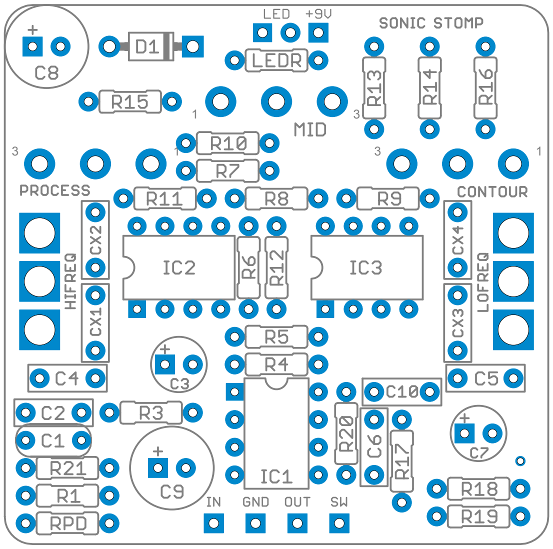
Lumin Equalizer

BBE Sonic Maximizer / Sonic Stomp

Overview



The Lumin Equalizer is a work-alike of the BBE Sonic Stomp, the single-channel stompbox version of the rackmounted Sonic Maximizer first released in the late 1980s.

The original unit uses a proprietary analog IC that contains basically the entire circuit. This is a reverse-engineered expansion of that chip, but it should be stressed that it is not a clone. (...Especially because the chip implementation is patented.)

The legendary “BBE process” is essentially a state variable filter, where the signal is split into three parts (50 Hz and below, 10 kHz and above, and the midrange between) that can be affected differently before being mixed back together. The idea is that it phase-corrects the frequencies before they are amplified and sent to the speakers, for optimal speaker performance.

Controls & Usage

- **Contour** adjusts the phase response of the split signal below 50 Hz (lowpass).
- **Process** adjusts the phase response of the split signal above 10 kHz (high pass).
- **Midrange** adjusts the phase response of the split signal between 50 Hz and 10 kHz (bandpass).
- **Hi Frequency** is a switch that allows you to change the frequency of the “Process” knob.
- **Low Frequency** is a switch that allows you to change the frequency of the “Contour” knob.

Modifications

Some have criticized the cutoff frequencies in the stock circuit as not being the most appropriate for guitar. Accordingly, I added switches for both the Contour and Process knobs so you can set which frequencies they control. You may find that you like having different settings for different guitar and amp combinations.

The stock circuit uses a **3n3** capacitor for the Process (high) knob, which is around 10 kHz. Since that’s already really high for guitar, you probably don’t want to add any settings to make this frequency higher. Try using **3n3** for **C4**, a **1n-1n5** capacitor for **CX1** and a **2n2-3n3** capacitor for **CX2**. This will give three settings on the switch: **3n3**, **4n3/4n8** and **5n5/6n6**.

The stock circuit uses a **47n** capacitor for the Contour (low) knob, which is around 50 Hz. While it may be useful to go lower, like with a 5-string bass, you probably only want to adjust this upwards with 50 Hz as the minimum setting. Accordingly, try using **22n** for **C5**, and then use **10n** for **CX3** and **22n** for **CX4**, which gives you three settings on the switch: **22n**, **32n** and **44n**. If you’d like to go down in frequency as well as up, try a **33n** for **C5**, **10n** for **CX3** and **22n** for **CX4**.

The Lumin uses dual op-amps everywhere instead of utilizing quads like many other Sonic Stomp projects. This gives you a much wider range of choices if you’d like to experiment with something other than the TL072. You could try higher-fidelity op amps such as the **NE5532** or **OPA2134**—just keep in mind that these also have much higher current draws than the TL072!

Parts

Resistors

R1	1k
R3	1M
R4	1k
R5	1k
R6	1M
R7	22k
R8	22k
R9	22k
R10	10k
R11	22k
R12	22k
R13	22k
R14	22k
R15	15k [1]
R16	10k
R17	56k
R18	100k
R19	1k
R20	47k
R21	47k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	100pF MLCC
C2	4n7
C3	10uF electro
C4	3n3 [2]
C5	22n [3]
C6	100pF MLCC
C7	10uF electro
C8	100uF electro
C9	47-100uF electro
C10	100n [5]
CX1	1n5 [2]
CX2	3n3 [2]
CX3	10n [3]
CX4	22n [3]

Semiconductors

IC1–IC3	TL072
D1	1N4002
LED	5mm LED

Potentiometers

Contour	50kB
Process	50kB
Midrange	50kB [1]

Other

High Freq	SPDT center off [2]
Low Freq	SPDT center off [3]

Build Notes

[1] **Midrange knob:** The original unit only has two knobs, with the midrange (bandpass) portion being fixed with a 39k resistor (R15). By putting a 15k resistor in R15 and adding a 50kB pot in series, we can adjust the midrange, with the center position of the pot being the exact stock position. If you do want to leave it off, use 39k for R15 and jumper lugs 2 and 3 of the pot's pads.

[2] **High frequency switch:**These parts form the frequency selector for the Process (high) knob. I recommend socketing **C4**, **CX1** and **CX2** and experimenting to find the frequencies that sound best to you. See page 1 for ideas. To omit this control, just leave off CX1, CX2 and the High Freq switch.

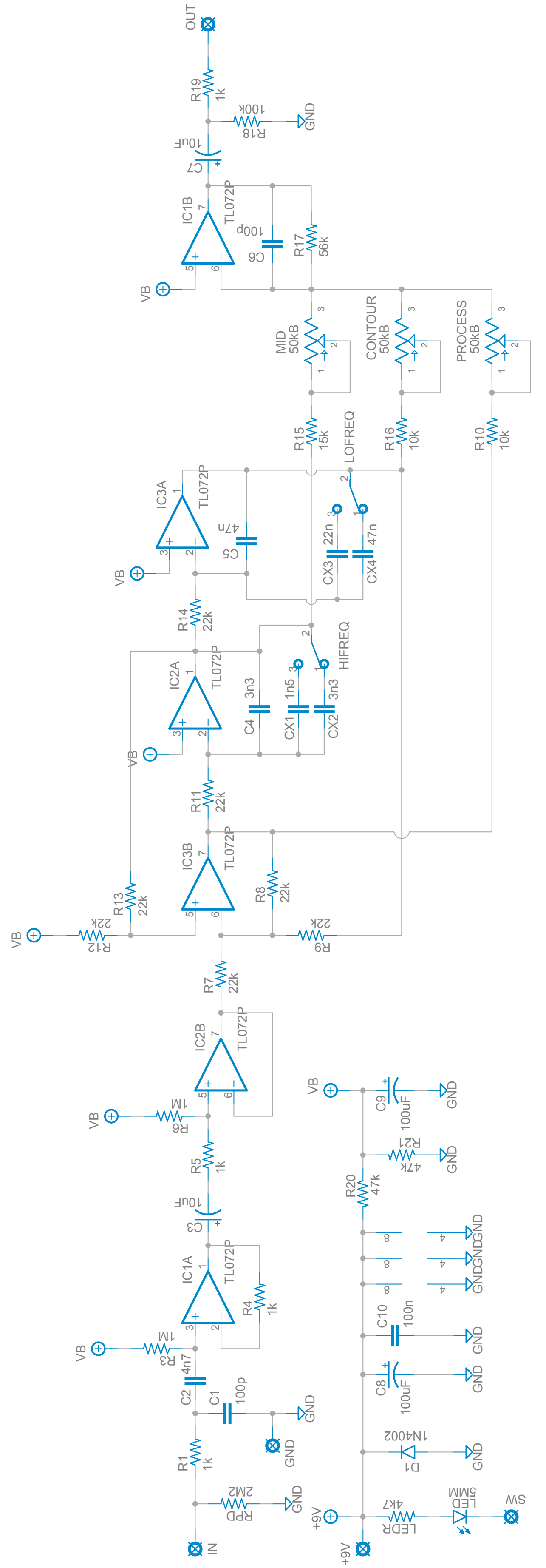
[3] **Low frequency switch:** These parts form the frequency selector for the Contour (low) knob. I recommend socketing **C5**, **CX3** and **CX4** and experimenting to find the frequencies that sound best to you. See page 1 for ideas. To omit this control, use **47n** for **C5** and omit CX3, CX4 and the Low Freq switch.

[4] **C10** was added in the 2015 revision (orders shipped after 12/2/2015). It provides additional power filtering.

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF. Many online suppliers do not use nanofarads, so you'll often have to look for 0.047uF instead of 47n, 0.0056uF instead of 5n6, etc.
- The PCB layout assumes the use of film capacitors with 5mm lead spacing for all values 1nF through 470nF. I prefer EPCOS box film or Panasonic ECQ-B/V-series.
- Potentiometers are Alpha 16mm right-angle PCB mount.
- Switches are Taiway (Small Bear) or Mountain Switch (Mouser) brand with solder lugs. I prefer the short-toggle variety, but that's just a matter of aesthetics.
- I recommend using these dust covers / insulators from Small Bear to insulate the back of the pots from the board and prevent shorts. If you don't use these, use some electrical tape or cardboard to act as insulation. The right-angle pots will make direct contact with the solder pads otherwise.

Schematic



OUT
R19
1k
R18
100K
GND
C7
10uF
IC1B
TL072P
C6
100p
R17
56k
VB
MID
50kB
CONTOUR
50kB
PROCESS
50kB
R15
15k
R16
10k
R10
10k
LOFREQ
IC3A
TL072P
C5
47n
CX3
22n
CX4
47n
R14
22k
HIFREQ
IC2A
TL072P
C4
3n3
CX1
1n5
CX2
3n3
R11
22k
IC3B
TL072P
R13
22k
R12
22K
R8
22k
R9
22K
R7
22k
IC2B
TL072P
R6
1M
R5
1k
C3
10uF
IC1A
TL072P
R4
1k
R3
1M
C2
4n7
C1
100p
R1
1k
RPD
2M2
IN
C9
100uF
R21
47K
R20
47k
C10
100n
C8
100uF
D1
1N4002
+9V
LEDR
4K7
LED
5MM
SW

General Build Instructions

These are general guidelines and explanations for all Aion Electronics DIY projects, so be aware that not everything described below may apply to this particular project.

Build Order

When putting together the PCB, it's recommended that you do not yet solder any of the enclosure-mounted control components (pots and switches) to the board. Instead, follow this build order:

1. Attach the **audio jacks**, **DC jack** and **footswitch** to the enclosure.
2. Firmly attach the **pots** and **switches** to the enclosure, taking care that they are aligned and straight.
3. Push the **LED**[1] into the hole in the enclosure with the leads sticking straight up, ensuring that the flat side is oriented according to the silkscreen on the PCB.
4. Fit the **PCB** onto all the control components, including the leads of the LED. If it doesn't fit, or if you need to bend things more than you think you should, double-check the alignment of the pots and switches.
5. Once you feel good about everything, **solder them from the top**[2] as the last step before wiring. This way there is no stress on the solder joints from slight misalignments that do not fit the drilled holes. You can still take it out easily if the build needs to be debugged, but now the PCB is "custom-fit" to that particular enclosure.
6. Wire everything according to the wiring diagram on the last page.

[1] **For the LED**: You can use a bezel if you'd like, but generally it's easier just to drill the proper size of hole and push the LED through so it fits snugly. If you solder it directly to the PCB, it'll stay put even if the hole is slightly too big. Make absolutely sure the LED is oriented correctly (the flat side matches the silk screen) before soldering, as it'll be a pain to fix later! After it's soldered, clip off the excess length of the leads.

[2] **Note on soldering the toggle switch(es):** It will require a good amount of solder to fill the pads. Try to be as quick as possible to avoid melting the lugs, and be prepared to feed a lot of solder as soon as the solder starts to melt. I recommend waiting 20-30 seconds between soldering each lug to give it time to cool down.

"RPD" and "LEDR" resistors

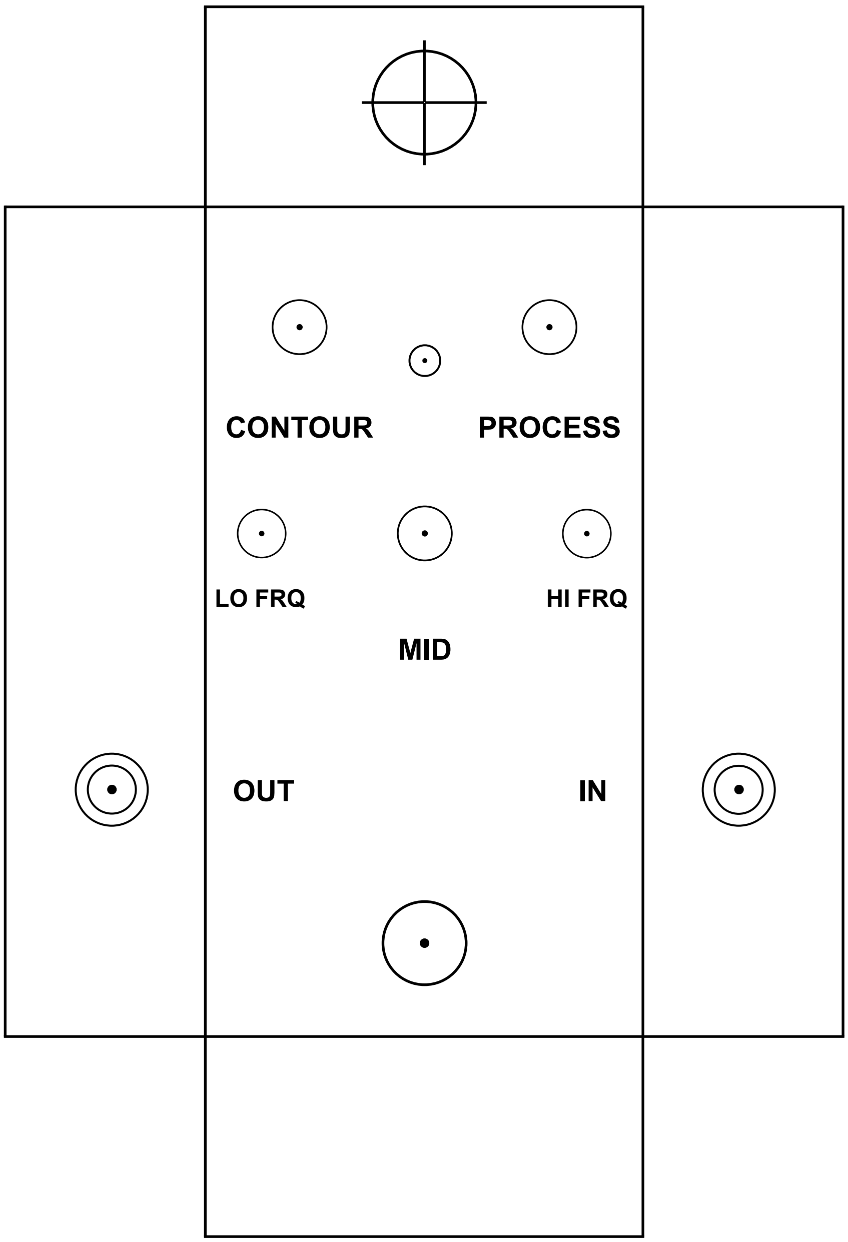
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

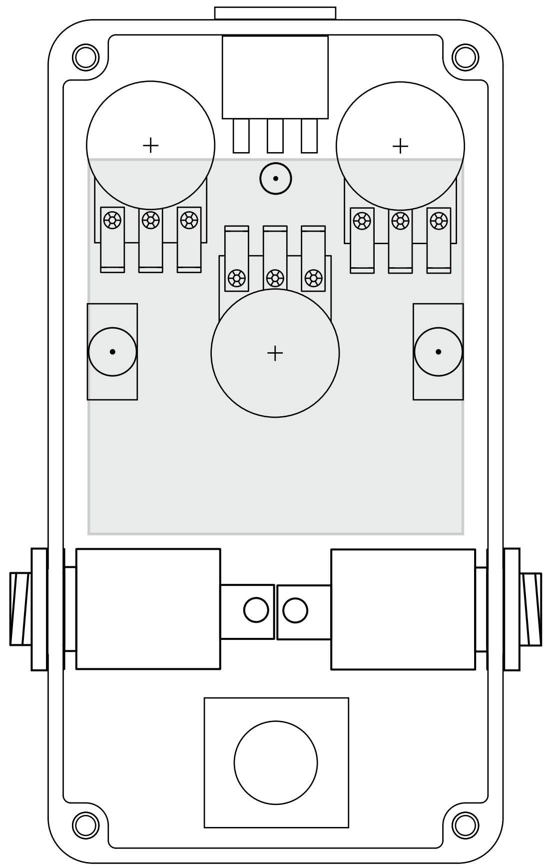
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and have an adult cut out the drilling template below for you. Tape it to the enclosure to secure it while drilling. Note that the holes are shown slightly smaller than they need to be, so drill out the holes as shown and then step up until they are the correct size for the components.



Hammond 1590B
(bottom/inside view)



Parts Used

- Switchcraft 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

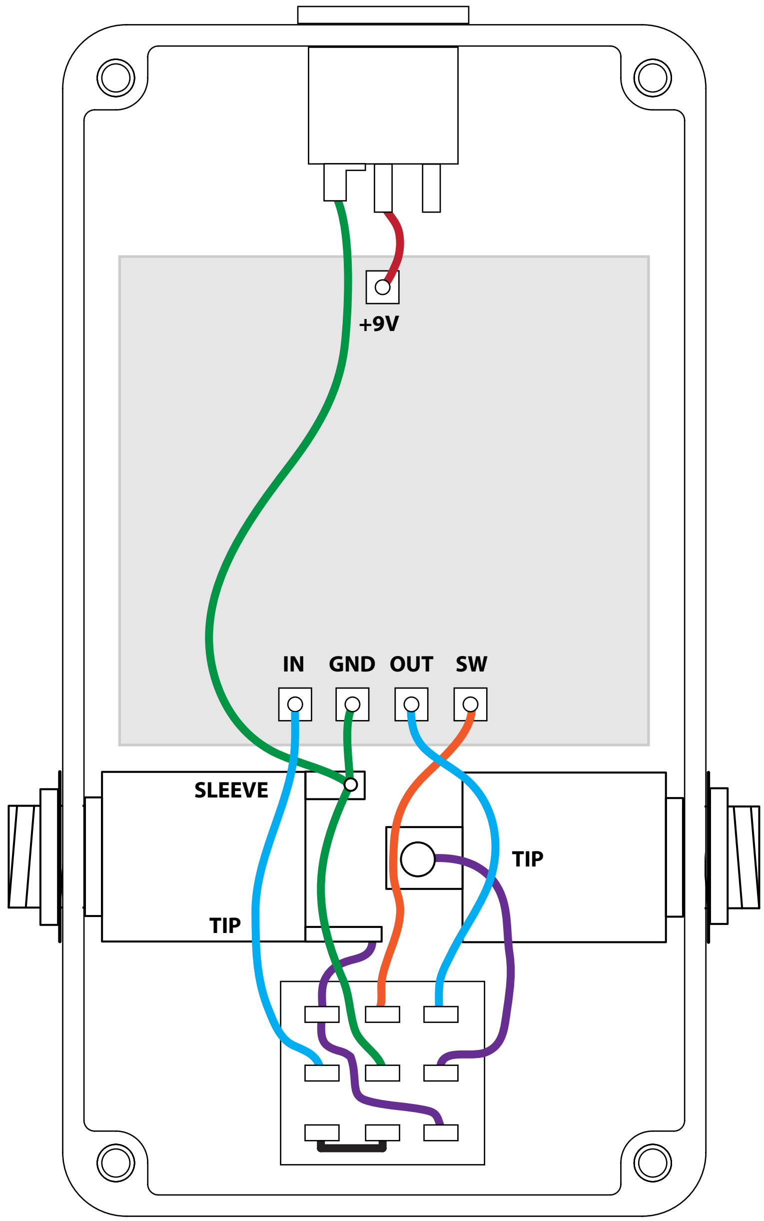
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

The **SW** pad is the cathode connection for the LED. This will connect to ground to turn it on when the switch is on. Usage of the on-board LED connection is not required if you have specific placement needs for your enclosure, but's incredibly convenient.

The wiring diagram also makes use of **star grounding** principles where all of the grounds connect to a single ground point (in this case the sleeve of the input jack). This is best practice to avoid added noise caused by improper grounding. The sleeve of the output jack is unconnected.

If using a painted or powdercoated enclosure, **make sure both jacks have solid contact with bare aluminum** for grounding purposes. You may need to sand off some of the paint or powdercoat on the inside in order to make this happen.

Make sure to double-check the markings of the pads on the PCB for your particular project – they are not always in the order shown here!



License / Usage

No direct support is offered for these PCBs beyond the provided documentation. It is assumed that you have at least some experience building pedals before starting one of these. Replacements and refunds will not be offered unless it can be shown that the circuit or documentation are in error. I have in good faith tested all of these circuits. However, I have not necessarily tested every listed modification or variation. These are offered only as suggestions based on the experience and opinions of others.

Projects may be used for commercial endeavors in any quantity unless specifically noted. No bulk pricing or discounting is offered. No attribution is necessary, though a link back is always greatly appreciated. The only usage restrictions are that **(1) you cannot resell the PCB as part of a kit**, and **(2) you cannot "goop" the circuit, scratch off the screenprint, or otherwise obfuscate the circuit to disguise its source**. (In other words: you don't have to go out of your way to advertise the fact that you use these PCBs, but please don't go out of your way to hide it. The guitar effects pedal industry needs more transparency, not less!)