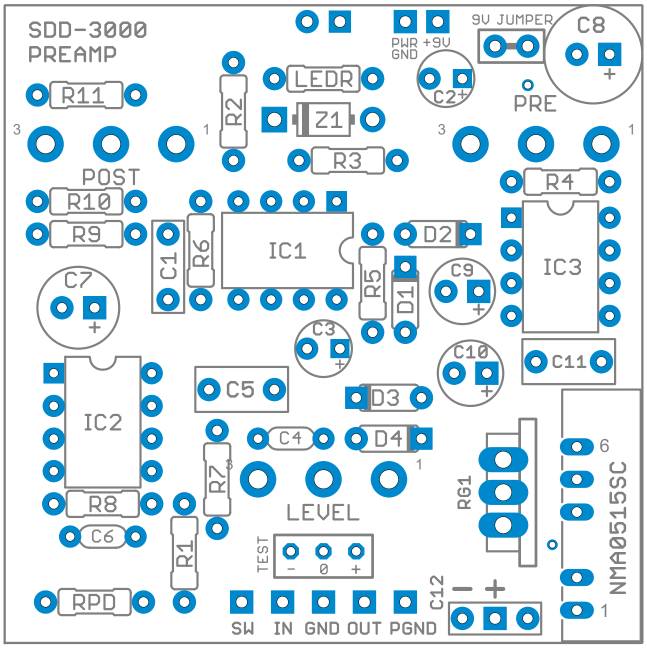
Eclipse Preamp

Korg SDD-3000 Delay (Preamp)

Overview



The Eclipse Preamp project is a pedal conversion of the preamp section of the Korg SDD-3000 digital delay rack unit. The Edge from U2 uses these rack units in bypass mode at the end of his signal chain to drive a Vox AC30, and many people (including Daniel Lanois, U2's producer) continue to swear by them.

Since the original was a studio rack unit, it incorporated a +/-15V power supply. This gives an enormous amount of headroom and—in theory—it is entirely unnecessary for guitar-level signals. However, some claim that the higher-voltage bipolar supply is crucial to the tone, and I've never played an original unit so I can't say they're wrong! As a result, this project has options for either +/-9V or +/-15V power.

Controls & Usage

The Eclipse Preamp is an op-amp booster with a great deal of gain and headroom due to its bipolar power supply. The controls are pretty basic:

- **Pre** and **Post** controls allow for attenuation of the signal before and after the op-amp gain stage. These were switches in the original, but pots allow for a finer level of control.
- **Level** controls the overall output. It gets *very loud*, so start on zero and turn it up slowly!

Power Supply

The default (or cheapest, or easiest) power supply configuration for this project is to use a TC1044S charge pump to generate +/-9V. **If you use this mode, you will need to jumper the two pads marked "9V JUMPER" on the board.**

Alternately, this project allows for the usage of a +/-15V DC converter (Murata NKA0515SC), which is an expensive component, but it allows you to use the exact voltage of the original unit while still being powered by a standard 9VDC supply. You will **not** be able to switch between the two power supplies easily, so choose ahead of time which one you want to use.

Right below the **Level** control on the PCB is a group of pads marked **TEST**. This is for verifying the bipolar supply voltage. When you first power up the unit, if you have a multimeter, set it to DC Voltage mode and touch the common (black) probe to the pad marked **0**. Touch the red probe to the **+** and make sure it's either 9V or 15V depending on which supply you used. Then, move the red probe to the **-** and make sure it's -9V or -15V. (Allow for some slight variance, within a volt or so.)

Note the revised wiring diagram on the last page - this one is slightly different than other projects.

Parts

Resistors

R1	5k6
R2	560k
R3	20k (22k) [1]
R4	10k
R5	3k3
R6	3k3
R7	110k (100k) [1]
R8	360k (330k) [1]
R9	22k
R10	220R
R11	470R
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	100n
C2	4u7
C3	4u7
C4	10pF
C5	470n
C6	4.7pF
C7	47uF
C8	100uF

Semiconductors

IC1	TL072
IC2	JRC4558D
Z1	1N4739
D1-D4	1N4148
LED	5MM

Potentiometers

Pre	250kA
Post	10kA
Level	100kA

+/-9V Charge Pump [2]

IC3	TC1044SCPA [3]
C9	10uF
C10	10uF
9V JUMPER	Jumper

+/-15V DC Converter [2]

DC1	NKA0515SC
RG1	LM7805
C11	330n
C12	100n, 100uF [4]

[1] **The original circuit uses a few uncommon resistor values.** The values in parenthesis are the nearest common resistor value. You can safely use these common values instead if you are unable to find the original values in a convenient way. With their particular locations in the circuit, they don't affect anything.

[2] **Do not use both of these.** Choose either +/-9V or +/-15V and use only the parts in that section, ignoring the parts in the other one.

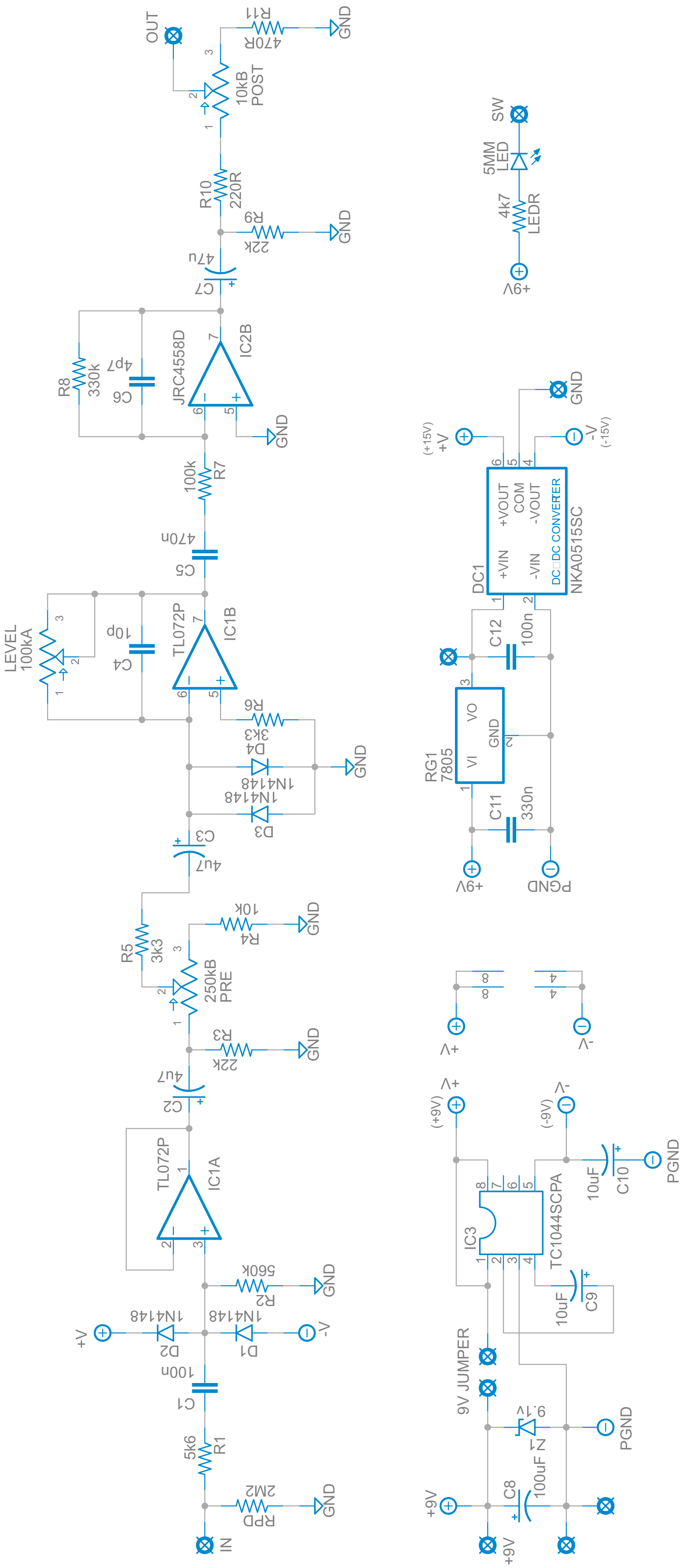
[3] **Can also use a MAX1044 here.** If using another equivalent charge pump, look at the datasheet and ensure that pin 1 is frequency boost, or you may get a high-pitched whine from the effect.

[4] **C12** is marked as a 100n capacitor in the schematic and build document, but I've seen other implementations of the DC-DC converter that call for a higher value electrolytic capacitor here (47uF or 100uF). Because of this I have included alternate pads for an electrolytic capacitor. Since it's filtering a regulated 5 volts, you can safely use a capacitor rated for 9V or 12V. The polarization is marked on the silkscreen.

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


IN
OUT
RPD
2M2
R1
5k6
C1
100n
D1
D2
1N4148
R2
560K
IC1A
TL072P
C2
4u7
R3
22K
250kB
PRE
R4
10K
R5
3k3
C3
D3
D4
R6
3K3
IC1B
C4
10p
LEVEL
100kA
C5
470n
R7
100k
IC2B
JRC4558D
C6
4p7
R8
330k
C7
47u
R9
22K
R10
220R
10kB
POST
R11
470R
GND
+V
-V
+9V
PGND
9V JUMPER
C8
100uF
Z1
9.1V
IC3
TC1044SCPA
C9
10uF
C10
(+9V)
(-9V)
C11
330n
RG1
7805
VI
GND
VO
C12
100n
DC1
+VIN
-VIN
+VOUT
COM
-VOUT
DC/DC CONVERTER
NKA0515SC
(+15V)
(-15V)
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

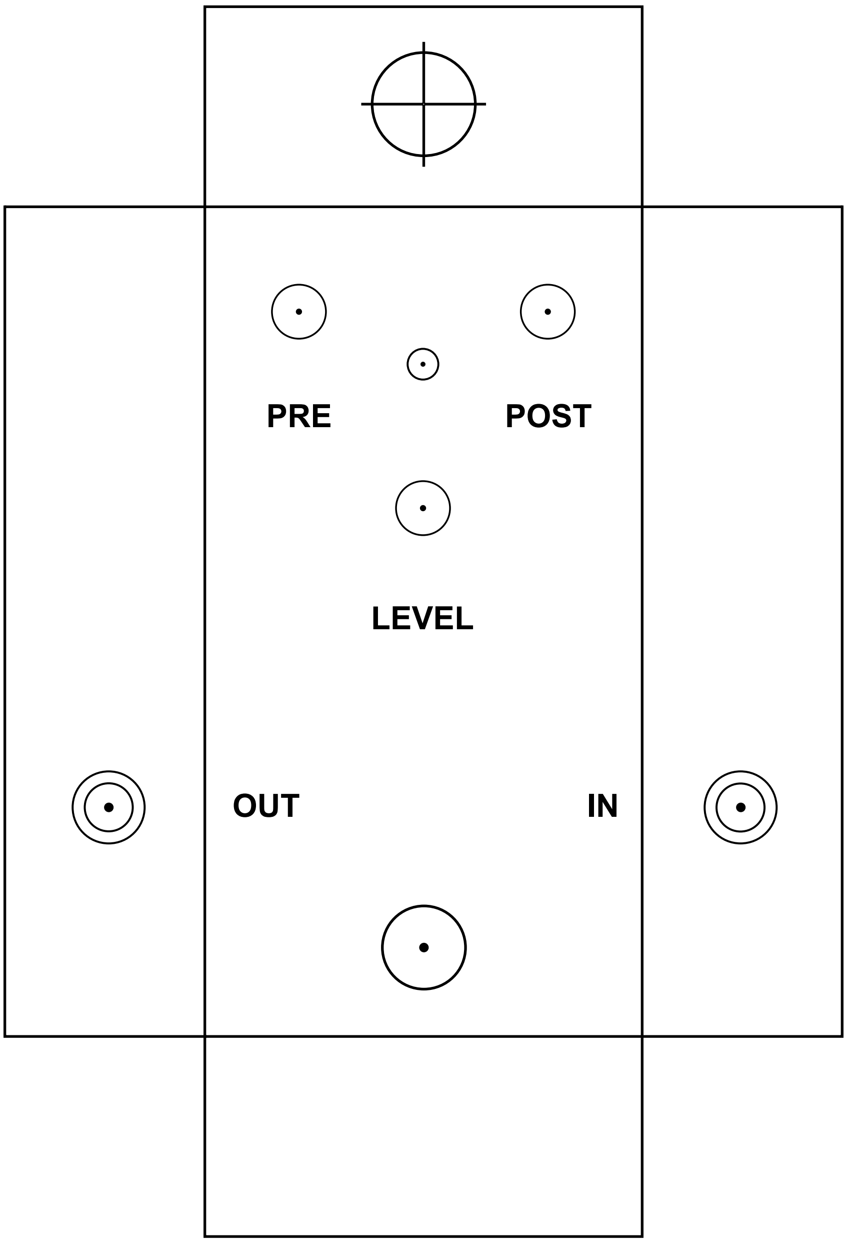
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

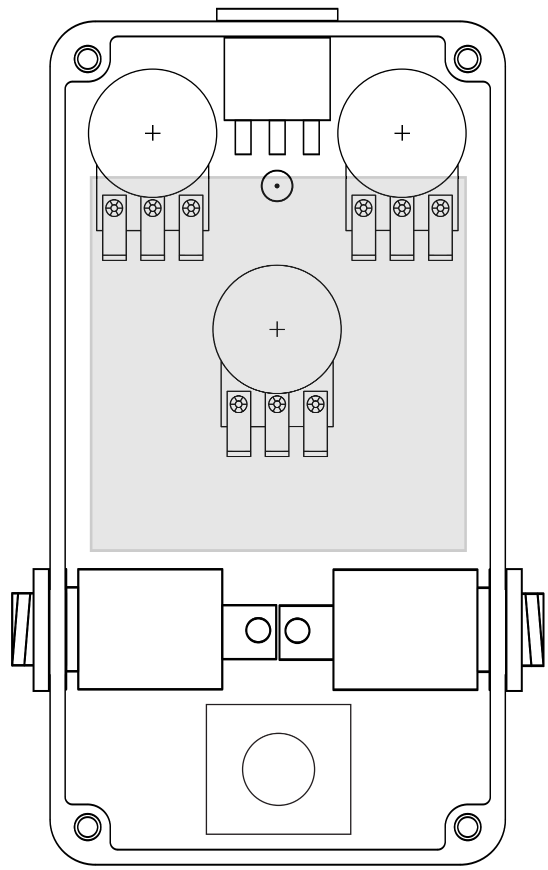
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

Standard Wiring Diagram

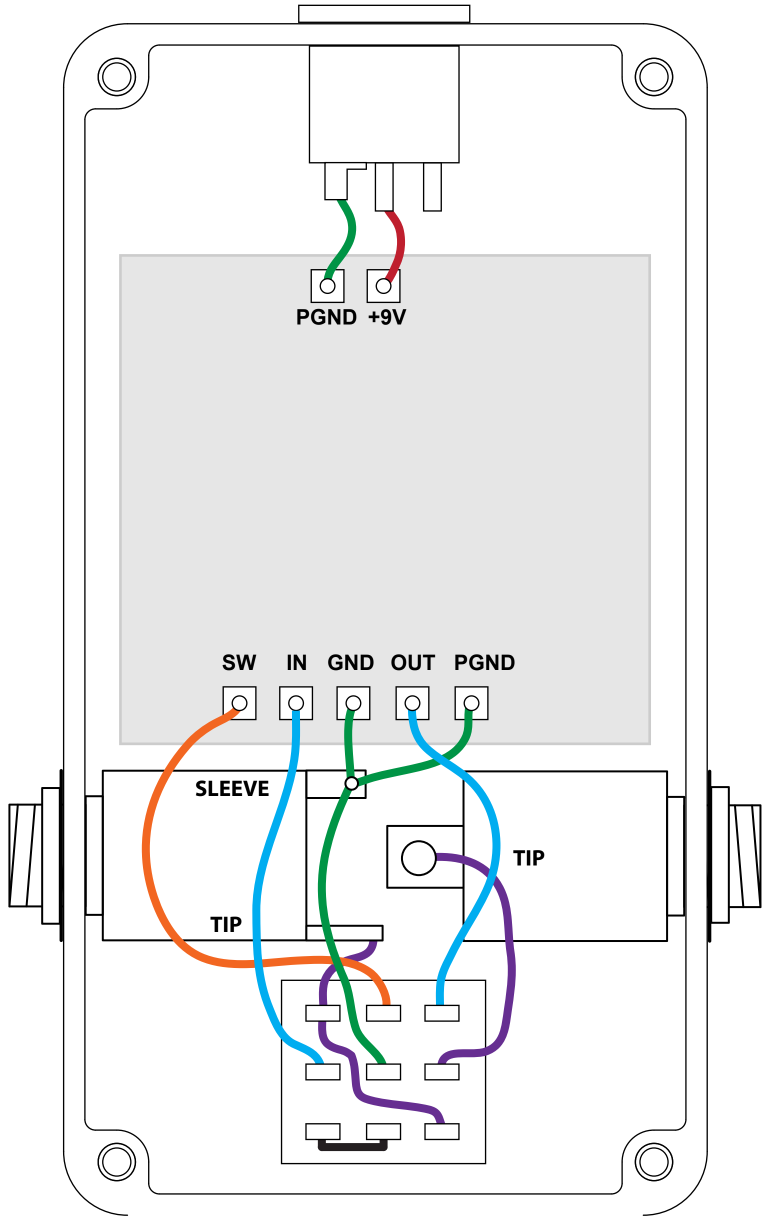
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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.

Note for SDD Preamp: The wiring diagram is slightly different due to the power supply. The power ground plane (marked **PGND**) is kept separate from the signal ground and joined together at the input jack.



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!)