PROJECT NAME

AEROLITH

aion DIY GUITAR EFFECTS

BASED ON
BOSS® BF-2 Flanger

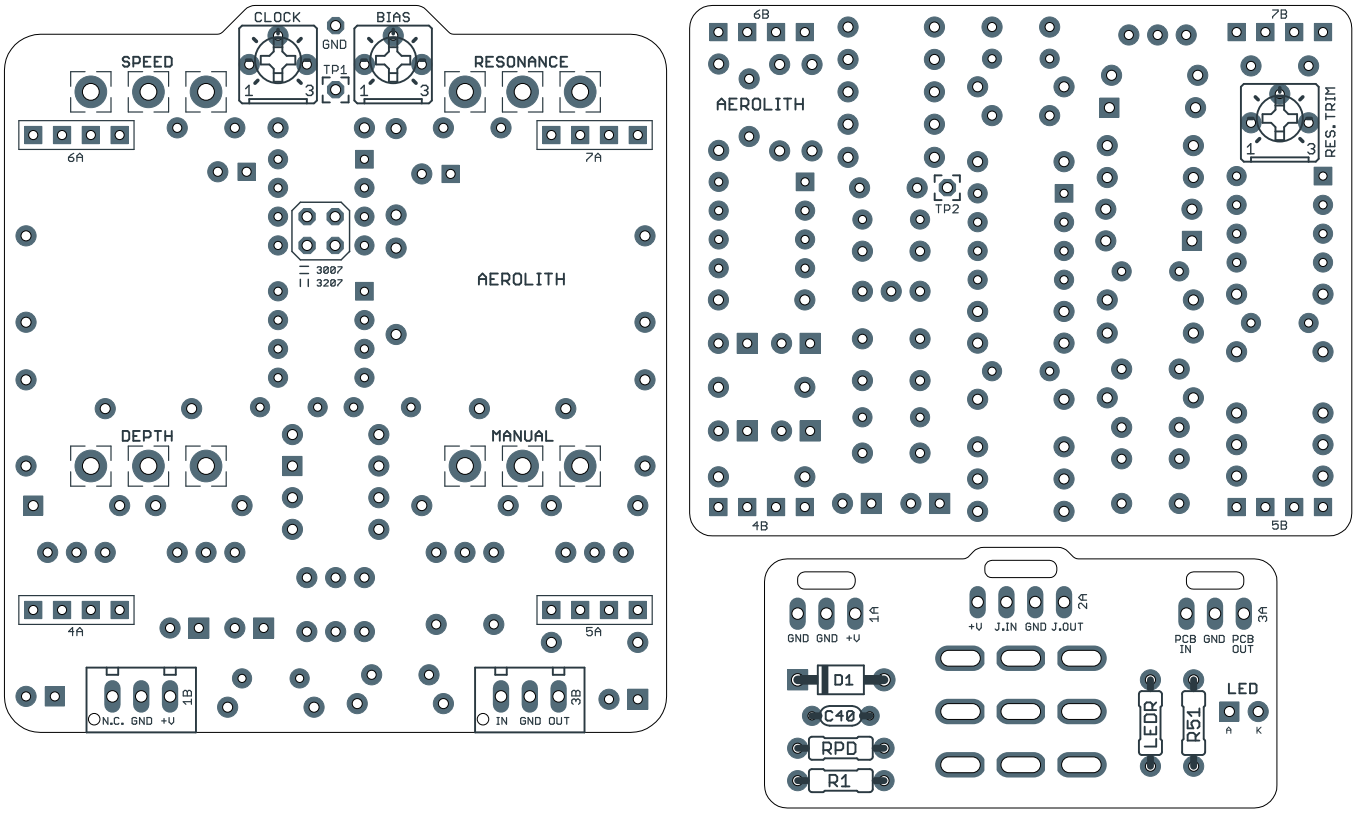
BUILD DIFFICULTY
Advanced

EFFECT TYPE
Flanger

DOCUMENT VERSION
1.0.0 (2026-07-03)

PROJECT SUMMARY

An early analog flanger that helped set the benchmark for 1980s modulation, from warm chorus tones to metallic resonance.



Actual size is 2.3" x 2.53" (bottom board), 2.3" x 1.85" (top board), and 1.78" x 0.87" (bypass board).

Since most of the components are mounted on the reverse side, a diagram of this side can be found on page 11.

IMPORTANT NOTE

This project has a specialized method of assembly that is different from most DIY builds, and because of this there are a lot of ways to make mistakes that are hard to fix. Please familiarize yourself with the assembly instructions on pages **7-8** before installing any of the components.

TABLE OF CONTENTS

INTRODUCTION

The Aerolith Analog Flanger is an adaptation of the BOSS BF-2 Flanger, one of the most popular flanger effects ever made. It has been used by a wide spectrum of different musicians, from Johnny Marr and Robert Smith all the way to Prince. It has also been used by Adam Jones and Justin Chancellor of TOOL and Stephen Carpenter of Deftones.

Originally released in 1980, the BF-2 was a compact upgrade to the BF-1 from 1977, which was one of the very first pedals that Roland produced under the BOSS brand name. Internally, the BF-1 used the SAD1024 with a CMOS clock, similar to the EHX Electric Mistress, while the BF-2 uses the MN3207/MN3102 pair. The circuits are otherwise very similar with the same four controls on the outside.

The BF-2 was in production from 1980 up until 2001 when it was replaced by the digital BF-3.

The Aerolith is a direct clone of the BF-2 circuit, converted to true bypass. Due to the size of the circuit, this project utilizes a stacked PCB layout in order to fit in a 125B enclosure. See pages 7-8 for instructions and diagrams for the PCB assembly process.

USAGE

The Aerolith has the following controls:

- **Speed** (Rate) controls the speed of the flange effect.
- **Manual** sets the base delay time, which is then modulated by the Speed control.
- **Depth** controls the width of the sweep, or in other words how far the delayed signal travels from the base delay time set by the Manual control.
- **Resonance** is a feedback loop that intensifies the effect as it's turned up. With no feedback, it's similar to a chorus effect.

PARTS LIST

This parts list is also available in a spreadsheet format which can be imported directly into Mouser for easy parts ordering. Mouser doesn't carry all the parts (most notably potentiometers) so the second tab lists all the non-Mouser parts as well as sources for each.

View parts list spreadsheet →

PART	VALUE	TYPE	NOTES
R1	1k	Metal film resistor, 1/4W	
R2	470k	Metal film resistor, 1/4W	
R3	10k	Metal film resistor, 1/4W	
R4	10k	Metal film resistor, 1/4W	
R5	47k	Metal film resistor, 1/4W	
R6	10k	Metal film resistor, 1/4W	
R7	47k	Metal film resistor, 1/4W	
R8	47k	Metal film resistor, 1/4W	
R9	10k	Metal film resistor, 1/4W	
R10	47k	Metal film resistor, 1/4W	
R11	470R	Metal film resistor, 1/4W	
R12	82k	Metal film resistor, 1/4W	
R13	220k	Metal film resistor, 1/4W	
R14	10k	Metal film resistor, 1/4W	
R15	47k	Metal film resistor, 1/4W	
R16	4k7	Metal film resistor, 1/4W	
R17	10k	Metal film resistor, 1/4W	
R18	10k	Metal film resistor, 1/4W	
R19	10k	Metal film resistor, 1/4W	
R20	100k	Metal film resistor, 1/4W	
R21	56k	Metal film resistor, 1/4W	
R22	330k	Metal film resistor, 1/4W	
R23	10k	Metal film resistor, 1/4W	
R24	10k	Metal film resistor, 1/4W	
R25	10k	Metal film resistor, 1/4W	
R26	10k	Metal film resistor, 1/4W	
R27	47k	Metal film resistor, 1/4W	
R28	22k	Metal film resistor, 1/4W	
R29	47k	Metal film resistor, 1/4W	
R30	27k	Metal film resistor, 1/4W	
R31	39k	Metal film resistor, 1/4W	
R32	33k	Metal film resistor, 1/4W	

PARTS LIST, CONT.

PART	VALUE	TYPE	NOTES
R33	33k	Metal film resistor, 1/4W	
R34	4k7	Metal film resistor, 1/4W	
R35	4k7	Metal film resistor, 1/4W	
R36	4k7	Metal film resistor, 1/4W	
R37	330k	Metal film resistor, 1/4W	
R38	220k	Metal film resistor, 1/4W	
R39	68k	Metal film resistor, 1/4W	
R40	150k	Metal film resistor, 1/4W	
R41	470k	Metal film resistor, 1/4W	
R42	4k7	Metal film resistor, 1/4W	
R43	10k	Metal film resistor, 1/4W	
R44	10k	Metal film resistor, 1/4W	
R45	100k	Metal film resistor, 1/4W	
R46	180k	Metal film resistor, 1/4W	
R47	1k5	Metal film resistor, 1/4W	
R48	220k	Metal film resistor, 1/4W	
R49	33k	Metal film resistor, 1/4W	
R50	33k	Metal film resistor, 1/4W	
R51	100k	Metal film resistor, 1/4W	
RPD	2M2	Metal film resistor, 1/4W	
LEDR	10k	Metal film resistor, 1/4W	
C1	47n	Film capacitor, 7.2 x 2.5mm	
C2	1uF	Film capacitor, 7.2 x 5mm	
C3	6n8	Film capacitor, 7.2 x 2.5mm	
C4	100pF	MLCC capacitor, NP0/C0G	
C5	6n8	Film capacitor, 7.2 x 2.5mm	
C6	100pF	MLCC capacitor, NP0/C0G	
C7	1uF	Film capacitor, 7.2 x 5mm	
C8	220pF	MLCC capacitor, NP0/C0G	
C9	47n	Film capacitor, 7.2 x 2.5mm	
C10	12n	Film capacitor, 7.2 x 2.5mm	
C11	150pF	MLCC capacitor, NP0/C0G	
C12	33n	Film capacitor, 7.2 x 2.5mm	
C13	1uF	Electrolytic capacitor, 5mm	
C14	1uF	Electrolytic capacitor, 5mm	
C15	33n	Film capacitor, 7.2 x 2.5mm	
C16	3n9	Film capacitor, 7.2 x 2.5mm	

PARTS LIST, CONT.

PART	VALUE	TYPE	NOTES
C17	8n2	Film capacitor, 7.2 x 2.5mm	
C18	330pF	MLCC capacitor, NP0/C0G	
C19	220n	Film capacitor, 7.2 x 3mm	
C20	1n	Film capacitor, 7.2 x 2.5mm	
C21	150pF	MLCC capacitor, NP0/C0G	
C22	220n	Film capacitor, 7.2 x 3mm	
C23	47n	Film capacitor, 7.2 x 2.5mm	
C24	47n	Film capacitor, 7.2 x 2.5mm	
C25	47pF	MLCC capacitor, NP0/C0G	
C26	5pF	MLCC capacitor, NP0/C0G	
C27	22uF	Tantalum capacitor, 044A	Can also be electrolytic. See build notes. Minimum 10V.
C28	47uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 6.3V.
C29	33uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 10V.
C30	33uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 10V.
C31	10n	Film capacitor, 7.2 x 2.5mm	
C32	470n	MLCC capacitor, X7R	
C33	47uF	Electrolytic capacitor, 5mm	
C34	22uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 16V.
C35	22uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 16V.
C36	22uF (7mm)	Electrolytic capacitor, 5x7mm	Max 7mm height. Minimum 16V.
C38	47uF	Electrolytic capacitor, 5mm	
C39	100uF	Electrolytic capacitor, 6.3mm	
C40	100n	MLCC capacitor, X7R	
D1	1N5817	Schottky diode, DO-41	
D2	1N914	Fast-switching diode, DO-35	
D3	1N914	Fast-switching diode, DO-35	
D4	1N914	Fast-switching diode, DO-35	
D5	1N914	Fast-switching diode, DO-35	
Q1	2N5088	BJT transistor, NPN, TO-92	
Q2	2N5088	BJT transistor, NPN, TO-92	
Q3	2N5088	BJT transistor, NPN, TO-92	
Q4	2N3906	BJT transistor, PNP, TO-92	
Q5	2N5088	BJT transistor, NPN, TO-92	
Q6	2N5088	BJT transistor, NPN, TO-92	
Q7	2N5088	BJT transistor, NPN, TO-92	
REG	LM78L05	Regulator, +5V, TO-92	

PARTS LIST, CONT.

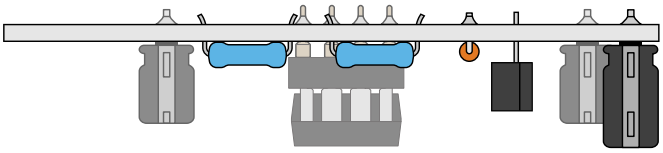
PART	VALUE	TYPE	NOTES
IC1	JRC4558D	Operational amplifier, DIP8	
IC1-S	DIP-8 socket	IC socket, DIP-8	
IC2	JRC4558D	Operational amplifier, DIP8	
IC2-S	DIP-8 socket	IC socket, DIP-8	
IC3	v3207D	BBD, 1024-stage, DIP-8	Available from Aion FX. Can also use MN3207.
IC3-S	DIP-8 socket	IC socket, DIP-8	
IC4	v3102D	Clock driver, DIP-8	Available from Aion FX. Can also use MN3207.
IC4-S	DIP-8 socket	IC socket, DIP-8	
IC5	TL022	Operational amplifier, dual, DIP8	
IC5-S	DIP-8 socket	IC socket, DIP-8	
BIAS	22k trimmer	Trimmer, 10%, 1/4"	Bourns 3362P or similar. Can also be 20k or 25k.
CLOCK	500k trimmer	Trimmer, 10%, 1/4”	Bourns 3362P or similar.
RES. TRIM	22k trimmer	Trimmer, 10%, 1/4”	Bourns 3362P or similar. Can also be 20k or 25k.
SPEED	250kC	16mm right-angle PCB mount pot	
MANUAL	50kB	16mm right-angle PCB mount pot	
DEPTH	50kB	16mm right-angle PCB mount pot	
RESONANCE	50kC	16mm right-angle PCB mount pot	
LED	5mm red	LED, 5mm, red diffused	
IN	1/4" mono	1/4" phone jack, closed frame	Switchcraft 111X or equivalent.
OUT	1/4" mono	1/4" phone jack, closed frame	Switchcraft 111X or equivalent.
DC	2.1mm	DC jack, 2.1mm panel mount	Mouser 163-4302-E or equivalent.
FSW	3PDT	Stomp switch, 3PDT	
ENC	125B	Enclosure, die-cast aluminum	Can also use a Hammond 1590N1.

ASSEMBLY INSTRUCTIONS

The Aerolith uses a stacked "sandwich" PCB design so that it can fit inside a 125B enclosure. It's not difficult to put together, but there's only one right way to do it and several wrong ways that may ruin your build if you're not careful. Make sure you have a good understanding of what the end result should look like before you begin installing any components.

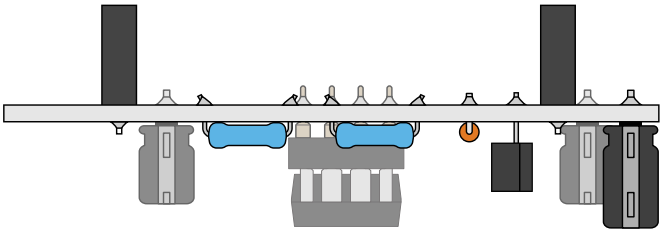
Step 1

Populate the PCBs according to the silkscreen. Unlike most other Aion FX projects, the components mount on the underside of both the main and secondary boards, the same side as the potentiometers. (The components on the bypass PCB mount on the top side as with other projects.)



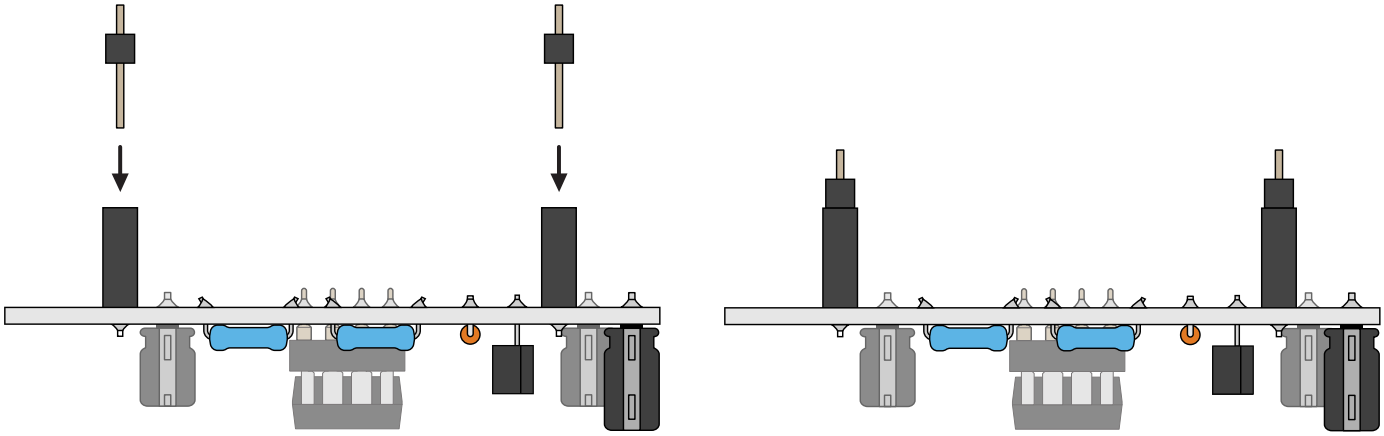
Step 2

Install the header sockets on the bottom PCB. It's recommended to turn the PCB upside down to hold all of them in place while soldering. Solder one leg of each header, then check them from the side to make sure they are straight and perpendicular with the PCB before soldering the remaining legs. If any of them are crooked, reflow the solder and adjust them as needed.



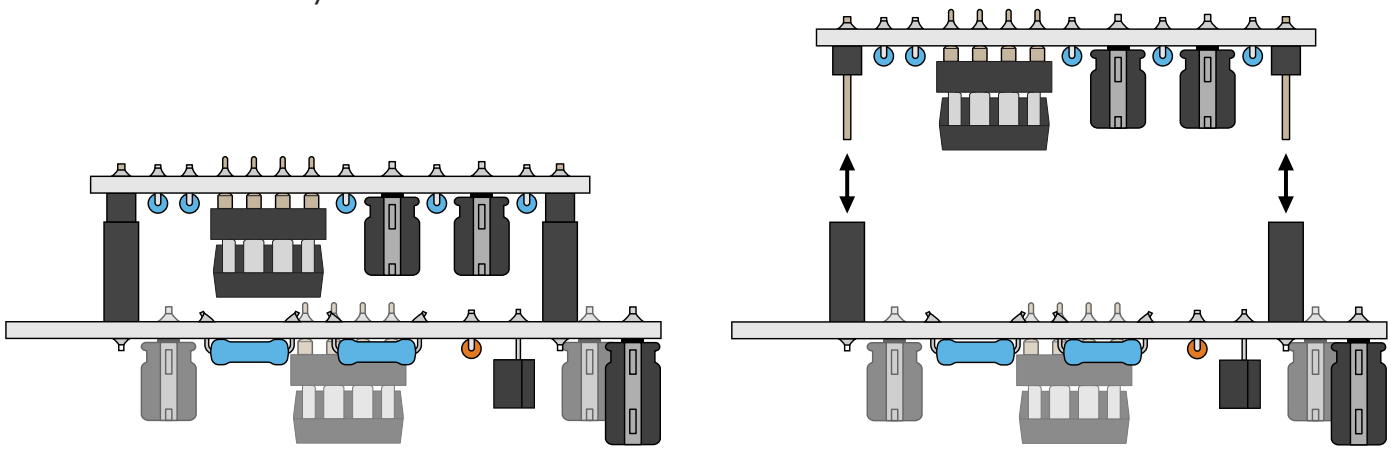
Step 3

With the header sockets installed to the bottom PCB, insert the male headers. The long side goes into the socket and the short side faces up.



Step 4

With the male header sockets in place, put the top PCB in position, components facing down. (The headers and pins should always mount to the side with the rectangular outline on the PCB silkscreen.) Once everything is in place, solder the pins to the top PCB. The top PCB can then be removed and set aside until final assembly.



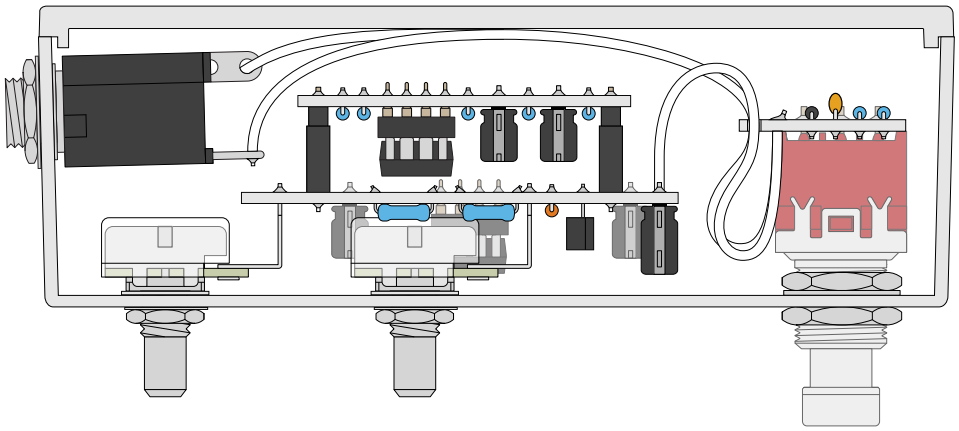
It's done in this order so that that the pins are perfectly coupled with the headers. If they were soldered separately from each other, the slight misalignments between the pins and headers would create stress that could potentially cause cracked solder joints over time.

From here, you can proceed with the rest of the build as normal. It's recommended to first attach the potentiometers to the drilled enclosure and then solder the lower PCB in place.

This way, the enclosure acts as a template that ensures the pots and switch are mounted at the correct height, and it will help compensate for any slight drilling inaccuracies in the enclosure.

Even if you decide to remove the PCB to test outside the enclosure before final boxing, this method will ensure there is no long-term stress on the joints of the PCB-mounted components once everything is reassembled.

Here is a diagram of the completed pedal once it's installed and wired in the enclosure:



BUILD NOTES

Headers and sockets

The Aerolith uses standard pin headers and sockets that are also used in many other types of DIY electronics such as Arduino shields. You'll need four 4-pin female sockets and one snap-apart male header that can be broken in to the matching sizes.

The best ones we've found are from Tayda Electronics. They're cheaper than the ones from Mouser and also make a much tighter connection with more tension. Here are the links:

- 4-Pin Female Header (4 needed)
- 40-Pin Snap-Apart Male Header (1 needed)

Electrolytic capacitor sizes

Due to the clearance between the PCBs, the electrolytic capacitors on the top PCB have a maximum height of 7mm and a maximum diameter of 5mm. If you use the Mouser parts spreadsheet, the listed capacitors are all the correct size, so you can start there if you're having trouble finding them.

All capacitors should be rated for a minimum 16V except for the following:

- **C29** and **C30** (33uF) and **C35** (22uF) can be rated as low as 10V since their operational voltage is at most 5.6V.
- **C28** (47uF) can be a minimum 6.3V since its operational voltage is 2.8V.

If you are having a hard time finding the right sizes in the specified values, then you can go lower (10uF) or higher (33uF or 47uF) for the filter capacitors C34-36. C28-30 should be kept the same.

C27 capacitor type

C27 is specified as a 22uF tantalum capacitor. These are somewhat more expensive than electrolytics, and the original BF-2 used an electrolytic here, so tantalum is not strictly required.

In this position of the circuit, the low ESR (equivalent series resistance) of tantalum is theoretically advantageous compared to an electrolytic. Several circuits with similar LFOs use a tantalum capacitor here, such as the Phase 90 and PH-1R, and even the Ibanez FL-9, which is a near-direct clone of the BF-2.

Whether you use electrolytic or tantalum, C27 can be as low as 10V.

Securing the top PCB

When the pedal is in playing position, gravity will be pulling against the top PCB, and it could potentially be knocked loose with enough shock. Once the pedal is fully biased and tested, you may want to attach some non-conductive adhesive foam to the inside of the lid that is thick enough to press down against the PCB when it's closed. Make sure the offboard wires are routed around it so that they aren't pressed up against the PCB.

Alternately, you could also use some hot glue on the headers, or any other methods you may come up with. Just ensure you've secured it somehow if you're planning on using the pedal in a live environment.

BBD selection

The original bucket-brigade delay chips used in the BF-2 have been discontinued for many years and old-stock chips are getting difficult to find. Fortunately, reproductions of the MN3207 are available along with the corresponding clock driver, and they perform just as well as the original devices in this circuit.

The Aerolith project is set up to allow either the MN3207 (v3207) or MN3007 to be used by soldering two jumpers in one configuration or another.

Similar flangers such as the Ibanez FL-301 used the MN3007/MN3101. However, the original BF-2 only ever used the MN3207, so unless you want to experiment, we recommend sticking with this configuration.

The MN3007 is capable of higher voltages, which means that if you know what you're doing, you could use a higher-voltage regulator for the BBD supply. Just be aware that the surrounding circuitry and LFO may need some adjustments if you change the operating voltage.

Clock selection

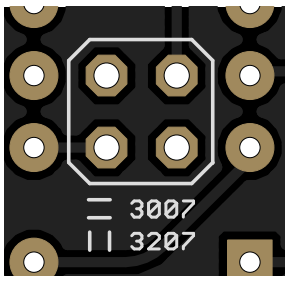
The MN3101 clocks are only compatible with the MN3007, and the MN3102 and v3102 clocks are only compatible with the MN3207 and v3207.

In late 2023, Cabintech made a discrete version of the MN3101 that acts as a drop-in replacement in DIP-8 format. These perform just as well as the originals, so it's recommended to use them in conjunction with the MN3007. We do not recommend trying to source old-stock MN3101s due to the risk of fakes or damaged parts.

Setting the jumpers

Underneath IC3, there are four jumper pads arranged in a square pattern. The MN3007 and MN3207 have their positive and negative supply pins inverted from each other, so the jumpers need to be set to route the supply voltages to the correct pins.

In the initial run of Aerolith PCBs, there is no silkscreen marking for the jumpers, so see the diagram to the right for which way the jumpers should be soldered. If using a MN3007, both jumpers should go horizontally. If using a MN3207 or v3207, the jumpers should go vertically. This silkscreen legend will be added to subsequent reorders of the project.



The MN3101 and MN3102 clock drivers have the supply pins reversed internally, so these two chips are cross-compatible and IC4 does not require any jumpers.

Setting the clock trimmer

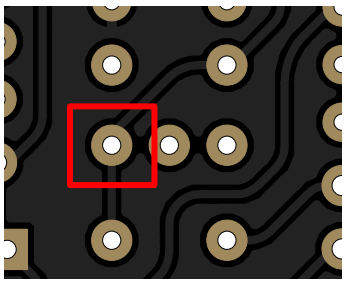
For this, you will need an oscilloscope or digital multimeter with a frequency counter function. The BF-2 service manual specifies that the clock frequency should be 40kHz with the Manual control at minimum. Set all four potentiometers to minimum and then use the "TEST" pads to measure the frequency. (It may help to solder short wires to these pads if your test probes can't easily fit between the two trimmers.) The top pad is ground and the bottom pad is the clock output, so if you're using a different ground point then you only need to use the bottom pad.

Adjust the CLOCK trimmer until you read 40kHz. Then, turn Manual all the way up and check to see that it reads 500kHz +/- 20%. In our prototype unit, it was around 586kHz, which is within spec. But if it's a little outside, don't worry too much about it—it just sets the maximum and minimum range of the Manual knob and was primarily just the manufacturer's means of ensuring consistency between units.

Setting the bias trimmer by ear

The BIAS trimmer adjusts the bias voltage to get optimum audio performance from the BBD. Set all four controls to minimum. Start with the bias at halfway, then adjust it left or right until you hear the least audible distortion. In our prototype unit, it was almost exactly 12:00.

Setting the bias trimmer with an oscilloscope



If you have an oscilloscope, you can look at the waveform and fine-tune the bias visually rather than by ear. Connect the oscilloscope to the emitter of Q3 (the far-left pad as you're looking from the top, highlighted in red in the diagram to the right). Use a signal generator to create a sine wave at 200 Hz, 3V peak to peak, and send this to the pedal's input.

Set all four controls to minimum. Then, adjust the BIAS trimmer until the waveform is equally clipped on the top and bottom. The waveform is not symmetrical, so you're looking for equal-width clipping but not necessarily equal-shape clipping.

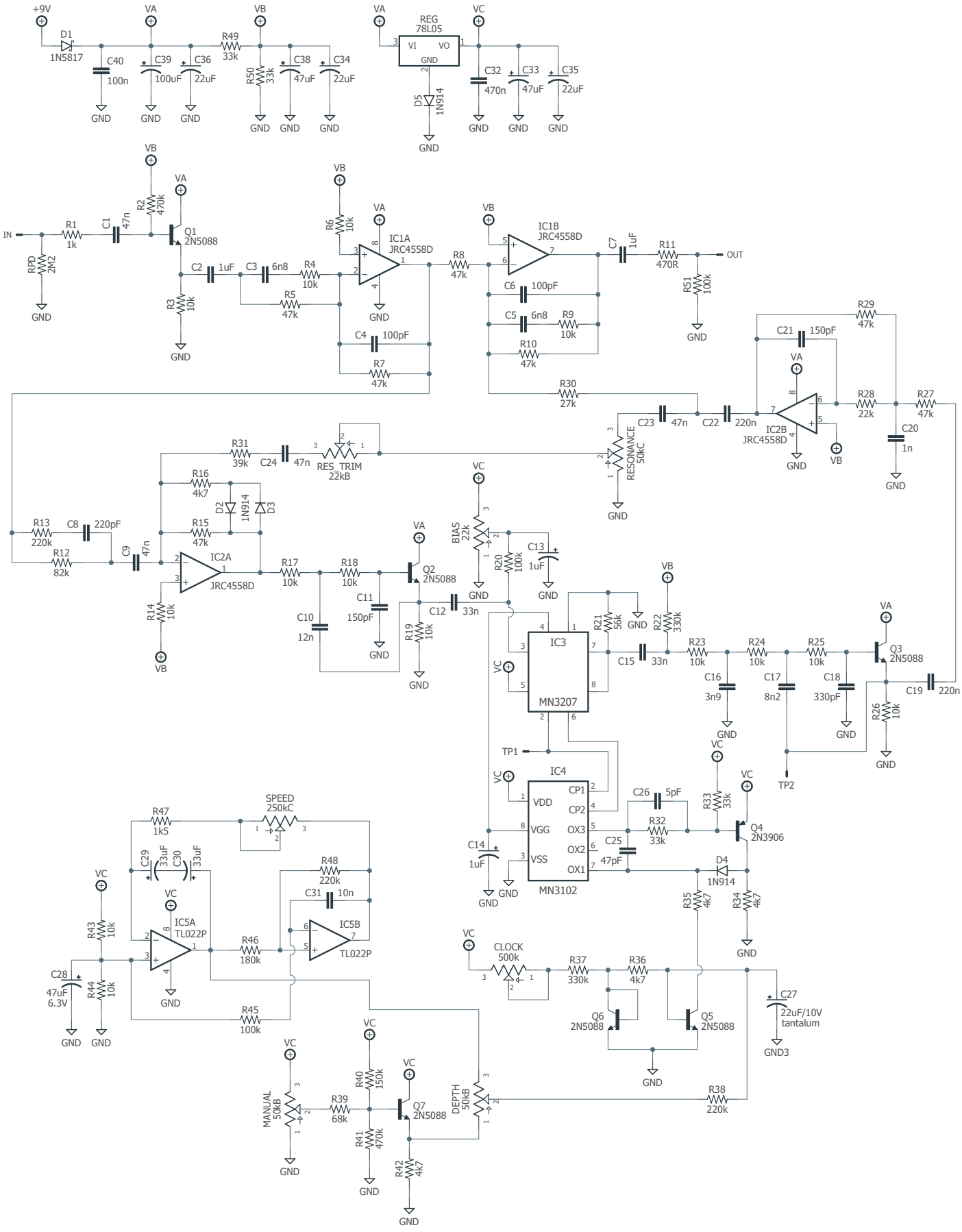
The BF-2 service manual specifies a test signal amplitude of 0dBm, which is around 600mV peak to peak. However, in prototyping, we found that the test signal amplitude needed to be significantly higher to get enough clipping that the bias could be reliably adjusted, which is why we've recommended 3V. The test signal is intended to be much larger than what the pedal will deal with in normal use, so the waveform will not clip like this in practice. It's just easier to optimize the symmetry when it's clipping.

Setting the Resonance trimmer

The Resonance trimmer is designed to limit the maximum setting of the external Resonance knob to prevent oscillation. Start by setting RES. TRIM on the top PCB to minimum and the Resonance knob to full. The Manual, Rate and Depth controls should be set to minimum.

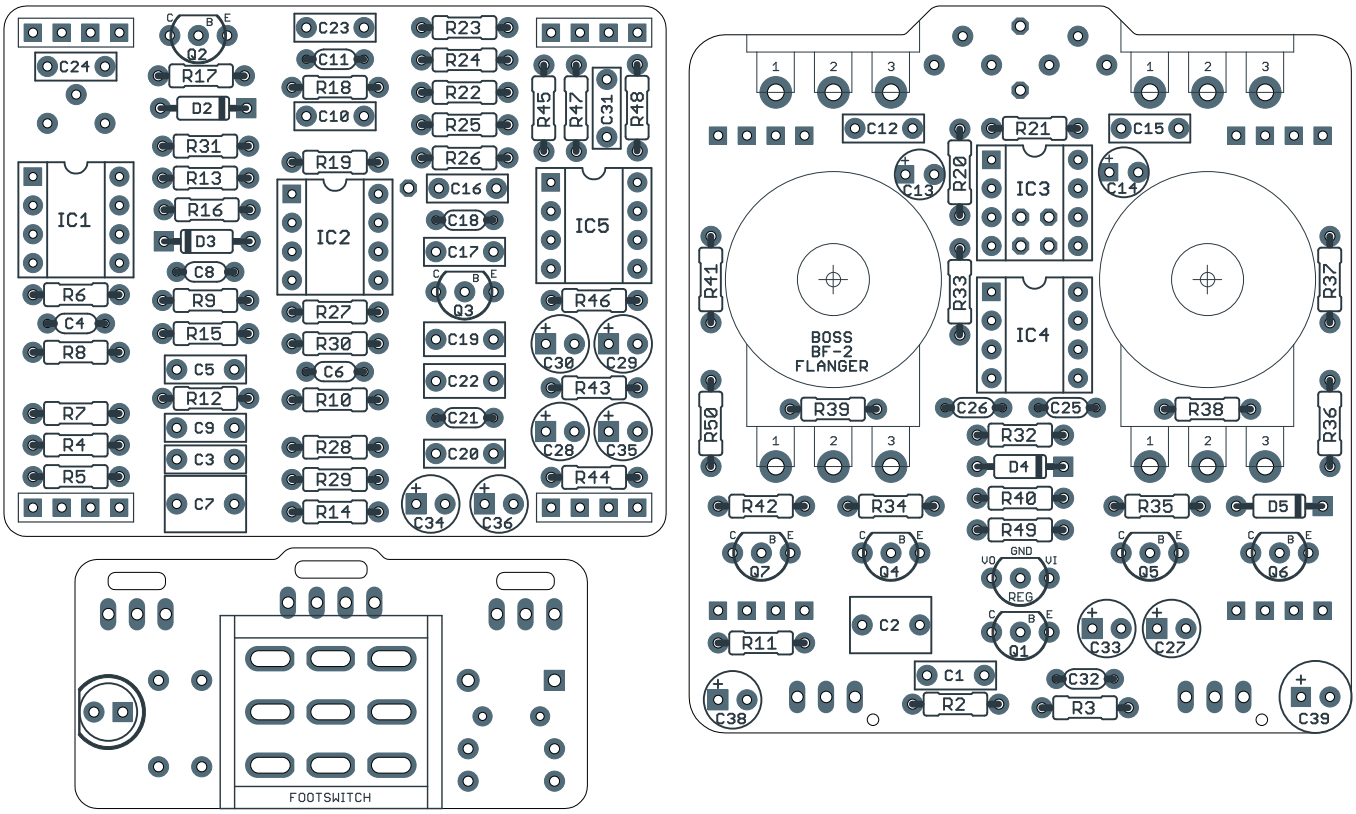
Now, turn on the pedal with the output to an amplifier and nothing plugged into the input jack. Slowly turn up RES. TRIM until you hear it start to oscillate. Then back off slightly until the oscillation goes away. This setting is purely to taste, so you may want to leave a tiny bit of oscillation at the top end of the range if you want to be able to access the out-of-control sound.

SCHEMATIC


+9V
D1
1N5817
C40
100n
VA
C39
100uF
C36
22uF
R49
33k
VB
R50
33k
C38
47uF
C34
22uF
REG
78L05
VI
VO
GND
VC
C32
470n
C33
47uF
C35
22uF
D5
1N914
IN
R1
1k
C1
47n
R2
470k
Q1
2N5088
RPD
2M2
R3
10k
C2
1uF
C3
6n8
R4
10k
R5
47k
R6
10k
IC1A
JRC4558D
C4
100pF
R7
47k
R8
47k
IC1B
JRC4558D
C6
100pF
C5
6n8
R9
10k
R10
47k
C7
1uF
R11
470R
R51
100k
OUT
R29
47k
C21
150pF
R30
27k
C23
47n
C22
220n
IC2B
JRC4558D
R28
22k
R27
47k
C20
1n
R31
39k
C24
47n
RES_TRIM
22kB
RESONANCE
50kC
R16
4k7
D2
1N914
D3
R13
220k
C8
220pF
R15
47k
R12
82k
C9
47n
IC2A
JRC4558D
R14
10k
R17
10k
R18
10k
Q2
2N5088
C10
12n
C11
150pF
C12
33n
R19
10k
BIAS
22k
R20
100k
C13
1uF
IC3
MN3207
R21
56k
R22
330k
C15
33n
R23
10k
R24
10k
R25
10k
Q3
2N5088
C16
3n9
C17
8n2
C18
330pF
C19
220n
R26
10k
TP1
TP2
IC4
MN3102
VDD
VGG
VSS
CP1
CP2
OX3
OX2
OX1
C26
5pF
R32
33k
C25
47pF
C14
1uF
R33
33k
Q4
2N3906
D4
1N914
R35
4k7
R34
4k7
SPEED
250kC
R47
1k5
C29
33uF
C30
33uF
R48
220k
C31
10n
IC5A
TL022P
IC5B
TL022P
R43
10k
R46
180k
R44
10k
C28
47uF
6.3V
R45
100k
CLOCK
500k
R37
330k
R36
4k7
Q6
2N5088
Q5
2N5088
C27
22uF/10V
tantalum
GND3
MANUAL
50kB
R39
68k
R40
150k
Q7
2N5088
DEPTH
50kB
R41
470k
R42
4k7
R38
220k
GND

PCB DIAGRAM (REVERSE SIDE)

The diagram on page 1 shows the front or top-facing side of the PCB, but since very few of the components are actually mounted on the front, it may be helpful to have a reference for the bottom side of the PCB as well.



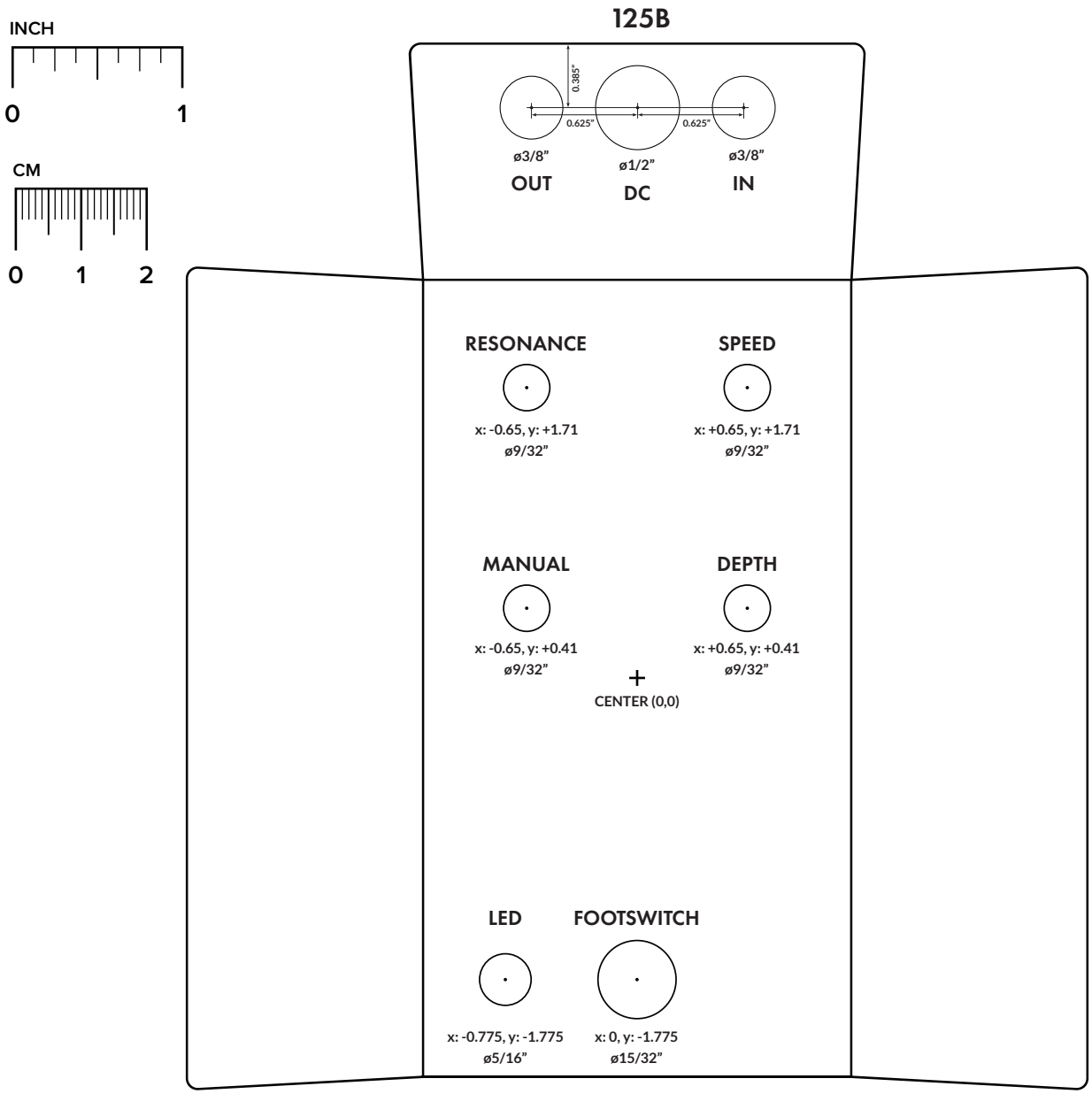
DRILL TEMPLATE

Cut out this drill template, fold the edges and tape it to the enclosure. Before drilling, it's recommended to first use a center punch for each of the holes to help guide the drill bit.

Ensure that this template is printed at 100% or "Actual Size". You can double-check this by measuring the scale on the printed page.

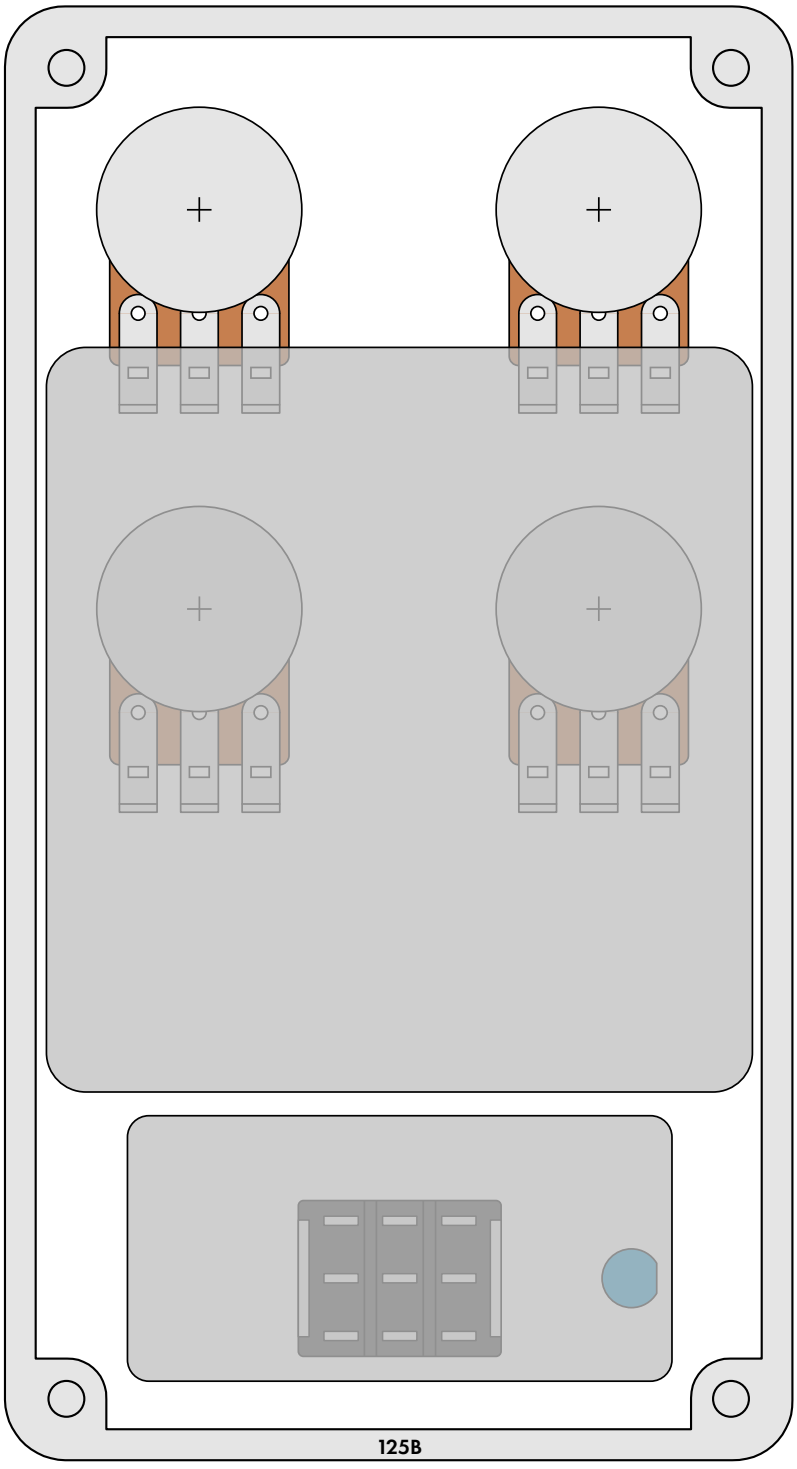
Top jack layout assumes the use of closed-frame jacks like the Switchcraft 111X. If you'd rather use open-frame jacks, please refer to the Open-Frame Jack Drill Template for the top side.

LED hole drill size assumes the use of a 5mm LED bezel, available from several parts suppliers. Adjust size accordingly if using something different, such as a 3mm bezel, a plastic bezel, or just a plain LED.

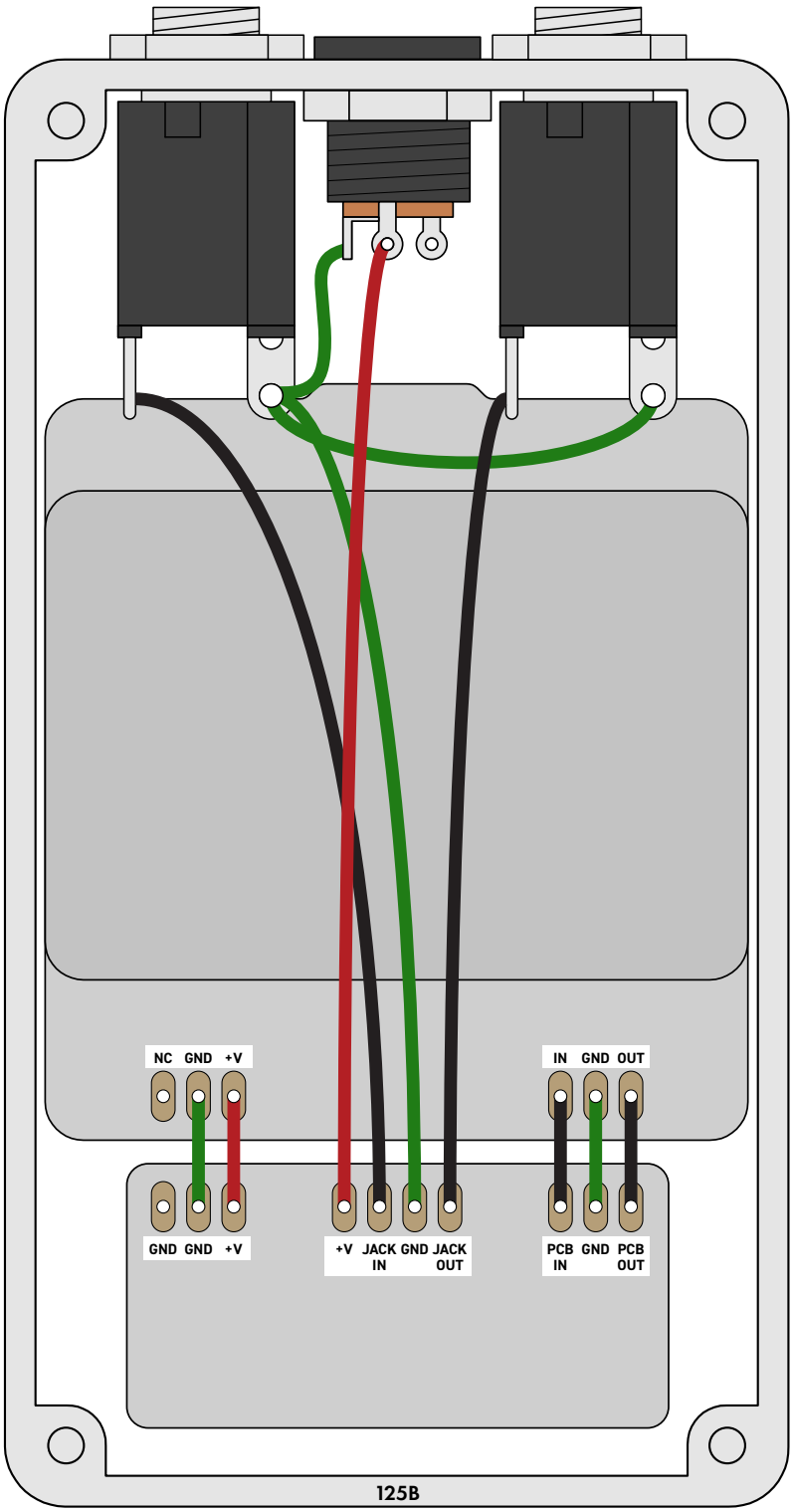


ENCLOSURE LAYOUT

Enclosure is shown without jacks and top PCB. See next page for jack layout and wiring, and see page 7 for a full side-profile view of the assembled pedal.



WIRING DIAGRAM



Note: The footswitch PCB has two ground wires, one for LFO ground and one for signal ground. Ensure that both grounds are connected or the effect will not work. The only wire that can be left disconnected is the one on the far-left side.

LICENSE & USAGE

No direct support is offered for these projects beyond the provided documentation. It's assumed that you have at least some experience building pedals before starting one of these. Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error.

All of these circuits have been tested in good faith in their base configurations. However, not all the modifications or variations have necessarily been tested. 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 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 without prior arrangement**, 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 industry needs more transparency, not less!)

DOCUMENT REVISIONS

1.0.0 (2026-07-03)
Initial release.