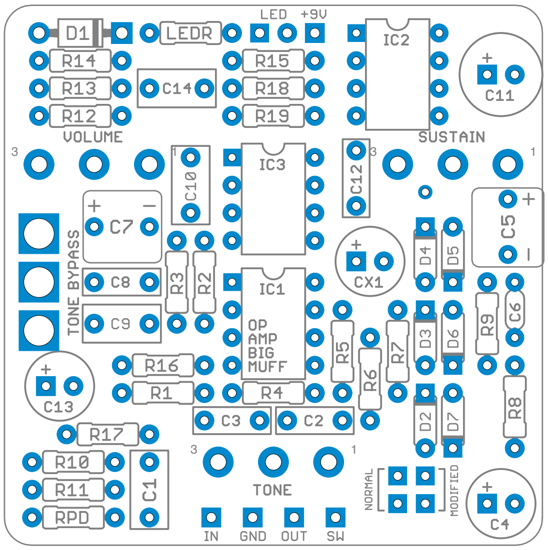
Corvus

EHX Big Muff Pi (IC/Op-Amp Version)

Overview

Corvus Project Link



The Corvus Fuzz is a clone of the 1978 Big Muff Pi, a major redesign of the classic Big Muff Pi circuit. It was designed by Howard Davis and Michael Abrams and incorporated op-amps instead of transistors for the amplification stages.

The op-amp Big Muff is perhaps best known as the fuzz pedal used on the Smashing Pumpkins' *Siamese Dream* album. The tone is much more hard-edged and the EQ is a little flatter than the classic Muff.

Two versions of this Big Muff were produced. The "Version 4" did not have a tone bypass switch and the "Version 5" did, but they were otherwise identical and were sold at the same time.

The op-amp Big Muff Pi also comprised half of the Deluxe Big Muff from the same year, which also included a Soul Preacher compressor and a knob to blend between the two.

Controls & Usage

- **Sustain** controls the amount of gain from the op amp that is fed through the clipping stage. This affects the overall sustain of the circuit as well.
- **Volume** is the output level of the effect.
- **Tone** allows you to pan between a bass boost and a treble boost, with the 12:00 center position being neutral. This stage is nearly identical to the classic Big Muff.
- **Tone Bypass** is a switch that allows you to completely bypass the tone section of the circuit, replacing it with a fixed R-C filter. When active, it causes the tone knob to have no effect. It also increases the volume of the circuit by 6-7 dB.

Modifications

One major modification reconfigures the tone knob for better control over the highs and lows. You can choose either "Normal" or "Modified" for the tone control by setting a jumper in one position or another. If you want to hear the difference, just put sockets on all four of these pads and use a U-shaped jumper wire to bridge either pair of pads. (Thanks to Analogguru for coming up with this modification.)

A second modification allows you to use a dual op-amp for the third stage. The original unit uses a dual op-amp (4558) for the first two stages and a single op-amp (741) for the third stage. Two different socket positions have been provided for the second op-amp, allowing you to use either a single (IC2) or a dual (IC3). This increases the flexibility for experimentation if you want to try out different options. When using the dual op-amp option, the second half of the op-amp is disabled.

Parts

Resistors

R1	56k
R2	330k
R3	10k
R4	47k
R5	560k
R6	62k
R7	47R
R8	8k2
R9	470k
R10	5k6
R11	1k2
R12	47k
R13	47k
R14	100k
R15	47R
R16	220k
R17	220k
R18	820k
R19	1M
RPD	2M2
LEDR	4k7

Capacitors

C1	150n
C2	10n
C3	4n7
C4	10uF electro
C5	1uF film
C6	330pF MLCC
C7	1uF
C8	100n
C9	120n
C10	150n
C11	100uF electro
C12	100n
C13	10uF electro
C14	220n
CX1	10uF electro [1]

Semiconductors

IC1	RC4558P
IC2	LM741 [2]
IC3	(omit) [2]
D1	1N4002
D2–D7	1N914
LED	5mm LED

Potentiometers

Sustain	10kB
Volume	100kA
Tone	10kB

Switches

TONE BYP	SPDT toggle

Build Notes

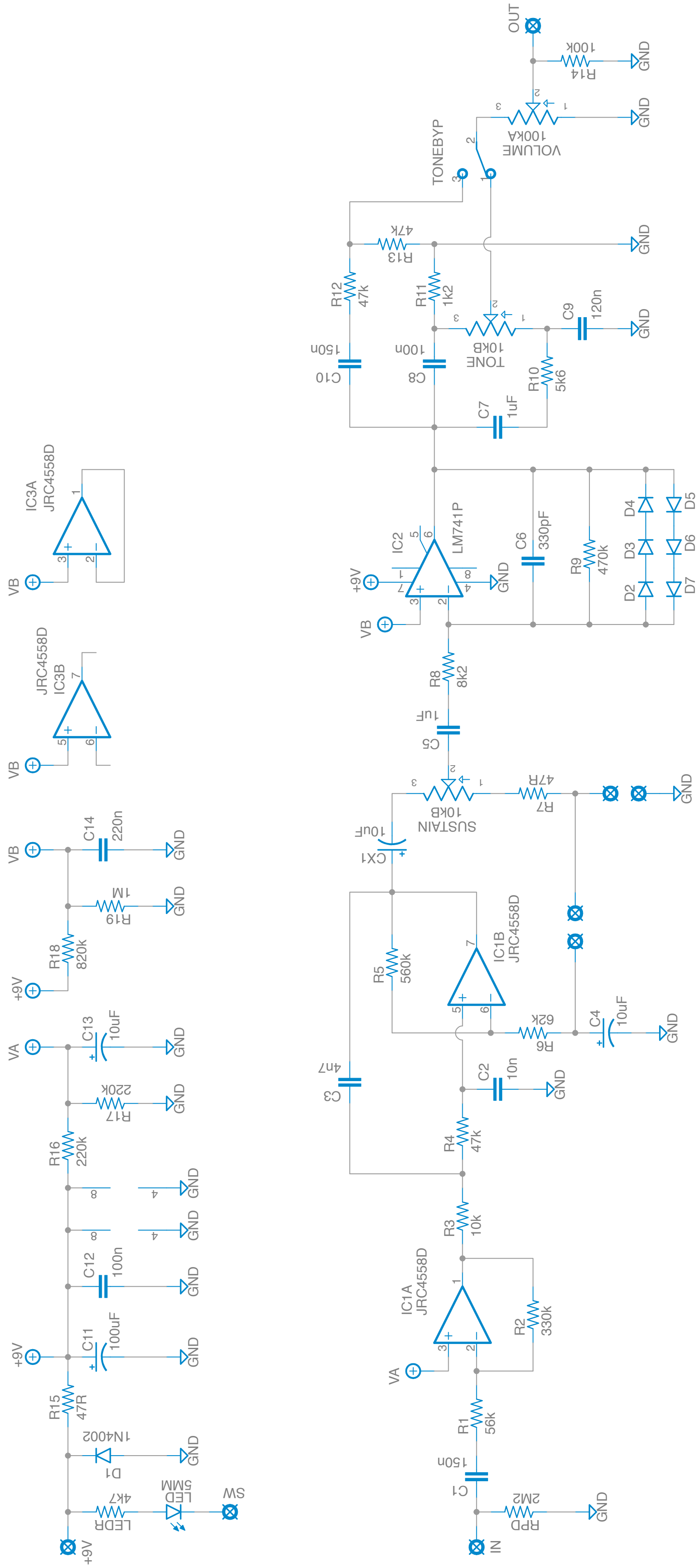
[1] **Optional:** CX1 is only needed if using the modified tonestack. It can be omitted if you are building a pure vintage unit.

[2] **One or the other:** IC2 is a single op-amp and IC3 is a dual. The two positions allow you to experiment with either type (e.g. 741 or OPA604 for single or TL072, 4558, or NE5532 for dual). Do not use both at once.

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.
- 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
R14
100K
GND
VOLUME
100KA
TONEBYP
R13
47K
R12
47k
R11
1k2
TONE
10KB
C9
120n
C10
150n
C8
100n
R10
5k6
C7
1uF
IC2
LM741P
+9V
VB
C6
330pF
R9
470k
D2
D3
D4
D5
D6
D7
R8
8k2
C5
1uF
SUSTAIN
10KB
R7
47R
CX1
10uF
R5
560k
IC1B
JRC4558D
R6
62k
C4
10uF
C3
4n7
C2
10n
R4
47k
R3
10k
IC1A
JRC4558D
VA
R2
330k
R1
56k
C1
150n
RPD
2M2
IN
IC3A
JRC4558D
IC3B
JRC4558D
C14
220n
R19
1M
R18
820k
C13
10uF
R17
220K
R16
220k
C12
100n
C11
100uF
R15
47R
D1
1N4002
LEDR
4K7
LED
5MM
SW
+9V

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

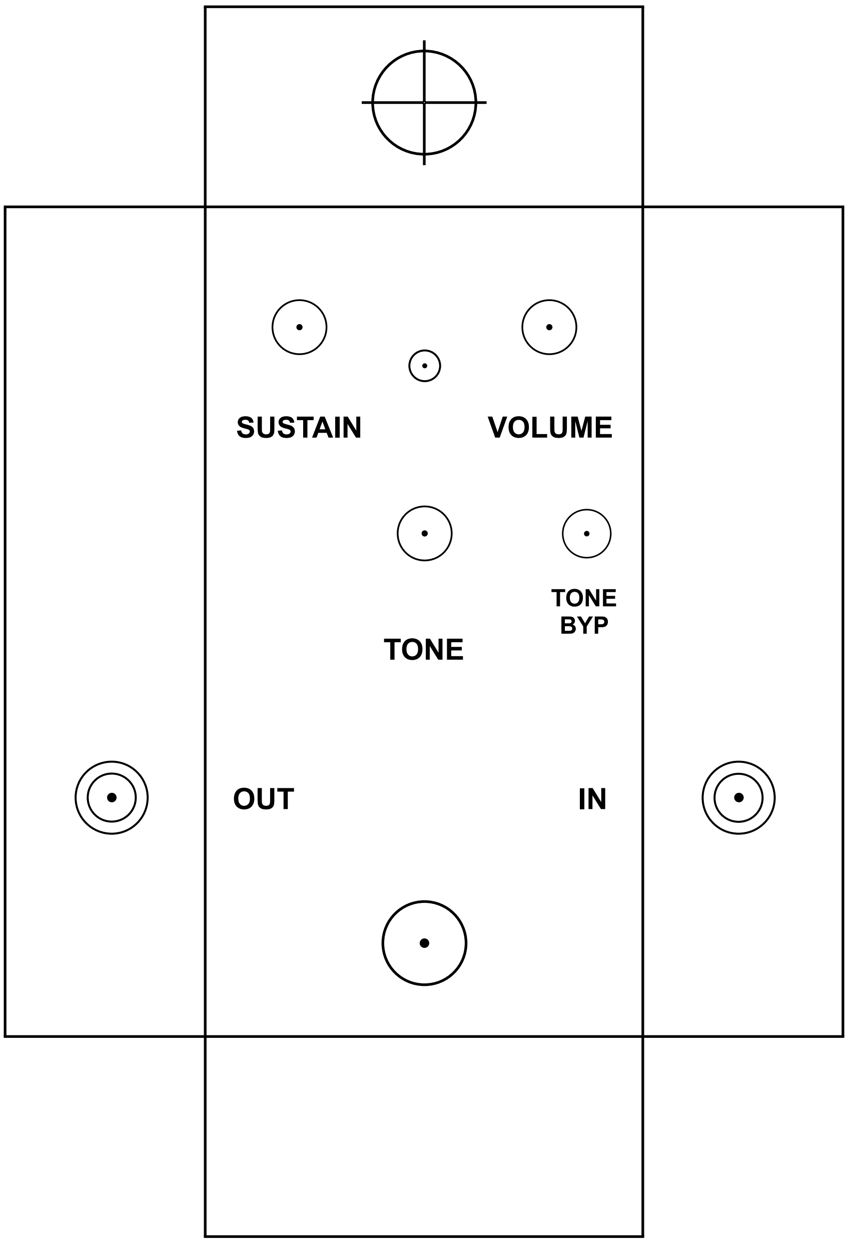
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

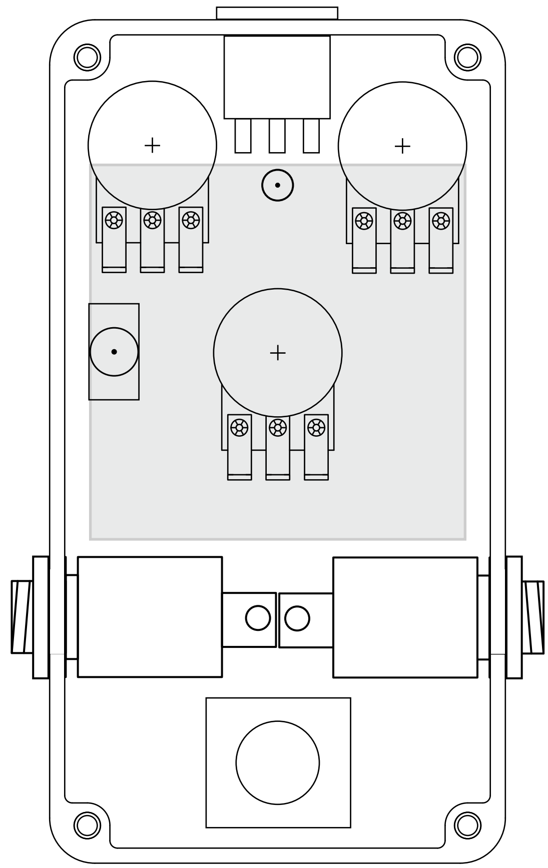
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and cut out the drilling template below. 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

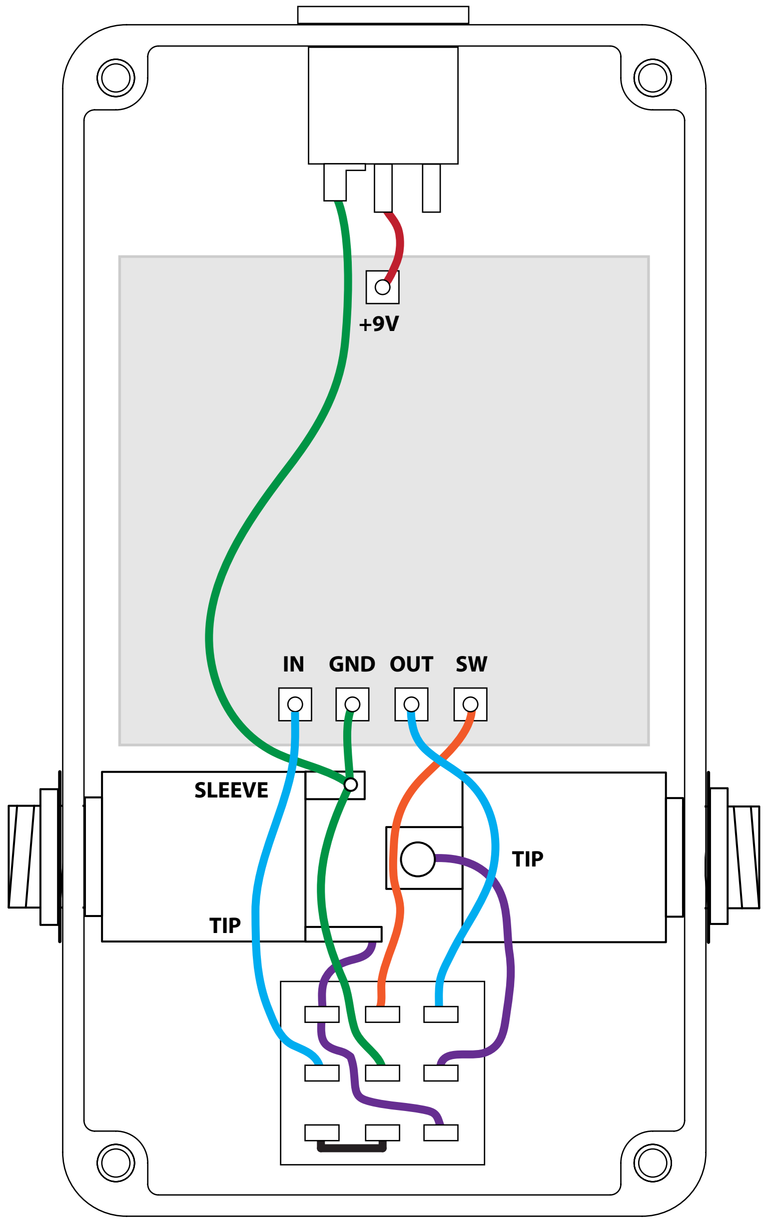
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)