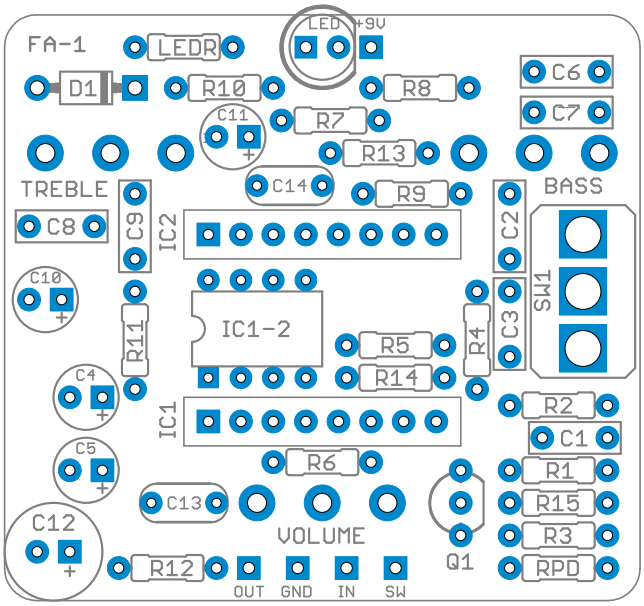
Prism Preamp

BOSS® FA-1 FET Amplifier

Overview

Prism Project Link



The Prism Preamp is a pedal conversion of the obscure but highly sought-after BOSS® FA-1 FET Amplifier. The original, released in the early 80's and discontinued within a few years, was an always-on effect that clipped to your belt and ran on a 9-volt battery.

It's a fantastic-sounding circuit, but the nonstandard implementation prevented it from getting much attention. This project uses the exact same circuit from the original unit, but the layout has been developed so that it can be built as a standard floor unit with a footswitch, LED and 9v adapter jack.

Controls & Usage

The FA-1 is essentially a booster with a ton of gain on tap, as well as tone-shaping controls. It doesn't produce much clipping on its own, but is wonderful when slamming the frontend of an amp or another overdrive pedal.

- **Treble** and **Bass** controls allow for EQ adjustments
- **Volume** controls the overall output
- **Low Cut** switch gives you the option of a flat response or a 6dB/octave low cut

Modifications & Experimentation

The original circuit uses the long-obsolete **HA1457W** SIL op amp. These are still readily available from sources such as Small Bear Electronics, but since BOSS most likely selected them for cost and convenience rather than sound, it's worthwhile to experiment with other ICs.

Additional pads and connections have been provided to allow you to try a dual op amp instead of the two single op amps. The pads for the dual op amp are the standard 4558 pinout, so you can also use a JRC4558, TL072, or any number of others. Try a **LM1458** (basically a dual 741) for a lower-fidelity type of op amp that's more similar in performance to the HA1457W. I've A/B'd the LM1458 with two HA1457s and could not hear the difference.

It's recommended that you socket all three of them to allow for experimentation—but make sure you only use one configuration or the other.

Parts

Resistors

R1	10k
R2	3M3
R3	10k
R4	470k
R5	15k
R6	470k
R7	8k2
R8	33k
R9	8k2
R10	10k
R11	470R
R12	100k
R13	22k
R14	6k8
R15	15k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	10n
C2	470n
C3	47n
C4	10uF [1]
C5	10uF [1]
C6	33n
C7	33n
C8	5n6
C9	5n6
C10	10uF [1]
C11	10uF [1]
C12	100uF
C13	100pF [2]
C14	100pF [2]

Semiconductors

Q1	2SK246
IC1	HA1457W [3]
IC2	HA1457W [3]
IC1-2	[3]
D1	1N4002
LED	5MM

Potentiometers

Treble	50kB
Bass	50kB
Volume	1MA

Other

SW1	SPDT or SPST

[1] **Can be electrolytic or tantalum.** C4, C5 and C10 are in the signal path while C11 is for filtering.

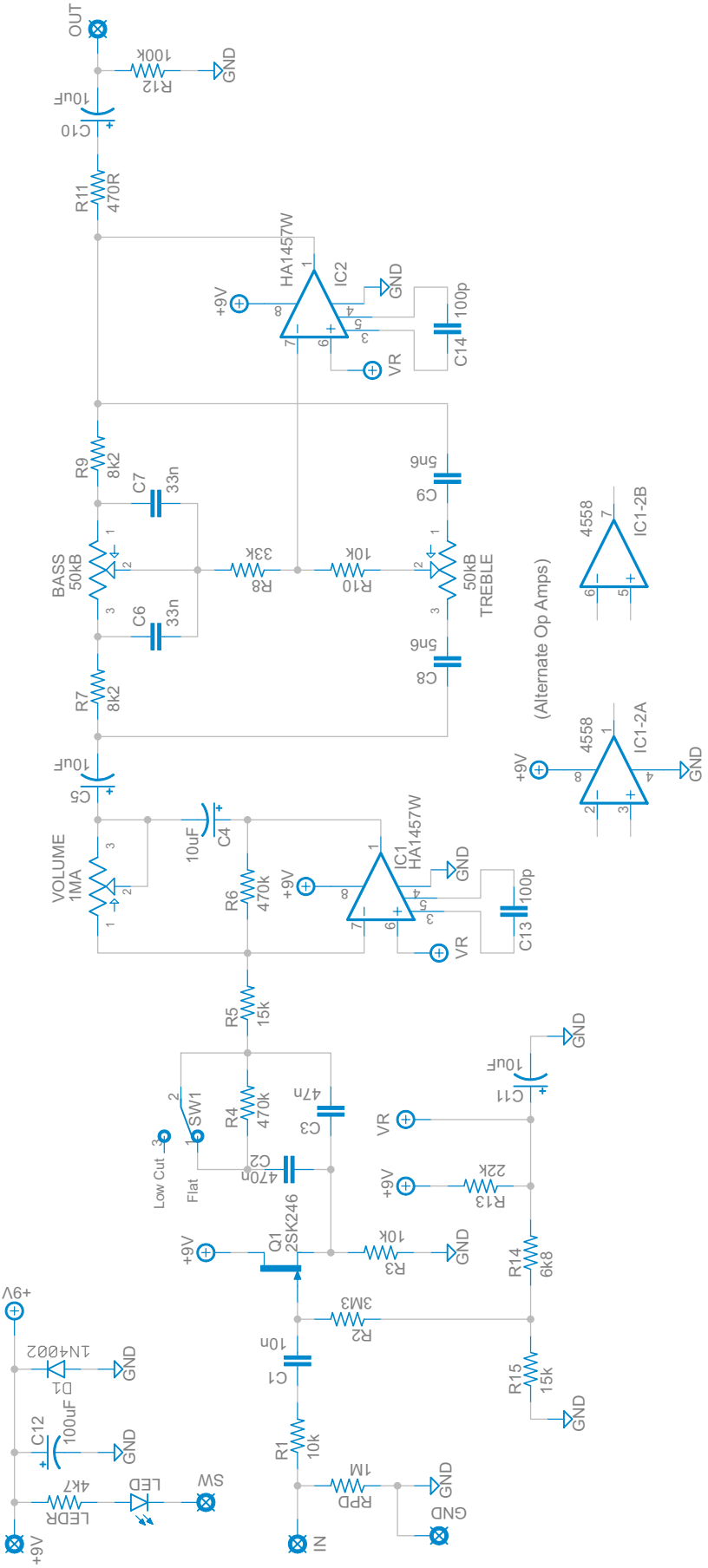
[2] **External compensation caps**; leave empty if not using HA1457W. I use multilayer ceramic or film for these but regular ceramic is fine as well.

[3] **Do not use all three of these.** Either use two HA1457Ws in IC1 and IC2 or use one 4558-style dual op amp in IC1-2. I recommend socketing all three so you can experiment.

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-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
100K
R12
GND
10uF
C10
R11
470R
HA1457W
IC2
+9V
GND
100p
C14
VR
R9
8k2
C7
33n
BASS
50kB
33k
R8
10k
R10
50kB
TREBLE
5n6
C9
C6
33n
5n6
C8
R7
8k2
10uF
C5
10uF
C4
VOLUME
1MA
R6
470k
IC1
HA1457W
C13
100p
R5
15k
SW1
R4
470k
47n
C3
Low Cut
Flat
470n
C2
VR
10uF
C11
GND
+9V
22k
R13
Q1
2SK246
10k
R3
R14
6k8
3M3
R2
10n
C1
R15
15k
R1
10k
1M
RPD
IN
1N4002
D1
C12
100uF
LEDR
4k7
LED
SW
+9V
(Alternate Op Amps)
4558
IC1-2B
4558
IC1-2A

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

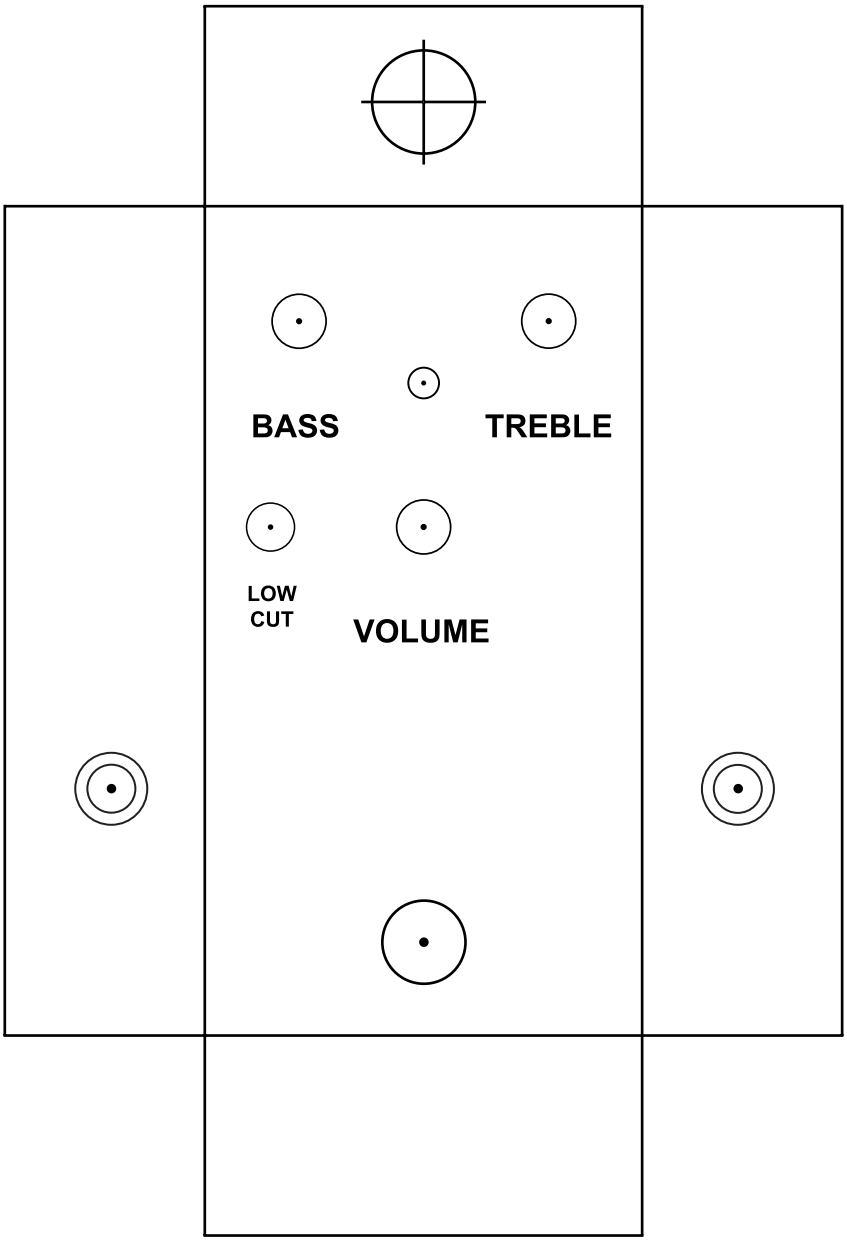
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

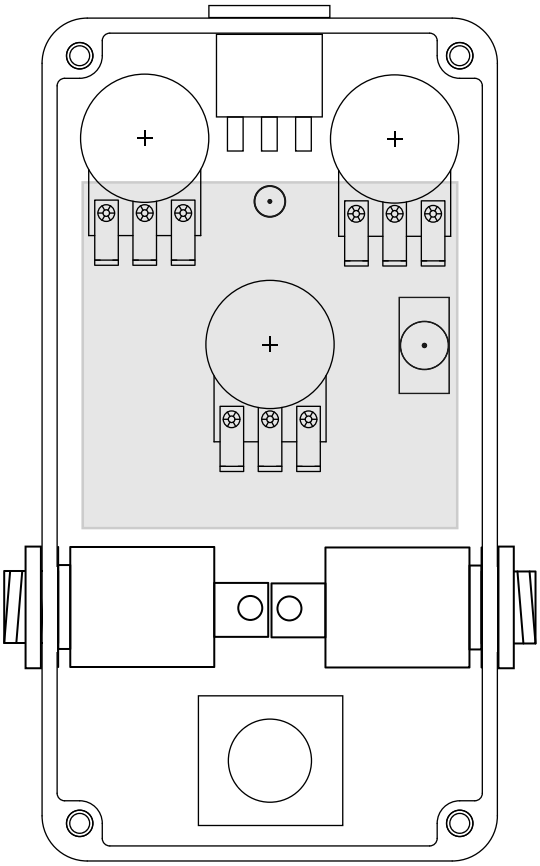
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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 #111A enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

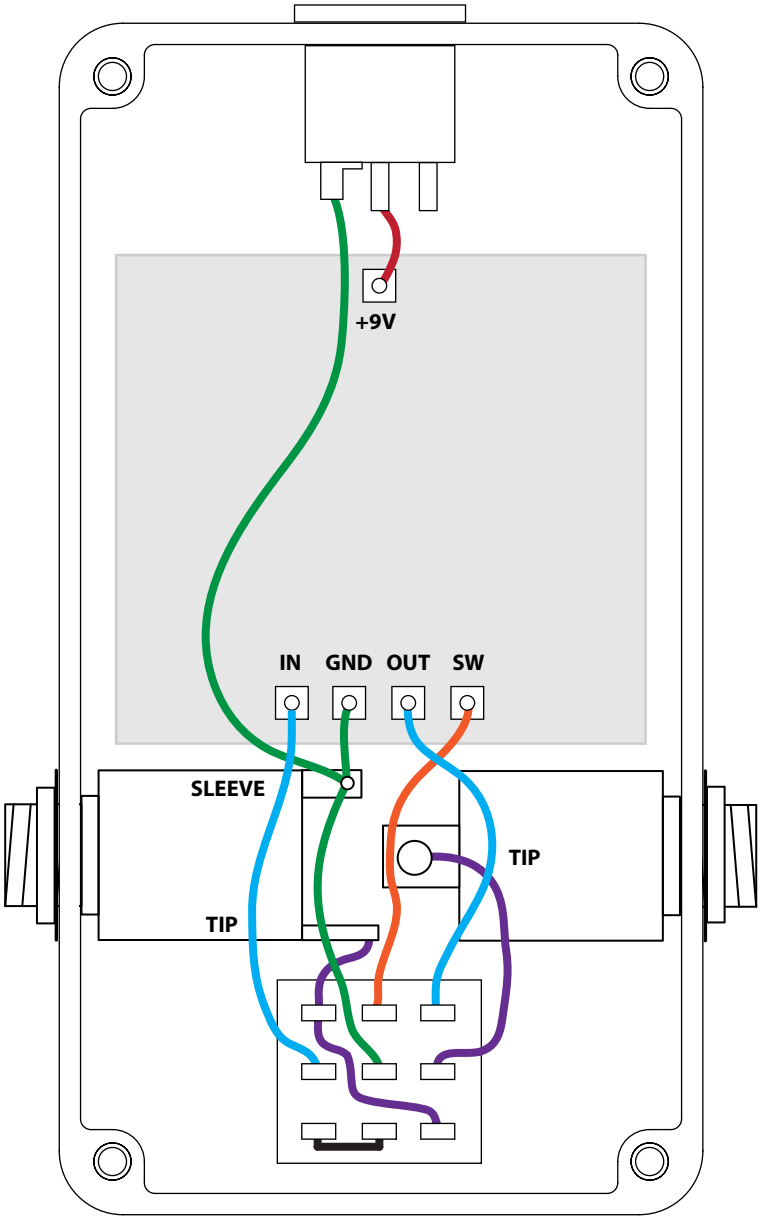
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)