PROJECT NAME

CADMUS

aion DIY GUITAR EFFECTS

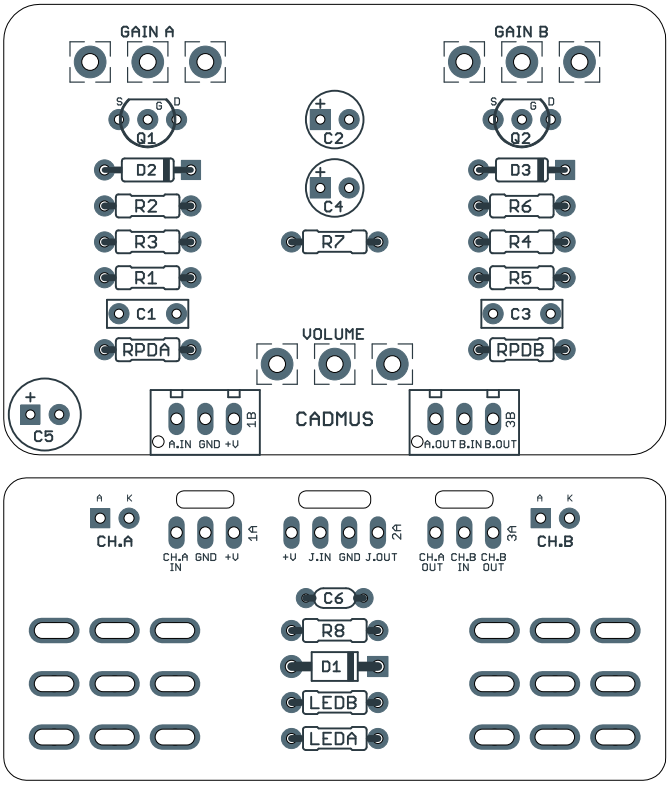
BASED ON
ZVEX Super Duper

BUILD DIFFICULTY
▮▮▯▯▯ Easy

EFFECT TYPE
Dual-stage boost

DOCUMENT VERSION
1.0.0 (2023-03-24)

PROJECT SUMMARY
A two-channel boost composed of identical MOSFET gain stages. The boosts can be cascaded in series or selected independently.



Actual size is 2.3” x 1.57” (main board) and 2.3” x 1.05” (bypass board).

TABLE OF CONTENTS

INTRODUCTION

The Cadmus Dual Boost is based on the ZVEX Super Duper, which is two of their SHO boost circuits in series and independently switchable.

The SHO is an extremely simple MOSFET booster, small enough that even two of them together have fewer components than most single effects.

The Super Duper is similar in concept to the Box of Rock, which adds a 3rd SHO stage as well as a tone control. That effect is more of an overdrive while this is a two-stage clean boost, though with both stages engaged this one can get into clipping if desired.

The Cadmus is a direct clone of the Super Duper with no added features other than pull-down resistors on each input to help prevent popping.

Note that since the channels are identical to individual SHO circuits, the two gain controls exhibit the characteristic “crackle” of the SHO when they are adjusted, since they are directly biasing the MOSFETs. This is normal.

USAGE

The Cadmus has three knobs:

- **Gain A** controls the amount of gain in the first boost.
- **Gain B** controls the amount of gain in the second boost.
- **Volume** controls the gain at the output of the second boost stage. It is disabled when Channel B is disengaged.

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—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	10M	Metal film resistor, 1/4W	
R2	10M	Metal film resistor, 1/4W	
R3	5k1	Metal film resistor, 1/4W	
R4	100k	Metal film resistor, 1/4W	
R5	10M	Metal film resistor, 1/4W	
R6	10M	Metal film resistor, 1/4W	
R7	5k1	Metal film resistor, 1/4W	
R8	100R	Metal film resistor, 1/4W	Power supply filter resistor. Labeled R23 in original release.
RPDA	2M2	Metal film resistor, 1/4W	Ch. A pull-down resistor. Can be as low as 1M.
RPDB	2M2	Metal film resistor, 1/4W	Ch. B pull-down resistor. Can be as low as 1M.
LEDA	10k	Metal film resistor, 1/4W	Ch. A LED resistor. Adjust value to change LED brightness.
LEDB	10k	Metal film resistor, 1/4W	Ch. B LED resistor. Adjust value to change LED brightness.
C1	100n	Film capacitor, 7.2 x 2.5mm	
C2	10uF	Electrolytic capacitor, 5mm	
C3	100n	Film capacitor, 7.2 x 2.5mm	
C4	10uF	Electrolytic capacitor, 5mm	
C5	100uF	Electrolytic capacitor, 6.3mm	Power supply filter capacitor. Labeled C17 in original release.
C6	100n	MLCC capacitor, X7R	Power supply filter capacitor. Labeled C18 in original release.
D1	1N5817	Schottky diode, DO-41	
D2-D3	1N5239B	Zener diode, 9.1V, DO-35	Can sub any 9.1V zener, such as 1N4739A.
Q1-Q2	BS170	MOSFET, N-channel, TO-92	Can sub 2N7000, but rotate 180 degrees.
GAIN A	5kC	16mm right-angle PCB mount pot	
GAIN B	5kC	16mm right-angle PCB mount pot	
VOL.	50kB	16mm right-angle PCB mount pot	
A.LED	5mm	LED, 5mm, red diffused	
B.LED	5mm	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.
A.FSW	3PDT	Stomp switch, 3PDT	
B.FSW	3PDT	Stomp switch, 3PDT	
ENC	125B	Enclosure, die-cast aluminum	Can also use a Hammond 1590N1.

BUILD NOTES

MOSFET selection and pinouts

The original Super Duper uses BS170 MOSFETs, which is what this layout is designed for. These are identical in specification to the 2N7000, but the 2N7000's pinout is reversed. If using the 2N7000, rotate it 180 degrees from the orientation shown on the PCB silkscreen.

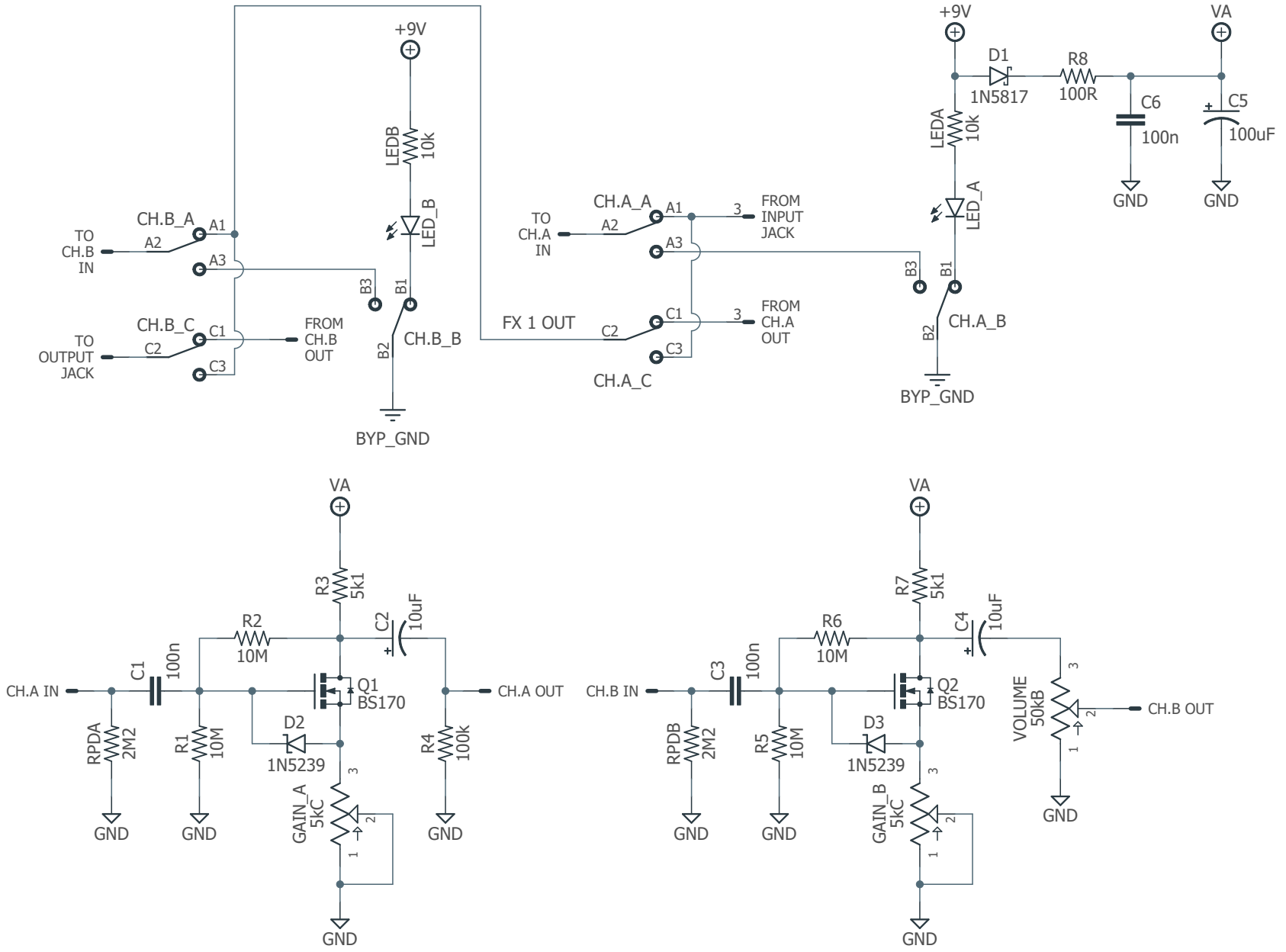
R23, C18 and C19

The initial run of Cadmus PCBs had a few parts on the PCB mislabeled as follows.

- C5 (main board) is labeled C18. It should be 100uF.
- C6 (footswitch board) is labeled C19. It should be 100n.
- R8 (footswitch board) is labeled R23. It should be 100R.

The PCB diagram on page 1 shows the correct numbering, as well as the schematic on the next page.

SCHEMATIC


+9V
LEDB
10k
LED_B
CH.B_A
A1
A2
A3
TO
CH.B
IN
CH.B_C
C1
C2
C3
TO
OUTPUT
JACK
FROM
CH.B
OUT
B3
B1
B2
CH.B_B
BYP_GND
FX 1 OUT
CH.A_A
TO
CH.A
IN
FROM
INPUT
JACK
CH.A
OUT
CH.A_C
+9V
D1
1N5817
R8
100R
C6
100n
VA
C5
100uF
GND
LEDA
10k
LED_A
CH.A_B
BYP_GND
VA
R3
5k1
R2
10M
C2
10uF
C1
100n
CH.A IN
Q1
BS170
CH.A OUT
RPDA
2M2
R1
10M
D2
1N5239
R4
100k
GAIN_A
5kC
GND
VA
R7
5k1
R6
10M
C4
10uF
C3
100n
CH.B IN
Q2
BS170
RPDB
2M2
R5
10M
D3
1N5239
GAIN_B
5kC
VOLUME
50kB
CH.B OUT
GND

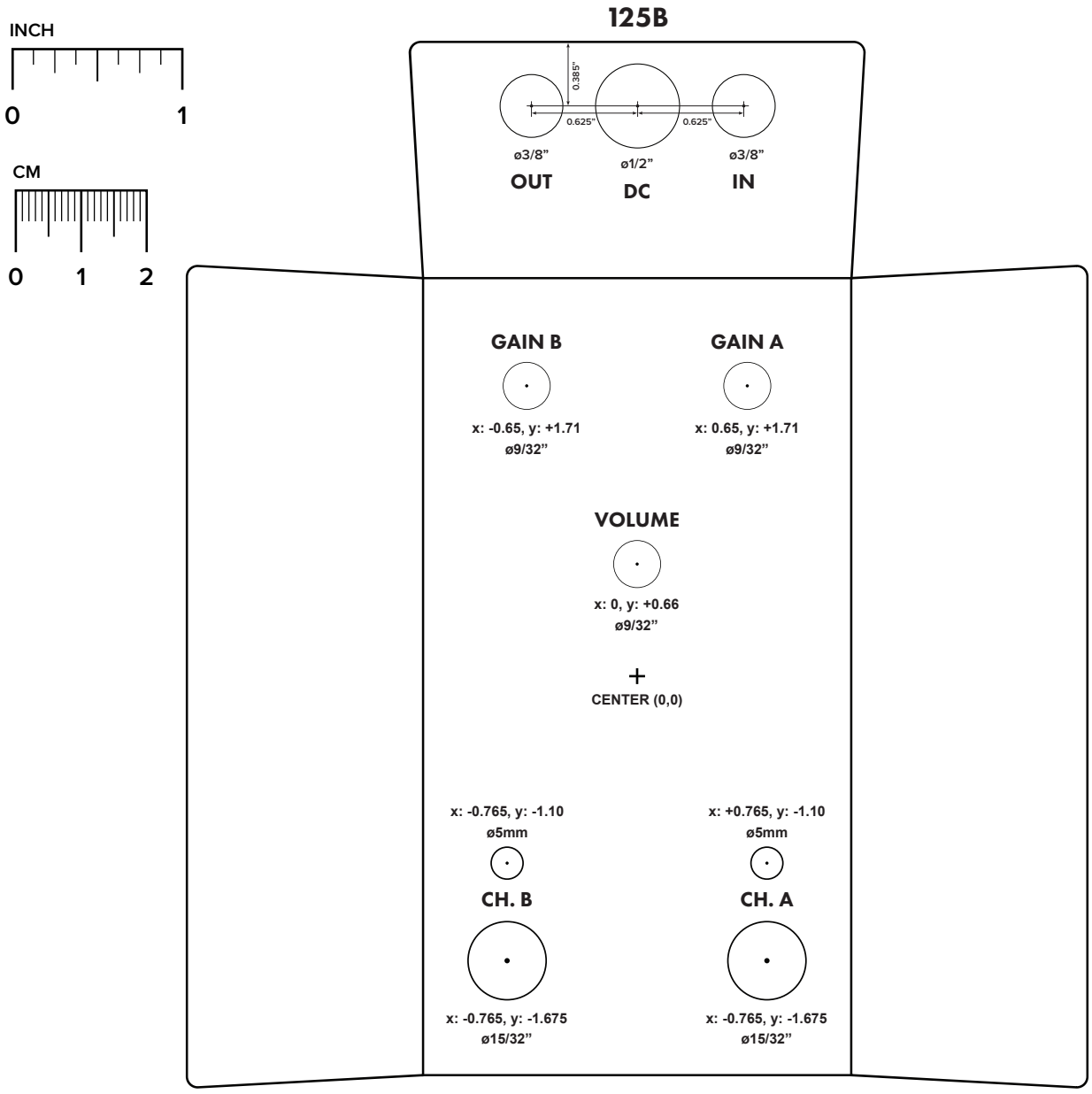
DRILL TEMPLATE

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.

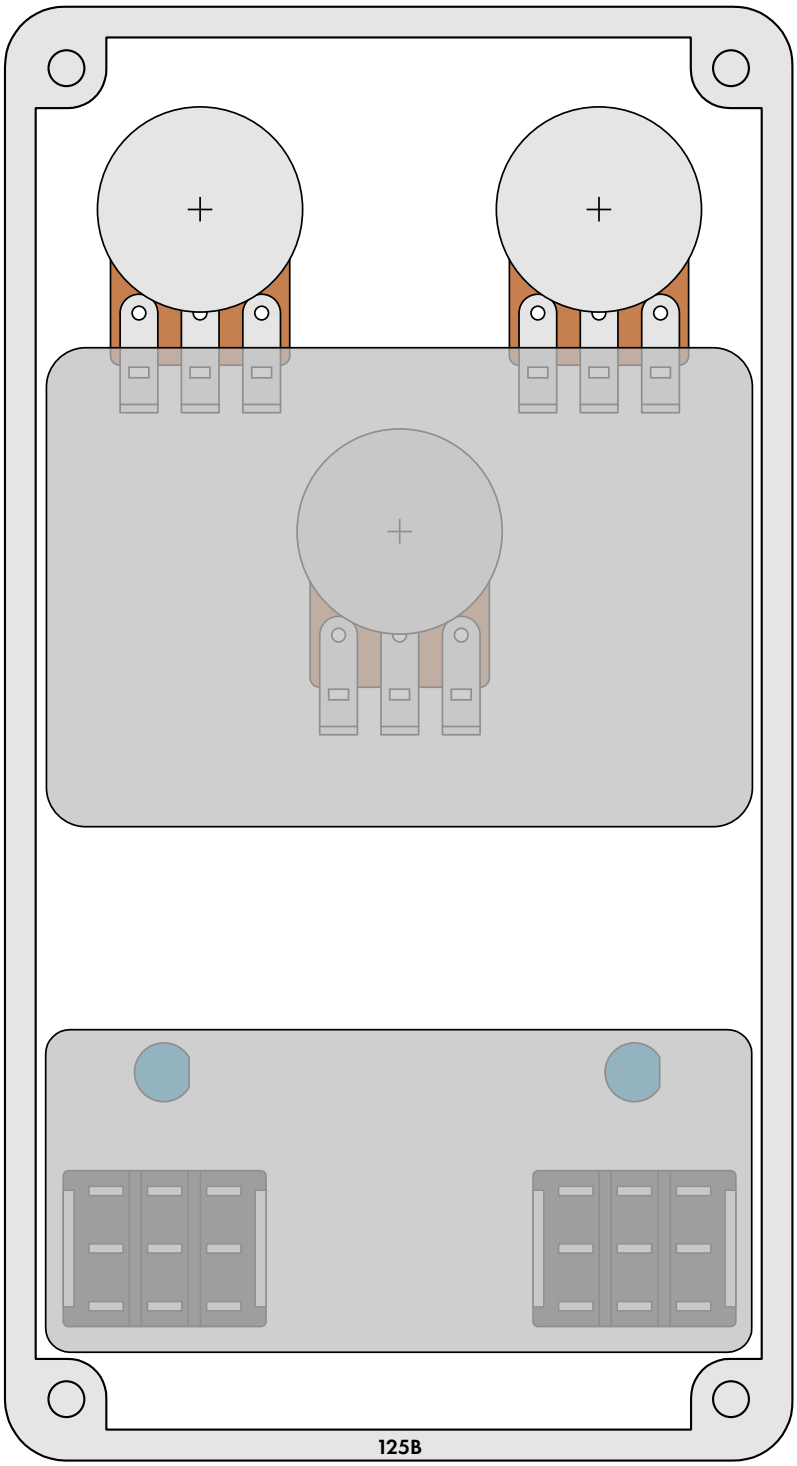
Top jack layout assumes the use of closed-frame jacks like the Switchcraft 111X. If you'd rather use open-frame jacks, please refer to the Open-Frame Jack Drill Template for the top side.

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.

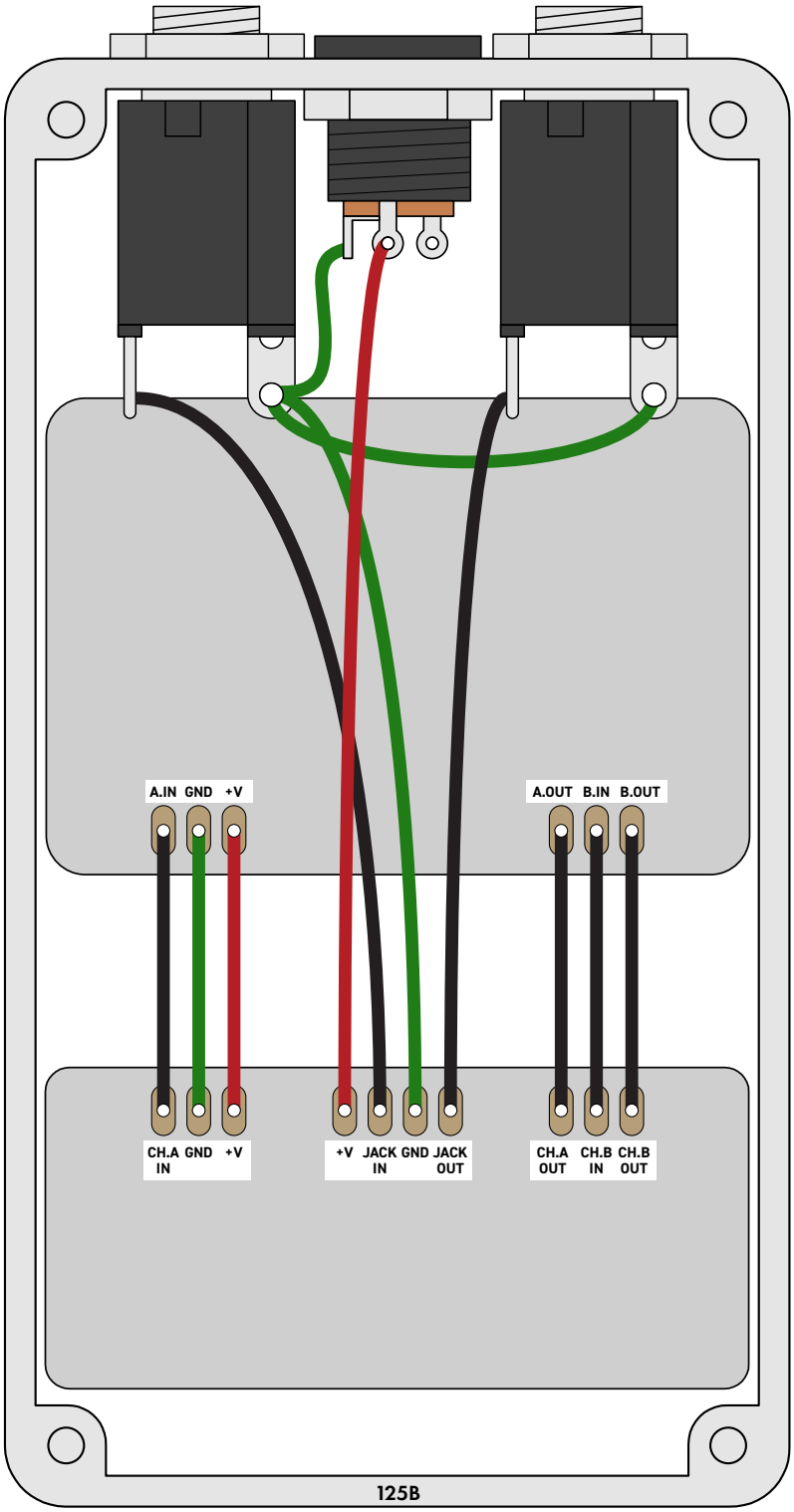


ENCLOSURE LAYOUT

Enclosure is shown without jacks. See next page for jack layout and wiring.



WIRING DIAGRAM



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 cannotbe 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 (2023-03-24)
Initial release.