltage rating of at least 35v is recommended. Place c24 on the top side of the board so it doesn't get in the way of the heatsink on the 5v regulator. Leave long enough leads on the cap so that it will lie flat on the PCB and not get in the way of the front panel.

3.) Solder in the 4 pin connector for the transformer with the friction lock on the north side of the board (this is in backwards on mine).

4.) Solder in the fuses. The PSU will still function without them, but do so at your own risk.

5.) Attach heatsinks to the regulators. You *must* use a heatsink on the 5v regulator, but you may be able to get away without them on the +-15v regulators. In either case be aware that these IC's can and will get *very* hot!

6.) Solder in the Regulators with the metal tab to the north side of the board.

7.) Solder in the big lytic caps. These should all be at least 35 volts except c22 which can be as low as 6.3v because it is after the 5V regulator. It's not a bad idea to use a 2200uF 16v cap for c22.

8.) Double check that all components except D41 are soldered in. Check for solder bridges by using the continuity tester on your DMM.

9) Connect the wires of the transformer's primary winding to the power jack and power switch. If you are using the Amveco transformer the wires should be connected Red and Yellow together to one pole of the switch and Black and Purple together to one tab on the power jack. Connect the other tab on the power jack to the other pole of the switch. In the Pic you can see that I soldered the power switch directly to the inlet jack. Obviously this won't work in a case, but while testing it kept the amount of wiring that was carrying AC mains to a minimum.

10) Connect the wires from the secondary windings on the transformer to the connector in the order Blue, Brown, Red, Green.

11) Place a 125v fuse in the power jack.

12.) Connect a power cable and test for AC voltage from Blue to Brown and Green to Red

13.) Connect the power connector to the PCB, power it on, and check for voltage across the two large capacitors. If there is at least 15v then check for +-15v at the outputs of the regulators (the LED's should be on and stay on).

14.) Solder in D41. BE ABSOLUTELY SURE THAT THE CAPS ARE DISCHARGED. Place a 10K resistor across their terminals for 10 seconds first if you are not sure.

EDIT: For those with the first run of PCB's I recommend soldering in D41 as shown in this pic

http://www.eight-oh-eight.org/Pics/MB808_PSUMod.jpg

This will reduce the amount of excess voltage going into the 5v regulator and probably provide a more stable PSU.

14.) Check the output of the 5V regulator.

NOTE: Those of you who got PCB's from the first batch should take care with the following PCB spacer screw, it is placed very close to the transformer input and if your case is not properly grounded there could be a chance of having an electrified panel (!!). To be absolutely safe I would recommend running a ground wire from the PCB spacer to ground, although the usual methods of grounding the case should be sufficient.

NOTE: Those of you who got PCB's from the second batch need to add a jumper wire to connect the ground planes between the analog (top 2/3rds) and digital (bottom 3rd) of the PCB. The pic shows how: use a flathead screwdriver (or something similar) to scratch away some of the green coating from the copper ground plane. Make a blob of solder there and then run a jumper like in the pic

CPOL = Electrolytic capacitor. R-US is a resistor. C-US is a poly or ceramic capacitor, only the ceramics are in the pF size with the exception of the 100n caps in the digital section. All the 2SC945's are marked on the board with a white line through the symbol right through the three legs, the 733's have no line. Still pay attention in the noise section tho to be sure you put the 2SC828 there instead of a 733 tho. All the "MA03", "MA02", "MA04" parts are pin headers on the board. The "78V" parts listed in the BOM are the 7815 and 7805 regulators (15v and 5v). The 7915 is the -15v regulator.

NOTE: Power capacitors are 2200uF rated at 35v not 63. The right part number is: 140-XRL35V2200-RC.
These heatsinks may be too big:
Heatsinks: 3x 532-504222B00
Mounting Kit: 3x 532-4880M
These may work: http://www.mouser.com/search/ProductDetail.aspx?R=532-577202B00

LEDs 638-1224UYOCS5302

See Power supply mod: http://www.eight-oh-eight.org/Pics/MB808_PSUMod.jpg

Transformers: rectifying ac to dc results in higher voltage. measure the dc output going into the voltage regulators (after the diode rectifiers) and you'll notice that the voltage is roughly 18vdc.
To add to this: AC trafos are rated in RMS V. The formula for calculating peak voltage is RMS V x 1.414. In a bridge rectified circuit, you also have to subtract off 1.4v for the two diodes so in the case of the MB808 you have 15-1.4 = 13.6 * 1.414 = 19.23VDC
just a warning to everyone that you should connect the analog and digi ground planes before connecting the 5v reg. when there is no reference voltaqage (ground) hitting the 7805 it sends 13-15 volt through the digi section's 5v line. this might cause some damage.

CORE

For additional information about the MidiBox Core module, see these pages:
http://www.ucapps.de/mbhp_core.html
http://www.avishowtech.com/mbhp/mbhp_coreR4d.html
http://www.ucapps.de/mbhp/mbhp_core_v3.pdf

PARTS:

1	100R	Resistor	R137
3	1K	Resistor	R10, R11, R18
1	1K2	Resistor	R139
2	10K	Resistor	R8, R143
3	220R	Resistor	R138, R140, R141
1	5K6	Resistor	R142
2	33p	Ceramic Disc Capacitor	C19, C20
2	100n	Multilayer Ceramic Capacitor	C17, C18
1	1N4148	Diode	D1
1	10MHZ	Timer Crystal	XTAL
2	3 Pin	Header	MIDI-IN, MIDI-OUT
2	8 pin	IC Socket	IC1, OPTO-ISO
1	40 pin	IC Socket	MCU
INSTALL LATER:			
1	PIC18F4X2DIL40	Microcontroller	MCU
1	24LC256P	EEPROM	IC1
1	6N138	OptoCoupler	OPTO-ISO
1	10KB	Pot	SWING
OPTIONAL:			
1	16 pin	Dual 8 Pin Header	SV4
2	10K	Trim Pot	R6, R7
1	BC337	Transistor	T1
1	4 Pin	Header	SV3
NOT NEEDED? (And not on main BoM...)			
1	8 pin	Dual 4 pin header	SV2

1.) Solder in all resistors

2.) Solder in the 2 100n (104) capacitors

3.) Solder in the Crystal, it does not matter which direction

4.) Solder in the Diode

5.) Solder in the 40 pin socket for the PIC and the two 8 pin sockets for the Opto Isolator and the EEPROM

6.) Solder in the optional components R6, R7 and T1 which are used to adjust the brightness and contrast of the LCD screen. Transistor T1 is a BC337 so if you want to use a 2n3904 you will need to bend the leads as the pinout is different (I'll check the datasheet and make a diagram).

7.) Solder in the two 33p capacitors next to the crystal

8.) Before putting any IC's in their sockets power the unit on and test for 5v across the 100n capacitors.

9.) Connect a MIDI cable to the output connector

10.) Using a program like MIDIOx or (preferably) MIOS Studio (http://miosstudio.midibox.org/) watch the input midi monitor when you power the unit on (remember to put the IC's in now). You should see a HEX message similar to F7 00 00 00 .. etc. If you see nothing start checking components around the MIDI out port.

11.) Alternately you can connect a LCD to the Core and see if the application boots up. If you can't see anything at first remember to adjust the brightness and contrast pots. I'll post some more detailed information about connecting the LCD's once I find it, but remember that this is all standard MidiBox stuff, so search the midibox forum and wiki! Be careful with a LCD hooked up.. in all likelyhood the 5v regulator is going to get *very* hot!

NOTE FOR Batch 2 PCB's
If you are going to use a LCD, be sure to solder a little trace connecting the ground for the LCD as shown in this pic:

http://www.ucapps.de/midibox_808/lcd_fix.jpg

R137 is 100Ohms
c17 and c18 multilayer ceramic
Transistor T1 is a BC337
http://www.midibox.org/users/jim_henry/building_a_midibox_lcd_cable.pdf

Make sure your LCD is hitachi 44780 compatible.

total of 15 100n (104) multilayer ceramic caps needed, 2 for the core, 7 for the DOUT and 6 for the DIN

DOUT

For more information about these circuits check out these sites:
http://www.avishowtech.com/mbhp/mbhp_doutR3.html
http://www.ucapps.de/mbhp_dout.html

PARTS:

7	100n	Multilayer Ceramic Capacitor	C1, C2, C3, C4, C5, C6, C7
2	3MM	LED	BAR-LED, BEAT-LED
2	.375	LED Standoff	BAR-LED, BEAT-LED
14	16 pin	IC Socket	IC9, IC10, IC11, IC12, IC13, IC14, IC15
			RN1, RN2, RN3, RN4, RN5, RN6, RN7

INSTALL LATER:

2	100R	Resistor Network	RN1, RN4
5	220R	Resistor Network	RN2, RN3, RN5, RN6, RN7
7	74HC595N	Shift Register	IC9, IC10, IC11, IC12, IC13, IC14, IC15

correct resistor networks:
5x 220 Ohm, Mouser part number: 652-4116R-1LF-220
2x 100 Ohm, Mouser part number: 652-4116R-1LF-100
left to right: 220, 220, 220, 100, 220, 220, 100

The top two rows of IC's in the digital section are the Digital Outs which are responsible for controlling the LED's and sending the Instrument and Accent trigger pulses.

TIP: It is not a bad idea to test for continuity between the 5v rail and ground after soldering in each Socket, it's much easier to identify solder bridges this way.

1.) Solder in the 16 pin sockets (Note, some early versions of the BOM have the count for 14 and 16 pin sockets mixed up, you will need 22 16 pin sockets total)

2.) Solder in the 100n (104) capacitors, one per set of sockets.

NOTE: Do not solder in the lighted switches yet, those will wait until just about everything else is built.

NOTE: In order to use all the LED's for the new firmware, be sure to make two additional connections from DOUT pins to the LED's in the Alt and Shift buttons and then lift up two of the leads that connect to the Record and Aux LED's which are connected to ground on the PCB. TK describes this on the second page of this thread and provided this pic:

Here the fixes for the 4 missing LED functions:

- missing connection between Shift LED and DOUT #6, Pin #2

- missing connection between Alt LED and DOUT #6, Pin #1
- ground connection (bridge, short circuit!) at DOUT #7, Pin 4, connected to Record LED
- ground connection (bridge, short circuit!) at DOUT #7, Pin 5, connected to Aux LED

I fixed the missing connections with two isolated wires from the resistors to the LED anodes

To fix the grounded connections, I put the appr. pins of the 74HC595 out of the IC socket, desoldered the resistors and connected them directly to the 74HC595 pins like shown here:

http://www.ucapps.de/midibox_808/led_fix.jpg

I've been looking closer at the DOUT section. To avoid the modification TK made, and to allow other modifications later (adding trigger/led/other outputs), it's going to be easier if we modify the circuit board a bit before we solder the IC sockets on, then once we have the sockets in place, we'll solder in some jumpers.

I'll take some pictures at some point, but the idea is to disconnect some of the solder pads from the ground plane - none of the pads for the resistor networks (RN1-7) should connect to ground, and none of the shift register outputs should connect to ground. There are five resistor network pads that need to be disconnected, and eight shift register pads that need to be disconnected. Take a small tool and scrape away the copper between the pad and the ground plane. Use a multimeter to make sure there is no connection to the ground plane.

Once the IC sockets are in place, we need to attach all shift register outputs to the resistor networks. Looks like there are four connections to make for that...

One other note - to allow the most flexibility for modification, you can leave off the resistor networks and sockets and use individual resistors as TK has done...

Sasha: I would suggest you to use some light source behind PCB so you can clearly see the bridges while you are removing it.

Batch two: grounds: http://hakkenmusic.googlepages.com/un-gnds.jpg

DIN

For more information about these circuits, please check out these sites:
http://www.avishowtech.com/mbhp/mbhp_dinR5.html
http://www.ucapps.de/mbhp_din.html

PARTS:

12	10K	6 Pin Resistor Network	RN13, RN14, RN15, RN16, RN17, RN18, RN19, RN20, RN21, RN22, RN23, RN24
6	100nf	Multilayer Ceramic Capacitor	C11, C12, C13, C14, C15, C16
6	16 pin	IC Socket	IC3, IC4, IC5, IC6, IC7, IC8

INSTALL LATER:

6	74HC165N	Shift Register	IC3, IC4, IC5, IC6, IC7, IC8
2	18 or 24 detent	Rotary Encoder	INSTRUMENT, TEMPO
33	5501	Switch with LED	MENU1, MENU2, MENU3, MENU4, A, B, GP1-GP16, SELECT, EXIT, LOOP, RWD, FWD, STOP, PLAY

TIP: It's not a bad idea to check for continuity between the 5v rail and ground after soldering in each IC socket.

1.) Solder in the 16 pin IC sockets.

2.) Solder in the 100n caps

3.) Solder in the in line resistor networks. They should have a marking on one end indicating the common pin. This pin should be connected to the 5v rail on the pcb (the thick trace) this pin is on opposite sides for each side of the IC. You can ignore the resistor that is silkscreend on the PCB. If your networks have an extra pin (or more) just bend the unused ones up out of the way.

4.) Don't solder in the switches, they will be the last components to go in the board.

there is an error in the BOM here...these should be 100nF caps, not 10nF

ok...I don't understand this...so the resistor networks are all 5 pins on the silk screening...the resistor networks in the BOM are 6 pins...you can bend the unused pin out of the way, but there is a resistor hole R1-R5,R27 that is directly under where that resistor network hole is....so what are you supposed to do, put R1-R5 on the other side of the board? or are those the resistors you're talking about ignoring...so R1-R5,R27 shouldn't be put in?

You've got it correct, for the networks on the north side of the sockets the 6th pin goes in the resistor hole that is there, on the south side of the sockets just bend up the unused pin. Be sure that the common pin (the one with the dot) is in the right place tho.

TRIGGER INDICATORS

TIP: Don't solder these components in right away if you think you might want to use the dual LCD's
http://www.eight-oh-eight.org/Pics/MB808/DigitalSectionDiagram.jpg
TIP: DON'T SOLDER IN THE Capacitors or any components marked with a red dot on the diagram!

PARTS:

12	1K	Resistor	R442, R445, R448, R451, R454, R457, R460, R466, R469, R472, R475, R478
24	100K	Resistor	R440, R441, R443, R444, R446, R447, R449, R450, R452, R453, R455, R456, R458, R459, R464, R465, R467, R468, R470, R471, R473, R474, R476, R477
12	2N3904	Transistor	T3, T4, T5, T6, T7, T8, T9, T11, T12, T13, T14, T15
12	3mm	LED	BD-SEL, CB-SEL, CH-SEL, CP-SEL, CY-SEL, HT-SEL, LT-SEL, MA-SEL, MT-SEL, OH-SEL, RS-SEL, SD-SEL
12	.375	LED Standoff	BD-SEL, CB-SEL, CH-SEL, CP-SEL, CY-SEL, HT-SEL, LT-SEL, MA-SEL, MT-SEL, OH-SEL, RS-SEL, SD-SEL

1.) Solder in the 100k and 1k resistors. Pay attention to their position, only most of them follow the same pattern.

2.) Solder in *ONE* transistor for each indicator. The second one (along with the cap) is there to make the LED blink brighter.. but it's not a very good circuit, so it's best you simply ignore it. Check above for a pic showing which transistors not to solder in.

3.) Using LED spacers *carefully* solder in each LED so that it is completely straight and true to the PCB. (To ensure perfect alignment, you may want to wait until you have a front panel.)

BASS DRUM

With the BD (and all the other instrument circuits) all you need to do is solder in the components, go by order of their height to make the physical assembly easier. DON'T solder in the pots, trim pots or switches, those will be the last things you solder in. Once you've finished each circuit check for continuity from the +/- 15v rails and ground to be sure there isn't a bridge there. Note that the BOM lists the Op-Amp as a TL072. It will work just fine in this circuit, but the original uses JRC4558's

PARTS:

1	1K	Resistor	R175
3	10K	Resistor	R153, R154, R172
4	100K	Resistor	R159, R163, R174, R176
3	1M	Resistor	R156, R161, R167
1	220R	Resistor	R171
4	22K	Resistor	R152, R160, R165, R168
1	2K7	Resistor	R204
2	4K7	Resistor	R155, R162
2	47K	Resistor	R164, R169
1	470K	Resistor	R170
2	6K8	Resistor	R166, R173
1	8K2	Resistor	R157
1	82K	Resistor	R177
1	220p	Ceramic Capacitor	C48
3	15n	Polyester Capacitor	C40, C41, C42
1	33n	Polyester Capacitor	C39
2	100n	Polyester Capacitor	C38, C45
2	.47u/50	Electrolytic Capacitor	C27, C47
1	1u/50	Electrolytic Capacitor	C50
1	33u/6.3	Electrolytic Capacitor	C43
1	2SA733	PNP Transistor	Q40
5	2SC945	NPN Transistor	Q39, Q41, Q42, Q43, Q44
2	1N4148	Diode	D52, D53
1	8 pin	IC Socket	IC17
INSTALL LATER:			
1	JRC4558	OpAmp	IC17
1	50KB	Pot	BD-TUNE
1	10KB	Pot	BD-TONE
1	100KA	Pot	BD-LEV
1	500KB	Dual Pot	BD-DECAY
OPTIONAL:			
1		SPDT Switch	BD-RT
2	2 Pin	Header	U$21, U$55

q40 is the transistor at the bottom of the circuit *without* the white line through the footprint for the legs.

NOISE

PARTS:

1	100R	Resistor	R282	
1	100K	Resistor	R280	
2	1M	Resistor	R278, R284	
2	22K	Resistor	R279, R285	
1	22p	Ceramic Capacitor	IC31	NOTE: attach to pins 1 & 2 of opamp.
1	18n	Polyester Capacitor	C154	
1	39n	Polyester Capacitor	C153	
1	1u/50	Electrolytic Capacitor	C152	
1	10u/16	Electrolytic Capacitor	R281	NOTE: Changed type of part from resistor to capacitor.
1	47u/16	Electrolytic Capacitor	C151	
1	2SC828	Transistor	U$30	
1	8 pin	IC Socket	IC31	
INSTALL LATER:				
1	JRC4558	OpAmp	IC31	
1	20KB	Trim Pot	CB-TUNE3	

NOTE: I've found some additional notes about the noise section in the 808, apparantly it was revised at some point to fix some problems that occasionally showed up:

Replace R283 with a jumper wire
Replace R280 with a 100K resistor (it was a 300K)
Replace R281 with a 10uF 16v *capacitor*
Solder a 22pF ceramic capacitor between pins 1&2 of the opamp

The noise generator produces both white and pink noise. Three instruments use white noise and three use pink noise.

SNARE DRUM

2	100R	Resistor	R32, R37	NOTE: R32 not clearly labeled.
2	1K	Resistor	R19, R28	
2	10K	Resistor	R9, R14	
2	100K	Resistor	R15, R24	
1	1M	Resistor	R22	
2	15K	Resistor	R30, R33	
1	220R	Resistor	R21	
1	2K2	Resistor	R40	
3	22K	Resistor	R12, R34, R36	
1	2M2	Resistor	R38	
1	27K	Resistor	R39	
1	330R	Resistor	R31	
1	33K	Resistor	R29	
1	4K7	Resistor	R13	
3	47K	Resistor	R23, R25, R35	
1	470K	Resistor	R26	
2	680R	Resistor	R17, R64	
1	8K2	Resistor	R20	
1	820K	Resistor	R16	
1	220p	Ceramic Capacitor	C35	
1	1n	Polyester Capacitor	C49	
2	1.8n	Polyester Capacitor	C51, C52	
1	2.7n	Polyester Capacitor	C53	NOTE: not clearly labeled.
3	6.8n	Polyester Capacitor	C28, C32, C33	
1	10n	Polyester Capacitor	C54	NOTE: not clearly labeled.
1	22n	Polyester Capacitor	C55	
2	27n	Polyester Capacitor	C29, C30	
2	47n	Polyester Capacitor	C31, C34	
1	100n	Polyester Capacitor	C46	
1	.47u/50	Electrolytic Capacitor	C37	
1	33u/6.3	Electrolytic Capacitor	C36	
1	47u/16	Electrolytic Capacitor	C44	
1	2SA733	PNP Transistor	U$2	
4	2SC945	NPN Transistor	U$1, U$4, U$5, U$6	
1	1N4148	Diode	D2	
2	8 pin	IC Socket	IC18, IC19	
INSTALL LATER:				
2	JRC4558	OpAmp	IC18, IC19	
1	10KB	Pot	SD-SNAPPY	
2	100KA	Pot	SD-LEV, SD-TONE	
OPTIONAL:				
1		SPDT Switch	SD-RT	
2	2 Pin	Header	U$57, U$71	

LOW TOM

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/lt.html
http://xlargex.xl.funpic.de/projects/808/schema/808_LT300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/LT_PartNames.jpg

MID TOM

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/mt.html
http://xlargex.xl.funpic.de/projects/808/schema/808_MT300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/MT_PartNames.jpg

HIGH TOM

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/ht.html
http://xlargex.xl.funpic.de/projects/808/schema/808_HT300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/HT_PartNames.jpg

CLAP

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/cp.html
http://xlargex.xl.funpic.de/projects/808/schema/808_CP300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/CP_PartNames.jpg

IMPORTANT NOTE The polarity markings on the PCB for C85 are backwards! Place the negative side where you see the "+" marker!

MARACA

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/ma.html
http://xlargex.xl.funpic.de/projects/808/schema/808_MA300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/MA_PartNames.jpg

RIMSHOT / CLAVE

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/rs.html
http://xlargex.xl.funpic.de/projects/808/schema/808_RS-CL300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/RS_PartNames.jpg

COWBELL

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/cb.html
http://xlargex.xl.funpic.de/projects/808/schema/808_CB300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/CB_PartNames.jpg

2 - 250KB Trim Pot CB-TUNE1, CB-TUNE2
BOM Parts list says 250kb but the part number says 100kb? is it 250kb or 100kb?

note of clarification - C140, the 3.3 nF cap, is next to the electrolytic at C142...

CYMBAL

Cymbal fix 18k: Also note the two 18K resistors soldered onto the board there. These are part of the cymbal section and I'll be adding a note to the assembly instructions about that. The only caveat would be that if you were using a TL072 for the opamp there the resistors need to be left out to keep the opamp stable... The resistors in the cymbal section add some positive feedback to the summing amps at the last stage before the output. When I was testing with TL072's they would oscillate at a high frequency and I really couldn't figure out what the resistors were doing for the design so I cut them. Turns out that the 4558's don't have this problem and the tone will have the wrong balance of tones and be missing some of the attack without them. I still don't understand exactly what is going on, but it seems that the two resistors provide some sort of compensation needed by the 4558's

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/cy.html
http://xlargex.xl.funpic.de/projects/808/schema/808_CY300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/CY_PartNames.jpg

NOTE: Those of you who got PCB's from the first batch should consider adding two 18K resistors as shown in the following picture. I have built some CY's that didn't seem to be affected by this (in particular I found that TL072's tend to oscillate with the resistors present) but other times the CY lost some of its character without them.

http://eight-oh-eight.org/Pics/Corrections/Thumbs/Batch1_CY18KResistors.jpg
http://eight-oh-eight.org/Pics/Corrections/Batch1_CY18KResistors.jpg

On the batch 2 board there are two resistors that are not on the above version (and hence not in the BoM). There is an R394 and a resistor below C179 of which the number is illegible.

Already found it, those are the R107 and R108 from the microLARGE schematics. 18k

the left side bottom of the resistor looks to me like its solder bridged between the two, is that correct?

yup, that is correct

HIHAT

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/hh.html
http://xlargex.xl.funpic.de/projects/808/schema/808_HH300dpi.jpg
http://www.eight-oh-eight.org/Individual_Circuits/Pics/HH_PartNames.jpg

Batch one: There is a missing bit of trace on the PCB (apparently Eagle can do this sometimes.. there is another unused trace in the Core section like this). Looking at the top side of the PCB the middle pin of the 3-pin header at the bottom of the circuit should be connected to that resistor directly above it. Make a little solder bridge :0) Both Hats can sound simultaneously

http://a203.ac-images.myspacecdn.com/images01/21/l_1b8dd3dac2de21c041472b4348435c0a.jpg

Main Outputs

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/mainouts.html

1.) Solder in the resistors EXCEPT R489-R499. These are where each instrument connects to the main outputs and it's easier to test by connecting them one at a time.

2.) Solder in the 2 diodes

3.) Solder in the IC socket

4.) Solder in the capacitors

5.) Solder in the transistors EXCEPT the one whose pins are not in a straight line. The silkscreen is not legible for it but this is a 2SK30 which is not on the BOM. It's part of the circuitry which mutes the outputs briefly when the power is turned on so that you don't hear a thump at the outputs.. however, I've not been able to get it to work..

NOTE: I borrowed an idea from the x0xb0x and used a different silkscreen or footprint for each transistor, but have yet to let anybody else know which is which.. I assume I explained this somewhere, as people have built their kits...

Here is an image which explains the markings, remember that the third transistor (the one with the 3904-like footprint) should be left out of this circuit.. it's supposed to be a 2SK30A FET transistor which helps in briefly muting the outputs when the unit is first turned on (the BD, SD and sometimes the toms will trigger).. however, regardless of how I'm worked with it.. it breaks the main outs..

http://www.eight-oh-eight.org/Pics/TransistorSilkscreen.gif

1 = unused
2 = 733
3 = 945
4 = 733

It's the white line through the symbol that matters. 733's don't have one, 3904 have a thin one, 945 have a thick one.

Why not all resistors on main out have same value? I used to see same values on similar simple mixer circuits.
Is it because not all voices have same output level or some have different impedance?

The levels differ. The real 808 also uses different resistors for different instruments.

MAIN-IN, MAIN-IN1 on the pcb are in fact OUT, not IN, right?

The MAIN-IN really *is* an input - wire each of the voices to the switched connector of the output jacks, then chain all the jacks together and wire them to the lower pin of MAIN-IN. That's how you end

up with the individual voice being removed from the main outs when you plug in a cable...

The two pin MAIN-IN1 is only on the second batch of boards. On the first batch, it was the three pin header labeled MAIN-OUT...

Accent

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/accent.html
http://www.eight-oh-eight.org/Individual_Circuits/Pics/Accent_PartNames.jpg

Are the .47u caps supposed to be 16v or 50v?
Anything higher will be fine

STAGE 2 OF ASSEMBLY

Now that you have all the Resistors, Capacitors, Diodes, Transistors and IC sockets soldered in for each circuit you should take a break, there is still a lot left to be done so it's best you're not feeling rushed.

If you haven't been cleaning your flux all along now is the time to do it. This is why none of the switches, trim pots and pots should be soldered in yet because they won't fare so well during the cleaning process. If you've been using organic flux solder (good idea!) all you need to do is get a brush (I use an old tooth brush) and rinse the board with warm water (no soap!) while brushing the flux off. If you used rosin core or whatever else you're going to need some cleaning agent too, I've used isopropyl alcohol but the 70% stuff you'll get at the drug store will only do so much good. You can get 99+% isopropyl at some places (alliedelec.com has it) but if you're not doing a lot of DIY then it's probably better to just do the extra scrubbing.

Once the flux is removed pat the board down with a towel and then set it somewhere to dry. If you've got a can of air around (or an air compressor) use that to blow the water off the board and out of the IC sockets. Otherwise just make sure it will get dry and give it plenty of time to do so.

http://nuxx.net/wiki/Flux_Removal

PARTS PREPARATION

Switches

Batch one: This step only applies to the first set of PCB's made which have the traces leading to the LED's inside the switches mixed up. To fix this you need to pop the cap of the switch and flip the LED around. Using the E-Switch switch this is a simple process and it doesn't seem to harm them at all

=

Potentiometers

To help things mount onto the front panel smoothly, I'd like to make a couple suggestions before you solder the potentiometers onto the pcb:

1) There is a little "nub" on the single pots, right above the solder terminals. This should be filed off so that the front panel will sit a little lower, allowing the switches to stick up a little higher relative to the panel. (If you're patient enough to wait for front panels from me, you won't have to do this – my panels have a slot machined in them for this tab.)

2) The dual pots don't stick up as far from the pcb as the single pots, and the threads are shorter, so I'd suggest getting a piece of non-conductive material, either a rectangle of .4" by .5", or a circle of .4" diameter, .125" thick, to go under each dual pot so that it sits up about the same height as the single pots. (Only necessary if you're going to use washers and nuts on the potentiometers to hold the pcb to the front panel.)

3) It's easier to trim the dual pot shafts to the same length as the single pot shafts before they are soldered in. The single pots are 1.175" tall (measuring from the top of the pcb) and the dual pots stick up 1.350" with the .125" spacer underneath, so I trimmed .175" off the shafts of my pots. (It's probably best if you actually assemble the pieces to get an accurate measurement before cutting.)

=

Now that most of the components have been soldered in and the PCB has been cleaned of the excess flux it is time to switch gears a bit. From here on out you won't be needing to sort through resistor and capacitor values or be a slave to the soldering iron (well.. there is a bit more soldering to do). What you will need most now is a spacious, organized and safe workplace. Remember that the PSU on this kit connects DIRECTLY to the wall, unlike the x0xb0x which used an external transformer. This means that the potential for harm from electrocution is MUCH greater now and you should not be working on this in an area where children or pets can get to easily. For the same reasons you should keep the work area clean and clear of excess material, esp. metals which can short out exposed leads.

Some recommended materials:
Bench clamps, as many as you can get your hands on. Not so much to hold the PCB but to hold the bits of the PSU connector (take a look at my pics) and to keep test wires out of the way.

-Digital Volt Meter (DVM). You really need one of these. I suppose you could work without it if you're never going to do another project in your life.. but the amount of extra trouble you're going to go through when troubleshooting may be obscene. A cheap one will do.

-Headphones. Cheap ones you don't care about, these will take the place of an oscilloscope for those who don't have access to one. If you have some kind of mixer I would use that too, many of the circuits are not able to drive a set of headphones from the areas you want to test and the sound will be badly distorted because of this.. still though, I was able to get useful information about a circuit with just a pair of cheapo headphones.

-Test leads. With alligator clips on the end. These come in useful over and over again.

Sequencer

If have one of the 15 kits from batch one be sure that you have flipped the LED's in your switches before you solder them in. Once that's done go ahead and solder the switches to the PCB and then place the IC's in their sockets as follows:

Looking at the component side of the PCB the first row of seven sockets should hold the 7 74HC595's, each with the notch (pin one) to the left side of the board.

The next row of sockets hold the resistor IC's that set the output current for each pin of the 595's. You have two different values here
5x 220 Ohm
2x 100 Ohm

They go in order as follows (again with the notch to the left side of the board)

220, 220, 220, 100, 220, 200, 100

The last row of 7 sockets hold the 74HC165 IC's. Again the notch faces the left side.

NOTE: Now is a good time to double check that the SIL Resistor IC's to either side of each 74HC165 are indeed soldered in correctly.

Now is also a good time to double check that the 5v power is functioning correctly.

Carefully place the Microcontroller into the large 40 pin socket at the bottom right side of the board, notch facing to the left again. It should not take a whole lot of pressure to get it in, and be careful about bending pins.

Above the MCU is an 8 pin socket which holds the 6N138, notch facing the left.

Below the MCU there is an 8 pin socket which holds the memory IC (the part number escapes me at the moment)
IT GOES IN WITH THE NOTCH FACING TO THE RIGHT!! It was a matter of keeping a cleaner PCB trace that made this one IC different from all the rest.

Check the 2 encoders that you've got. If you ordered from my Mouser list you've got 2 different encoders with a different number of clicks per turn. I recommend using the one with fewer detents on the left hand side of the board for the Instrument select encoder and the other for the pitch control.

The "Swing" pot is a 10KB.

Now that you have all this done go ahead and power it up. You should see LED's flash briefly and then a few second pause while MIOS boots up the sequencer application. After that you should see a standard 4 kick drum beat displayed on the GP buttons and pressing play/stop should have the desired effect. While the sequencer is playing you should see the trigger LED's light up when their instrument is being triggered. If you want to add more information to the pattern it works like this:

The leftmost menu button is the works as the instrument select button while you are in pattern edit mode. It is programmed to be a momentary switch so it is only activated while you are holding it down. Once it is activated the GP buttons become shortcuts to selecting the instrument track you wish to edit. They go in the following order:

AC, BD, SD, LT, MT, HT, CP, MA, RS, CB, CY, OH, CH

Add some data to each track to be sure each trigger LED lights up.

To save this pattern press the rightmost menu button. While in pattern edit mode this will put you in the save menu. While in the save menu the first 8 GP buttons display the bank and the second 8 display the pattern. You should see steps 1 and 9 light up meaning that your currently in the first bank and first pattern. Pressing the select button will confirm that you want to save here. Depending on the exact firmware your PIC shipped with you may occasionally hear a glitch when you save. This is normal as was due to a coding error on my part.

If you've been able to do all of this.. congrats! Your sequencer is working!

Accent

Before we get to the test procedure I'll explain a little about how the Accent and triggering work. The accent signal is connected in parallel to each instrument, which means that they all share the exact same signal on pretty much the exact same copper trace. Remember that the 808 only has a global accent feature in the sequencer which is why it is possible for each instrument to use the same accent line. Now, you may be thinking that the accent signal is only present when the accent step is activated, but this is not the case. The accent signal is present on *every* step that is activated, but on steps where the Accent is activated the accent signal is modified to increase its voltage.

Each instrument uses a simple AND gate to send the trigger signal to the tone generator. This means that there needs to be two positive signals at the input of the gate to send the trigger along. One of these signals will be coming from the accent line, which is present on every activated step and will have varying voltage depending on the setting of the accent knob and whether or not the current step had the Accent activated. The other signal will come from one of the 74HC595's in the sequencer section. This signal will only be present when a step on that instruments track is activated. Make sense?

This means, however, that without the Accent circuit working properly none of the instruments will trigger, so that is why we are doing it first.

So: solder in a 10KB pot in.

Now, power it up and check that, looking at the panel side of the PCB, the leftmost pin on the pot reads 5v, the rightmost pin reads 15v and the middle pin sweeps between those two voltages.

If you didn't already add some accent data to the pattern you were using to test the trigger LED's do that now.

The simplest way to test the accent line for the proper signal is to touch the headphones to it (like you may have done on the x0xb0x). You can use the leftmost pin of the 2 pin header at the bottom of the BD section to do this (facing the panel side of the PCB). With the accent knob turned all the way to the left you should hear a short pulse. The more you turn up the accent the louder the pulse should become.. but this is a very unscientific means of testing this so I would not try to get too specific as to how much louder or even exactly what "louder" will sound like..

If you have a scope what you are looking for is a 1ms pulse occurring on every step that is activated and varying in voltage from 5v to 15v correlating to the setting of the Accent Knob.

Troubleshooting: First: remove the PIC from the socket to protect it from damage, and remove the accent pot and the accent LED. This will remove any relationship between the 5v rail and the accent circuit so we can figure out if the problem is in the sequencer section or the accent section.

Also, you've mentioned that the 7805 itself has the proper 5v appearing at its pins.. but on the 5v header you only see the ~1.5v. This is unusual! Can you verify this for me and also check the value and polarity of:

C22: 2200uF

C112: 100n

Unfortunately what I think you're dealing with is a small solder bridge somewhere in the digital area of the PCB. This means you're going to need to get out a magnifying glass and find where there is a pad with some messy solder causing the problem. Have you tested for continuity between the 5v rail and the ground plane?

While you're at it: triple check the values of all the resistors in the accent circuit and also make sure the SIL resistors attached to the DIN IC's are in the correct way.

Bass Drum

What we have tested so far is that the sequencer operating correctly and therefore sending out good looking pulses on all the instrument trigger lines and that the accent line has been doing the same. As mentioned before this is so the AND gates at the front of each instrument can do their job and trigger the tone generators.

The most common element of the tone generators in the 808 is the "Bridged Twin-T" circuit, which uses resistors and capacitors in the feedback loop of an op amp. By using specific ratios of values for the Rs and Cs the opamp can be made prone to oscillation, not unlike a self oscillating VCF. The important part of the comparison to such a VCF is the ability to use a short pulse to "ring" a filter that is on the edge of oscillation... just like the pulses generated by the sequencer & accent.

The BD is a good place to start because it consists of a single oscillator and the exact reasons the controls work don't need to be understood to troubleshoot problems.

So, solder in a 100K Audio taper pot (104A) for the BD-LEV,
10K Linear (10KB) for the BD-TONE
50K Linear (503B) for the BD-TUNE
500K Linear Dual (B500K) for BD-DECAY

The switch BD-RT is not a part of the standard kit.

Insert a 4558 opamp

Power it on and (assuming again that you made yourself a test pattern) start the sequencer while listening to the middle pad of the BD-RT switch... Do you hear a sound that is affected by each of the knobs?

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_BD_Top.gif

What you need to check for is that at point #1 you hear (or see on the scope) a pulse that varies in volume according to the accent signal and that at points #2 you hear an oscillation that is affected by the BD-Tune knob. The remedy for not finding the right signal is simple... double and triple check the components that are soldered in and that they are soldered in well. I've built these circuits many times now and I still make mistakes.. so it's best to assume that you've made a few too.

I've compared the xlargex parts list with the individual mb-808 lists as well as the mouser list. According to the xlargex info, the original BD tune was a 47k resistor. It looks like the Moogah schematic (and parts list) has a 10K pot in series with a 22k resistor, which doesn't actually get up to the 47k of the original. The xlargex mod replaces the 47k resistor with a 100ka pot, so it seems reasonable that we use a 50k pot in series with the 22k resistor. (We could also use a 100k pot and use a piece of wire instead of the 22k resistor to increase the tuning range...)

Troubleshooting: my BD Tone pot have no effect on the sound at all, sounds as if the pot was turned all the way to the right so, what should I check on this one? tune and decay works fine.

check that R171 = 220R
R172 = 10K
C45 = 100n
C47 = .47uF

(BD fixed, I had a 0.01uf cap instead of 0.1uf on C45)

Noise

Since many of the instruments require the noise source, it's best we get it working now.

The noise source is located under the SD section to the right of the BD-Decay knob. It consists of a single op amp, a trim pot, a transistor and a few R's and C's

First solder in the 20KB trim pot, which is labeled "CB-TUNE3" Due to an oversight on my part, then insert a 4558 in the socket.

The noise source works by amplifying the voltage noise that naturally occurs in a transistor. The sound of the noise will vary greatly by different types of transistor and some are clearly better than others in this circuit. Looking at the PCB the left side of the OpAmp is amplifying the (very weak) signal from the transistor and outputting that to the SD and CP. The right side of the OpAmp is configured as a simple low pass filter to create the pink noise for the Tom's

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_Noise_Top.gif

Now, with the unit powered up and looking at the panel side of the PCB you should hear white noise at point #1 and pink noise at point #2

Snare Drum

Now that you've gotten the BD working (and hopefully a little bit of satisfaction finally) we can get the SD ready. Whereas the BD consisted of a simple Twin-T oscillator which was connected to the output through a transistor.

The SD consists of 2 oscillators and a very simple envelope control connected to the noise source. The 2 oscillators are mixed together at the SD-Tone knob and the resulting sound is mixed with the noise at the output amplifier.

Solder in a 100K Audio (104A) pot in for SD-Lev and SD-Tone
Solder in a 10K linear pot (10KB) for SD-Snappy

Insert 2 4558 op amps into their sockets.

Power it up and listen at point #4 while playing your test pattern (if you still haven't made one.. well.. do it now Smile

If things are not all gravy, here is what to check:

Can you hear the trigger pulse at point #5

At points #1 and #2 can you hear 2 different tones, with #1 being lower in pitch?

At point #3 you should hear the output signal very much the same as point #4

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_SD_Top.gif

Low Tom

The 3 Tom circuits all share a similar layout to the BD. They both use one OpAmp as an oscillator and another OpAmp to control the decay time. Unlike the BD, however, the toms use a third opamp as an output amplifier.

Solder in a 100K Audio pot for LT-LEV and LT-RVB
Solder in a 500 Ohm (or 5K, or 2k, etc) pot for LT-Tune
Solder in a Dual 1M pot for LT-Decay

Solder in the DPDT switch WITH PATIENCE AND CARE! Getting the switches in flush and even can be tricky business.. and I know from experience that getting the switches out is *very* difficult if they are soldered in crooked.

Here is what I recommend, first hold the switch in place on the PCB and take a long hard look at it from various angles, get an idea of where the play is and what could go wrong. Next, carefully apply a *small* amount of solder to a single pin and then re-check that the switch is seated properly. Repeat this process for each pin, and don't take it for granted.. even I screwed this up a couple times..

Now insert the TL074 quad opamp into the socket. This is one of the few areas where I did not strictly follow the original schematics. I dare any of you to find a difference in the sound here though Wink Also, I need to give credit to Sebastian Lindstat (http://xlargex.xl.funpic.de/) for this modification.

Starting with this instrument I'm going to try a different type of illustration, please give me some feedback on which is the most useful.

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_LT_Top.gif

The PINK Traces carry the Pink Noise signal through the circuit
The RED Traces carry the oscillation signal from the oscillator opamp and the decay control opamp.
The GREEN Traces carry the trigger pulse to the envelope on the noise source
The ORANGE Traces carry the signal to the output amp
The BLUE Traces carry the signal to the output bus
The YELLOW Traces carry the trigger pulse

(Repeat above for Mid and High Toms)

Clap

Up until now we have been dealing with fairly simple circuits and testing them has been mostly a matter of whether they work properly or simply don't work at all. The CP is different, this is one of 2 circuits that I've always needed to give some special attention to before it behaved properly.

The CP sound is comprised of two different white noise sources each with its own envelope. One of the envelopes is rather simple, a medium attack with a longish decay that creates the reverb sound, the other envelope is an interesting beast. It is made up of a series of short saw tooth shaped peaks which create the "clap" sound. It is this envelope which usually is out of whack and has a major effect on the sound.

To finish the assembly: solder in the 10K CB-OFFSET trim pot and the 100K CB-Lev pot. Don't forget to put the LM399 comparator and 4558 opamp in their sockets.

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_CP_Top.gif

You should be able to hear both sounds on the PINK traces. If there is no reverb check that white noise is coming on the GREY traces and that there is reverb like sounds coming from the ORANGE traces. If you hear nothing at all on the PINK traces check that trigger pulses are coming in on the YELLOW traces. If you hear reverb but the clap part of the sound is not proper first adjust the offset pot to see if it gets better. If the trim pot doesn't help then you need to very carefully check all the components around the LM399 comparator... sorry to be so brief but to get in more depth really requires an oscilloscope! Triple check each component value and be absolutely sure each solder joint is well done.. after that let me know what you hear and I'll try to extrapolate what the problem could be..

Troubleshooting: If you have a scope: take a look at the light blue traces, they should have that envelope shape (3 short toothy look peaks and a 4th longer peak). In one case I was able to get the clap working by replacing C85 with a different brand of capacitor.. but this doesn't really make any sense..

I think I have the same problem as MP. The clap is just a burst of noise without the characteristic envelop.

I only have an old analog scope, so capturing a slow changing voltage like the envelop is pretty tricky, but I think I've managed it, and it looks odd.... It's just a pulse train with the length of the trigger. There's a bit of decay at the end which gives you some of the reverb sound, but the triple trigger isn't there.

http://www.dompselaar.org/MB808/clapenv.jpg

Ahh! Excellent pic! Now, can you tell how long it is between the start of the pulses and when the final slow envelope begins? It should be 30ms, and in which case your envelope is generating *many* extra spikes.. but I suspect in your case this is much shorter? (so what you end up hearing is just a little bit of the end piece there and then no dynamic after that..). While I was tweaking the last board I have here I found that C85 was discharging much faster than the typical rules for RC constants dictate.. which is unusual.. so I swapped C85 for another capacitor of the same size from a different manufacturer and then the envelope shape was proper.. however this would be a rather unique situation.. and I wonder if

the trouble wasn't more related to a cold solder joint around C85.

NOTE: I've found something, and it's not in the CP section, but instead in the noise generator...

It turns out on later models of the 808 the noise generator was modified a bit to remove DC offset in the noise signal.. this could def. cause issues with the CP sound!

Here are the changes:

Replace R283 with a jumper wire
Replace R280 with a 100K resistor (it was a 300K)
Replace R281 with a 10uF 16v *capacitor* (component sex change Razz)
Solder a 22pF ceramic capacitor between pins 1&2 of the opamp

This made quite a difference in the sound of mine here, the CP and SD both became brighter and more crisp.. try it out!

That does make it as bright as I know the 808 clap.

Now I only need to find why it retriggers so fast...

As the whole envelop circuit sits below ground and C85 is charged with -15V, I thought it more logical that its polarity should be reversed, and lo and behold, I get the normal amount of triggers. There still is something else wrong, as the noise still doesn't follow the envelop, but at least the envelop looks okay now.

DING!

That makes quite a bit of sense! Your right that the voltages in the envelope have a baseline of -15V and go up from there to below 0v to create the spikes, and as such the positive side of C85 should be connected to ground. On the PCB this is reversed (and on the original schematics as well!). It is possible that the other caps I have here that "fixed" the problem are bi-polar.. which makes a much neater explanation of the situation.

SO, everybody else: switch the way C85 is soldered in!

I have just reverb on the handclap too, and the trim pot does not change anything at all, the thing is, the mods on the noise generator are already there and working I think -the snare drum sounds like it should- , so... "carefully check all the components around the LM399 comparator", can somebody elaborate more on this please? and btw. this could affect the maracas sound too?(having a faint click, no maraca sound at all)

I don't think the MA and CP issues are related.

Try a few easy things first: turn up the white noise trim and adjust the CP-Offset trim and see if you can hear any difference.

If that does not make a difference then check for the noise signal at pin 1 of IC21 (the 4558). If you can hear the noise signal there then the problem may be that the envelope is not triggering the VCA (the

CA3080).

At this point it gets tricky without an oscilloscope... So your best bet to start with is to double check all the components around the 399 for the correct value and good soldering.

I can hear the noise signal at pin 1 of the 4558 fine (replaced the chip just in case).

still, just the reverb sound... dead CA3080 maybe?

btw, some of the resistors aren't the exact value(same thing the BOM), e.g.
5k6 =5k62
330R = 332R
82k = 82k5

and finally... *drum roll here *...
It was the CA3080 itself!!!, replaced it with a new one and my mb808 handclap is working now

update:

It was all right finally or so I thought, after a some chatting with Jeff, I found out the reverb delay bug on my box is still there, how to reproduce it/find out if you have it too:

Start a pattern with some CP hits, the first step (some times more than one step, up to four in my case) don't have any reverb on it, some hits later the CP reverb starts to sound fine.

Stop the pattern and wait some time, go play it again and the bug comes back.

It sounds like it has something to do with C14 (original schematic). Just like it needs more than one CP trigger to get charged and once charged stays at a certain level until it is stopped for too long.

Moogah: I've been tracing this problem on one of the units I have here, and what I've found is exactly what Etaoin describes (except I think he meant C76, which is the 47n green cap near the 2 1M resistors). Watching this cap on the scope, the thing that really bugs me is that the voltage settles to -10mV (?!) and I can't figure out why it's able to settle to a point below ground.. and we know from basic transistor theory that the 945 which creates the envelope here won't turn on until the voltage is above 60mV. When I worked out the RC time constant equation for this it made perfect sense that the cap needed a bit of "pre-charging" before there would be any appreciable envelope voltage above the 60mV cutoff point. My thoughts about this are: either the transistor should be biased closer to the 60mV turn on point, or there is a problem with the ground plane in this area which is causing the voltage to be lower than it should be....

Perhaps this problem is solved easily by reducing the 1M resistor which the trigger pulse comes in through, this would allow the cap to charge faster without affecting the discharge rate (I think..)

By the way, I just came across a post on the Dutch synth forum where someone mentions (about a real TR808) "you shouldn't hear the reverb on the first clap, only on the subsequent ones" and someone else agrees. So at least some TR808's exhibit the same behavior.

MOD: By the way, if you exchange R148 for a pot, you can change the "pitch" of the clap. According to the TR808 service manual , IC21 and the components around it, is a band pass filter. When the resistance of R148 changes the cutoff of the filter change.

Indeed! This is a nice easy mod to do, and actually has a decent effect on the sound. microLARGE included this on his PCB's Also, try this: Adjust the white noise trim (CB-TUNE3) while the clap and snare are playing.. some pretty gnarly sounds can be had!

Maraca

(nothing on the forum)

Rimshot/Clave

The sound generators for the RS and CL are essentially two separate circuits (joined at the final opamp output stage) which are switched between by the RS/CL switch. The RS is created by a few transistors configured as a simple VCA which distorts the sound. The CL sound is created by two separate oscillators which are mixed together at pin 1 of IC25.

To finish the assembly solder in the RS/CL switch *carefully*. It is important that this switch be soldered in flat to the PCB and it is *very* difficult to de-solder if a mistake is made. First, solder in a single pin with as little solder as possible and TRIPLE check the positioning before soldering in another pin. Solder in a 100KA (104A) pot for the RS/CL-LEV control

Don't forget to put 4558s in the sockets.

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_RS_Top.gif

If you hear nothing at all (or just a faint pulse) check the components around the YELLOW (trigger pulse) and ORANGE (output opamp) traces. This is the final output stage.
If everything looks good there then check that there is a tone at the BLUE and RED traces. You should hear the separate oscillators for the CL there. If you can hear these tones, but with the RS/CL switch down (CL position) then you need to double check all the components in the output opamp. If you still don't see a problem check all the components around R125-129, this is where the RS sound is created and also where the RS and CL circuits connect.

Cowbell

The CB circuit uses a Schmitt Trigger IC (IC30) to create two square waves a which are then mixed together and a simple envelope applied to it.

There are two 100K (250K if you have them) trim pots which control the frequency of the square waves, solder these in now.

Solder in a 100KA pot for CB-LEV

Don't forget to put a 4558 and a 4584 in the sockets.

http://www.eight-oh-eight.org/Individual_Circuits/Pics/808_v2_CB_Top.gif

Listen to the blue trace and adjust the top trim pot, you should hear it change pitch. Same for the red trace and the bottom trim pot. If they both make sound listen to the output (orange) traces. The two sounds are mixed and the envelope applied at the top of the circuit.

Troubleshooting: while testing the hand clap on a real TR808 found out that my mb808 cow bell pitch is too low compared to the real thing, and envelope is too fast too.

I used 250k trimpots found by subatomic on Mouser :

http://www.mouser.com/search/ProductDetail.aspx?R=25PR250KLFvirtualkey57700000virtualkey858-25PR250KLF

cow bell sound comparison mp3, careful with the volume levels!!!:

http://hakkenmusic.googlepages.com/CB-comparison.mp3

First sample is from a friend's TR808

Second sample is from machines-hyperreal

Third sample(w/DB) is from CB tune mp3 found in the mb808 front page

Fourth sample is from my mb808

any clues to fix this?

Moogah: What I hear is a problem with the envelope combined with being out of tune. The 250K pots are the correct size, but they also mean that it will take more careful adjustment to get in tune right.

Check these components which are responsible for converting the trigger and accent signals into the envelope control:

R224: 4K7

R241 & 242: 22K
R243: 10K
R260: 1K
C134: .47uF Check polarity too
R219: 33K

The problem could also be in the mixing of the two frequencies from the Schmitt trigger to create the sound:

R244 & 245: 100K
R250 & 251: 10K
R252: 2K2

sinesurfer: If I short R266(100k) with R270(121k) -on the leads near to the cb tune 2 pot, pin 3- the envelope works fine!!!?
pitch is a little bit low though.

evilx: my envelope sounds too short too....did you guys ever fix this? also what should the tuning be? the accent seems to really drive the hell out of the envelope...with 0 accent the envelope is very short...at full accent it's reasonable

MP: I have also problem with too short cowbell. I temporarily solved the problem by soldering a 0.33 nF electrolytic capacitor in parallel with c135, but there must be a better way to fix it.

I just remember that I change one of the resistors R263 or R266 to get the tuning right. If I understand it correctly those resistors and the cowbell trimmers control the tuning of the oscillators. According to the TR808 service manual one of the oscillators should be tuned to 540 Hz and the other 800 Hz. I haven´t for tried to fix the cowbell for some months after I accidentally put 15 Vac on the accent pin, resulting in some smoke. Smile

TR808 service manual:
http://www.matrixsynth.com/blog/media/roland/TR-808%20Service%20Manual/

I have also noticed that. The samples of the TR808 cowbell I listened to are tuned higher. According to synthsecrets in S.O.S the frequency should be something like 587 and 845 Hz.

http://www.soundonsound.com/sos/Sep02/articles/synthsecrets09.asp

Main Outs

The first thing to do is check that you have stuffed the correct components according to this BOM:

http://www.eight-oh-eight.org/Individual_Circuits/BOMs/mainouts.html

At one point there were a bunch of 1m resistors listed there.. this is not correct.

Also note that the FET transistor (below the opamp, has the pins in a triangle footprint) should be left off. At some point I hope someone can point out my stupid mistake that causes that transistor to break the main out section.... It is there to provide the brief muting of the main outs when the unit is first turned on.

Now, we need to connect each of the individual instruments to the outputs. This is done on the PCB through the "*-RT" switches at the top of each instrument. The middle pin of each switch is connected to the output of each instrument and the top pin is connected to the input of the main out. Following the middle pin of each switch back you will also find a 2 pin header which I think is best used to connect the individual output for each instrument.

In order to get the sound to the main outs I used the lead clippings from the resistors to run a little jumper between the middle and top pins of each switch, soldering them in in such a way that they are easy to remove later. I would recommend doing this to one instrument at a time and then powering on to check the result (remember to put the opamp in..).

. that should be it for the main outs.. it's a very simple circuit so if it doesn't work at all just check that everything is well soldered into place

On the first batch of kits there is a ground pad missing on the 3 pin header at the main outs. The middle pin carries the signal, each of the two outer pins should be connected to ground. Just scrape off some of the solder mask on the component side and solder a little bridge there once the header is in place and you'll be fine..

The "Main-In" 2 pin header doesn't do anything right now Smile

And once you get this thing working, I really wanna hear some samples! How did the CP, CB and CY sections come out? These have always taken me some time to get sounding just right, and I'm not sure that I've got the CB nailed just yet (we need to get some of the original IC's..)

Also, in the CY section if you seem to be lacking some punch on the attack there are two resistors missing on the layout, both 18k. Stay tuned.

Looking at the component side of the board you can see that the PCB on the lower third is darker than the top (also notice that the traces stand out more on the bottom). This is because on the top (Analog) side of the PCB everywhere there is no trace there is extra copper left on the PCB (and on the bottom where there is no trace the copper has been etched away). All this extra copper is the ground plane, and anywhere a pin should be connected to ground, if you look closely, you will see that there are 4 little traces in a + pattern connecting to the ground plane. This is true for the digital section too, but it's on

the panel side of the board (go ahead and look). So.. why does the copper look green? Because anywhere you will not be soldering is covered in "Stop Mask" (Solder Mask). Whereas bare copper attracts solder, stop mask.. well.. stops it. This helps prevent solder bridges and generally makes the PCB easier to work with. Now.. stop mask is just a coating and it really does not affect the PCB if you scratch away some of it (I used a razor) now you can create the solder bridge that the stop mask prevented and connect a pin to ground with a little blob of solder.

On batch 2, the main out is actually labeled "MAIN-IN1", right?

Correct, and it also only has 2 pins and one is properly connected to ground, no scratching needed

APPENDICES

How to Order Parts from Mouser

- Go to Mouser.com
- Login or create a new account.
- Click on "My Mouser" near the top of the page.
- Click on "Import a Bill of Materials" in the left column.
- Copy and paste the parts list into the text entry box.
- Click "Import BOM" above the text entry box. (Choose a different project if desired.)

Firmware

http://miosstudio.midibox.org/
http://ucapps.de/mios_download.html.

Current Firmware
http://www.ucapps.de/midibox_808.html

Old Seq2 Firmware
http://eight-oh-eight.org/phpbb2/viewtopic.php?t=157
http://eight-oh-eight.org/firmware

Old Seq3 Firmware
http://eight-oh-eight.org/firmware