WARNING / DISCLAIMER

PLEASE READ THE FOLLOWING BEFORE PURCHASING OR BUILDING THIS PROJECT!

1. NO DIRECT TECHNICAL SUPPORT: This project has been verified to be working. I have done my best to provide extremely thorough documentation, including information to help you troubleshoot. But on a project this size, I have to reiterate that **Aion Electronics cannot provide direct technical support for this project** or others. I love helping people bring these circuits to life, but my availability is very limited. If you post your question on one of the DIY forums and send me a link, I will do my best to chime in. Just know before purchasing this PCB that **there is no implied guarantee of the final product**, because the biggest factor is outside my control: **you!** Your experience and your attention to detail are the most important ingredients in making sure this works. My role is to provide the recipe and some cooking utensils.

2. IT WILL TAKE AWHILE: Be prepared to invest some hours into putting this together. You'll want to be doubly careful when populating the board since it'll be much more difficult to track down a problem if you were to make even the most basic of mistakes (for instance, accidentally using a 100k resistor somewhere instead of a 10k). **You can't be too cautious.** I'd recommend measuring each resistor with a multimeter before putting it into place. Triple-check your wiring. The more time you spend on the initial build, the less time you'll have to spend troubleshooting.

3. IT'S COMPLICATED: This is a very complex circuit in a very small enclosure and it takes experience and a lot of attention to detail in order to pull it off. Hopefully it goes without saying, but **if you've never built a guitar pedal before, this shouldn't be your first**. If you haven't successfully built at least ten or fifteen, including a few choruses or delays, you may not be ready for this one yet.

4. YOU'VE GOT TO BUILD IT AS IT WAS INTENDED: I approached the project as though I was designing a completed product for market. Everything has been designed to be built using methods you'd see in a high-end pedal (for instance, PCB-mounted switches, a standoff for mounting stability, and components with specific sizes and characteristics) and a full bill of materials has been provided so that it is very easy to order all the parts from Mouser. We all build pedals in our own style, but with this one, if you try to "freestyle" by doing your own enclosure layout or using parts other than the ones specified, you might back yourself into a corner. Please do things my way—you'll end up with a very professional and durable end product and you might even learn a few things in the process!

5. IT'S NOT CHEAP: Between the PCB, enclosure, hardware, potentiometers, and the on-board components, expect to spend a minimum of **$100 USD** and probably closer to $125. Please don't try to cut corners on the parts selection by using poor-quality components or by substituting "close enough" components that you have laying around. You're putting a lot of time and effort into this build, so it's worth a few extra dollars to use the right parts. Expect to order from more than one web store to get everything you need.

6. IT NEEDS TO BE BIASED: This pedal will not function properly without biasing the BBDs. It can be done by ear with passable results, but for optimum sound quality and noise control, it requires an oscilloscope. If you don't have one, be prepared to either buy one or to make friends with someone who does own one.

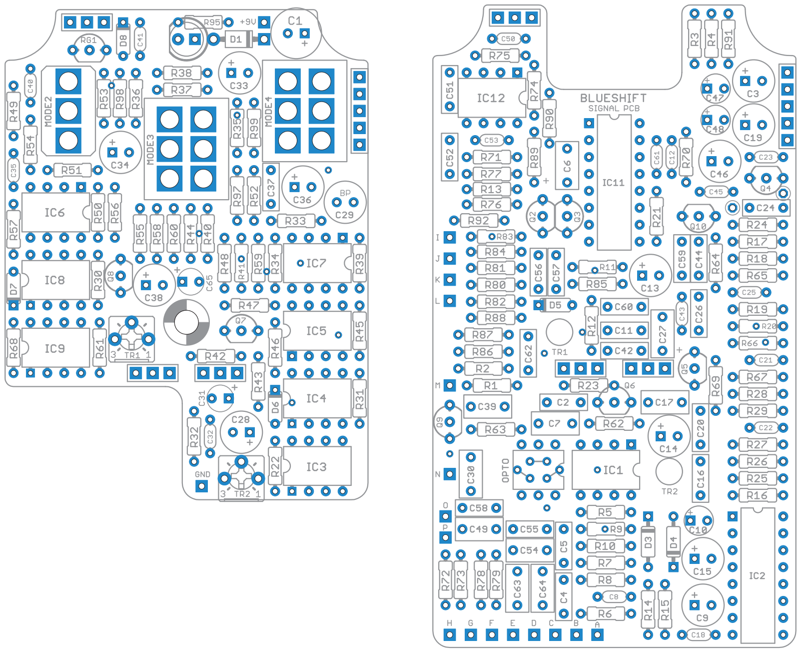
Now that you've been properly warned: on to the fun stuff!

Blueshift Chorus

BOSS DC-2 Dimension C

Overview

Blueshift Project Link



See pg. 8 for a larger image of the PCB layout.

This is a faithful clone of the BOSS DC-2 Dimension C, widely regarded as the best stompbox chorus ever made as well as being one of the most coveted Boss pedals out there. Originally manufactured from 1985 to 1989, it was a guitar pedal adaptation of the revered Roland SDD-320 Dimension D rack unit.

It's different from other choruses in that there are actually two clock+BBD pairs, each powered by either the inverted or non-inverted output of a single LFO. These BBDs then modulate the dry signal so that signal "A" reaches its maximum delay time when signal "B" reaches its minimum and vice versa. Since the average remains constant, this effect is referred to as *motionless* because it doesn't have the signature "warble" or unsteadiness of a traditional chorus with a single BBD.

Table of contents

Usage

On one end of the guitar pedal spectrum, you've got something like the Fuzz Factory where half of the components in the circuit are potentiometers and you have far more control over the device than you could ever want. The DC-2 is the very opposite: it has an obscene number of parts crammed into a small box and you only get three toggle switches to control it.

Because of this, usage is pretty basic: just plug in a guitar, send the output to either one amp or two, and flip the toggle switches until you like the sound. In general, the lower presets are more subtle and the higher presets are more noticeable, but it doesn't ever get too extreme.

Stereo operation

The DC-2 was designed to accept a mono input and split it into stereo, with one BBD controlling each side of the stereo signal. However, the unit can also mix the signal back down to mono if you are not running a stereo configuration.

If you have a cable plugged into output "B", the signal will be split into stereo between "A" and "B". If nothing is plugged into output "B", the signal will be summed to mono.

Differences from the original circuit

The Blueshift is faithful to the original Dimension C circuit, with only the following changes.

Radio buttons

The original unit had what are called "radio" buttons, meaning that when a button is pressed, it pops all of the others up so that only one can be down at a time. Since these radio buttons are highly customized and outside the reach of DIYers, I converted them to toggle switches.

The function is the same, but you'll notice there are only three of these switches while the original unit has four. This is because the first button on the DC-2 does not actually connect anything—it only resets the other buttons. Accordingly, in the Blueshift, if all of the toggles are down (off) then that is the equivalent of preset 1. If the first toggle is up, that's preset 2, and so on.

The advantage of this setup over the original is that you can combine the modes to produce a total of seven different settings.

Bypass

The original unit uses the standard BOSS flip-flop bypass. The Blueshift has been designed to use a stomp switch and opto-FET in a configuraton that allows you to choose from two different bypass options. The first method mimics the signal path in the original, while the second method only passes the signal through a single op-amp buffer before splitting it into effect and output.

See page 15 for more information on these bypass methods.

Parts

Resistors

R1	1M
R2	100k
R3	10k
R4	10k
R5	1M
R6	10k
R7	47k
R8	10k
R9	100k
R10	33k
R11	2k2
R12	68k
R13	10k
R14	10k
R15	10k
R16	100k
R17	10k
R18	10k
R19	10k
R20	680k
R21	680k
R22	56k
R23	10k
R24	10k
R25	10k
R26	10k
R27	10k
R28	10k
R29	10k
R30	100R
R31	100R
R32	100k
R33	33k
R34	100k
R35	100k
R36	330k
R37	180k
R38	22k
R39	33k
R40	10k
R41	1k5
R42	8k2

Resistors

R43	6k8
R44	4k7
R45	47k
R46	100k
R47	5k6
R48	47k
R49	4k7
R50	100k
R51	47k
R52	100R
R53	100R
R54	5k6
R55	8k2
R56	1k5
R57	6k8
R58	10k
R59	10k
R60	10k
R61	100k
R62	10k
R63	10k
R64	10k
R65	10k
R66	10k
R67	680k
R68	56k
R69	680k
R70	10k
R71	33k
R72	1k
R73	100k
R74	10k
R75	47k
R76	47k
R77	10k
R78	100k
R79	1k
R80	180K
R81	180K
R82	47k
R83	33k
R84	33k

Resistors

R85	47k
R86	39k
R87	220k
R88	220k
R89	1M
R90	1M
R91	1M
R92	33k
R95	3k9
R97	470k
R98	82k
R99	220k

ICs

IC1	OPA2134 [1]
IC2	NE570/571 [2]
IC3	MN3207 [3]
IC4	MN3102 [3]
IC5	TL072
IC6	TL072
IC7	TL022
IC8	MN3102 [3]
IC9	MN3207 [3]
IC11	NE570/571 [2]
IC12	OPA2134 [1]
OPTO	H11F1
RG1	78L06

Transistors

Q2	2N5088
Q3	J113
Q4	2N5088
Q5	2N5088
Q6	2N5088
Q7	2N5087
Q8	2N5087
Q9	2N5088
Q10	2N5088

Parts

Capacitors

C1	100uF electro
C2	47n
C3	47uF electro
C4	47n
C5	4n7
C6	1uF film
C7	1uF film
C8	100pF MLCC
C9	10uF electro
C10	0.47uF tantalum
C11	22n
C12	100pF MLCC
C13	10uF electro
C14	10uF electro
C15	10uF electro
C16	1n
C17	1uF film
C18	100pF MLCC
C19	10uF electro
C20	1n8
C21	220pF MLCC
C22	470pF MLCC
C23	220pF MLCC
C24	1n8
C25	470pF MLCC
C26	22n
C27	1uF film
C28	10uF electro
C29	4u7 bipolar electr.
C30	220n
C31	1uF electro
C32	100pF MLCC
C33	10uF electro

Capacitors

C34	10uF electro
C35	100pF MLCC
C36	47uF electro
C37	10n
C38	10uF electro
C39	1uF film
C40	100n MLCC
C41	100n MLCC
C42	22n
C43	470pF MLCC
C44	1n8
C45	220pF MLCC
C46	10uF electro
C47	0.47uF tantalum
C48	0.47uF tantalum
C49	1uF film
C50	47pF MLCC
C51	4n7
C52	4n7
C53	47pF MLCC
C54	1uF film
C55	3n3
C56	1n2
C57	1n2
C58	3n3
C59	18n
C60	18n
C61	100pF MLCC
C62	18n
C63	1uF film
C64	1uF film
C65	1uF electro

Potentiometers

TR1	100k (3362P)
TR2	100k (3362P)

Switches

MODE2	SPDT toggle [4]
MODE3	DPDT toggle [4]
MODE4	DPDT toggle [4]

Diodes

D1	1N4002
D3	1N5225
D4	1N5225
D5	1N914
D6	1N914
D7	1N914
D8	1N914
LED	5mm or 3mm

Jacks

IN	111X
OUT A	111X
OUT B	112BX (TRS)
DC JACK	miniature [5]

Other hardware

0.625" hex standoff, 6-32 thread (qty 1): This is used to secure the PCB to the enclosure so that stress is not placed on the solder joints of the toggle switches.

1/4-40 hex nut (qty 6): These are used to space the inside nuts of the toggles to match the standoff.

6-32 x 3/8" pan head screw (qty 2): This is used to secure the PCB and enclosure to the hex standoff.[6]

Pin headers: These are required, but there are a couple of options. See next page for details.

[1] **Dual op-amps:** The original DC-2 uses SIL op-amps for layout purposes, all of which are obsolete. Since IC1 and IC12 are in the signal path, you'll want to use op-amps that are designed specifically for audio. The OPA2134 is a great choice, but a regular TL072 should do fine as well.

[2] **Compander:** The NE570 and NE571 companders (both ICs are cross-compatible) are out of production in DIP format. They can still be found NOS, the NE570 being much more common, but if you'd rather use something that is new and in production, the CoolAudio v571 clone is available from Small Bear Electronics.

[3] **Clocks and BBDs:** The original DC-2 uses the MN3207/MN3102 combination for the BBDs and clocks. These are out of production but can still be obtained relatively cheaply. However, you can also opt for one of two current-production BBD+clock pairs, the CoolAudio v3207/v3102 or the Belling BL3207/BL3102. Both of these are available at Small Bear Electronics.

[4] **Toggle switches:** The toggle switches must have at least a 0.35" (8.89mm) bushing. This is the most common bushing length, but a few from Mouser's "Mountain Switch" brand do have a shorter 0.28" (7.1mm) bushing. This shorter bushing will not allow enough space for the top PCB to clear the jacks.

[5] **DC jack:** This project uses a miniature unswitched DC jack like this one from Small Bear Electronics (available from several other suppliers as well). You can *probably* fit a standard-sized DC jack here—the cutout on the top PCB is wide enough—but the main issue will be the clearance above the bottom PCB. There's very little margin for error, such that even the height of the solder on the pads on the bottom PCB will be a factor.

[6] **Screws:** If you have a countersink drill bit, you can countersink the hole on the face of the enclosure and use a flat-head screw on the outside so it's level with the enclosure. This is purely cosmetic. If you choose to use the countersink method, you'll want a 6-32 x 1/4" flat head screw for this.

Pin headers

The two PCBs are connected by pin headers: three 3-position headers and one 5-position header. In order to have enough clearance between the PCBs, the female headers must be at least 0.325" (8.25mm) in height when mounted to the PCB. Tayda Electronics has them for about 5 cents each, and in my experience they connected more tightly than the ones from Mouser. Their 5-pin header is here and the 3-pin header is here.

Mouser carries a 5-pin header in the correct height, but not a 3-pin. So if you want to order from Mouser, you'd be better off getting one 16-pin header that snaps apart. Use a razor blade to separate these—if you try to snap them apart by bending them, the plastic housing will often crack. You'll need 14 of the 16 sockets.

You'll also need the male pin headers as well, the type with pins on both ends. Mouser has one here and Tayda has one here. Tayda is significantly cheaper and the quality is the same.

MN3007 and MN3101

The Blueshift does not support the MN3007/MN3101 chipset. These ICs require a minimum 9V supply voltage. Since the DC-2 circuit uses a regulated voltage from the main supply for the BBDs, and regulators typically require about 2V of headroom to function, 7V is as high as we can reasonably get from a 9V source. The original Dimension D rack unit ran at a much higher voltage and was able to use the MN3007/MN3101, but it's just not possible with the DC-2 circuit as it is. They also use a different pinout.

Mouser parts spreadsheet

Since this is such a complicated build, I created a spreadsheet of parts that can be imported directly to Mouser. Over 95% of the parts can be obtained from Mouser and their prices are great. I spent a great deal of time selecting the parts, and the PCB layout is designed around them, especially the boxed film & electrolytic capacitors, so by using this spreadsheet to order, you know you are getting the best possible result.

For international DIYers, Mouser recently upgraded their international shipping options, so it's very likely that you will be able to get free shipping with a project this size. Their prices are very competitive as well.

The BOM does assume you have zero parts on your bench, so you may be able to save a few dollars by comparing the spreadsheet with what you've got already and removing what you don't need, especially things like the IC sockets. But just know that while it may save a little bit of money to use your own, I guarantee it will save a lot of potential mistakes to receive them all in labeled bags!

With that said, I have to give the disclaimer that **this spreadsheet is being provided only as a convenience**. You are responsible for checking through the parts to make sure they are the ones you want, and Mouser is responsible for sending you the correct parts. I tried to pick high-availability components from well-known manufacturers that are stocked in large quantities, but with 70 different parts, it's likely that one or two of them will be out of stock at any given time. You will have to find your own replacements if that happens.

Calibrating

In order to get the highest amount of headroom out of the unit, you'll need to adjust the trim pots using a signal generator and an oscilloscope. If you don't have an oscilloscope, you can get pretty close just adjusting by ear.

Most BBDs do not handle signal symmetrically, so the point of highest headroom is not generally going to be at the halfway point of the supply voltage like it would be with op-amps. And BBD manufacturing processes were not very precise, so the circuit must be tuned to the exact BBDs being used.

Whether you do or don't use an oscilloscope, the first step is to **trim the BBD's bias voltage to half of the supply voltage**. This sets the starting point for the calibration. The regulated supply for the BBD section should be around 6.8V, so you'll want to adjust the trimmers until you read **3.4V** on pin 3 of each BBD. This can be done on just the bottom PCB without the top PCB being attached as long as the unit is powered. It can even be done before you solder the toggle switches in place.

TR1 adjusts **IC9** and **TR2** adjusts **IC3**.

Calibrating without an oscilloscope

While not as accurate as using an oscilloscope, setting the bias voltage by ear will lend good results that should be enough for most usage. It's pretty basic: you just want to adjust the two trimmers until you have the least amount of distortion in the audio signal. However, it's not very easy to hear, and you certainly won't be able to get it tuned with the same level of accuracy if you use an oscilloscope.

So, without an oscilloscope, your best bet is to just set the voltage to the average of different BBDs that have been tested, then adjust from there if you can hear audible distortion. I came up with a spreadsheet that will be periodically updated as people send me their voltages.

In my testing (which involved measuring an original DC-2 as well as testing the different brands of BBDs in the Blueshift), the optimum bias point was always between 3.3V and 3.8V.

Calibrating with an oscilloscope & signal generator

This assumes you know how to operate a signal generator and an oscilloscope and know how to interpret the readings. If you don't have one or don't know how to use one, please use the earlier "Calibrating without an oscilloscope" section. If you're planning on building the DC-2 for someone else in any kind of quantity, though, please do invest in an oscilloscope—you shouldn't be selling your work if it's not been professionally calibrated!

Here is how to set the bias using an oscilloscope.

1. If the unit is wired up inside the enclosure, make sure it's set to "On" using the footswitch. If the unit is not yet wired up, you'll need connect pad "N" to ground to turn on the optocoupler. (This is the same for either bypass method, but the jumpers on the optocoupler need to be set already.)

2. Set the signal generator to output an 8 kHz sine wave at 2V peak-to-peak. Feed this signal into the pad marked "A" (circuit input), taking care to connect the signal generator's ground wire to a common ground.
3. Connect the oscilloscope probe to the pad marked TP1, taking care to connect the ground wire of the probe to a common ground.
4. Adjust TR1 until the positive and negative halves of the waveforms are symmetrical. They will not be a pure sine wave like the input, but they should be rounded off in about the same shape as each other.
5. Repeat this process with TP2 and TR2.

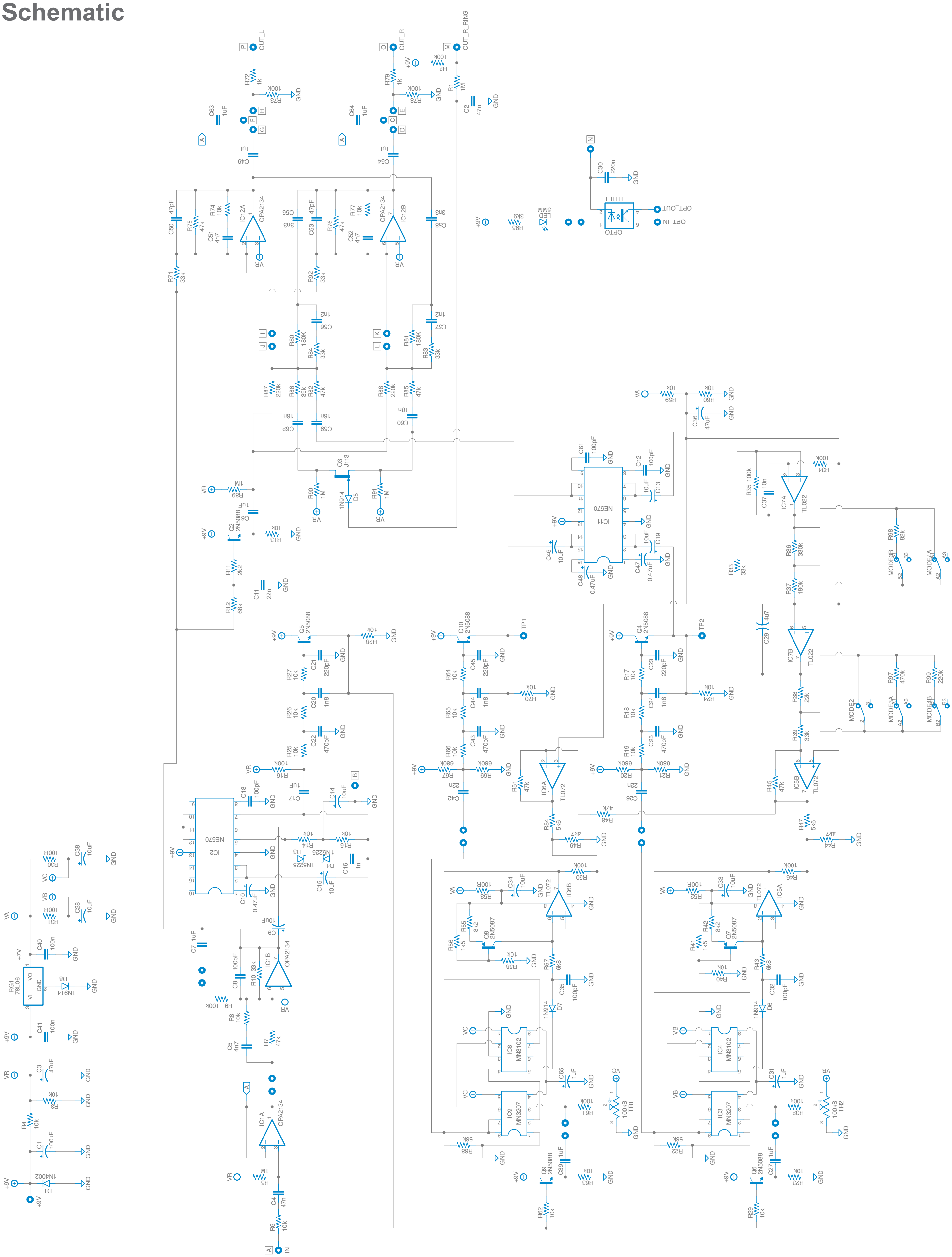
What type of oscilloscope should you use?

I use and recommend the LabNation SmartScope, a relatively inexpensive (USD $230) software-based oscilloscope that also includes signal generation functionality. The hardware connects via USB, and you can use a computer, smartphone or tablet to control it and view the display output. I haven't used it extensively, but it has gotten excellent reviews from others and definitely did what I needed it to for this project. There are a few tutorials and videos available online if you want to see more about it.

What should the waveform look like?

Here is a sample waveform taken from a test point on a real DC-2 unit. This is a screenshot from the LabNation SmartScope software. The voltage range is set to 1V and the timebase range is set to 50µs. Notice that the waveforms are symmetrical between top and bottom, but that they are not a perfect sine wave. They are ever-so-slightly lopsided toward the right side of the wave.

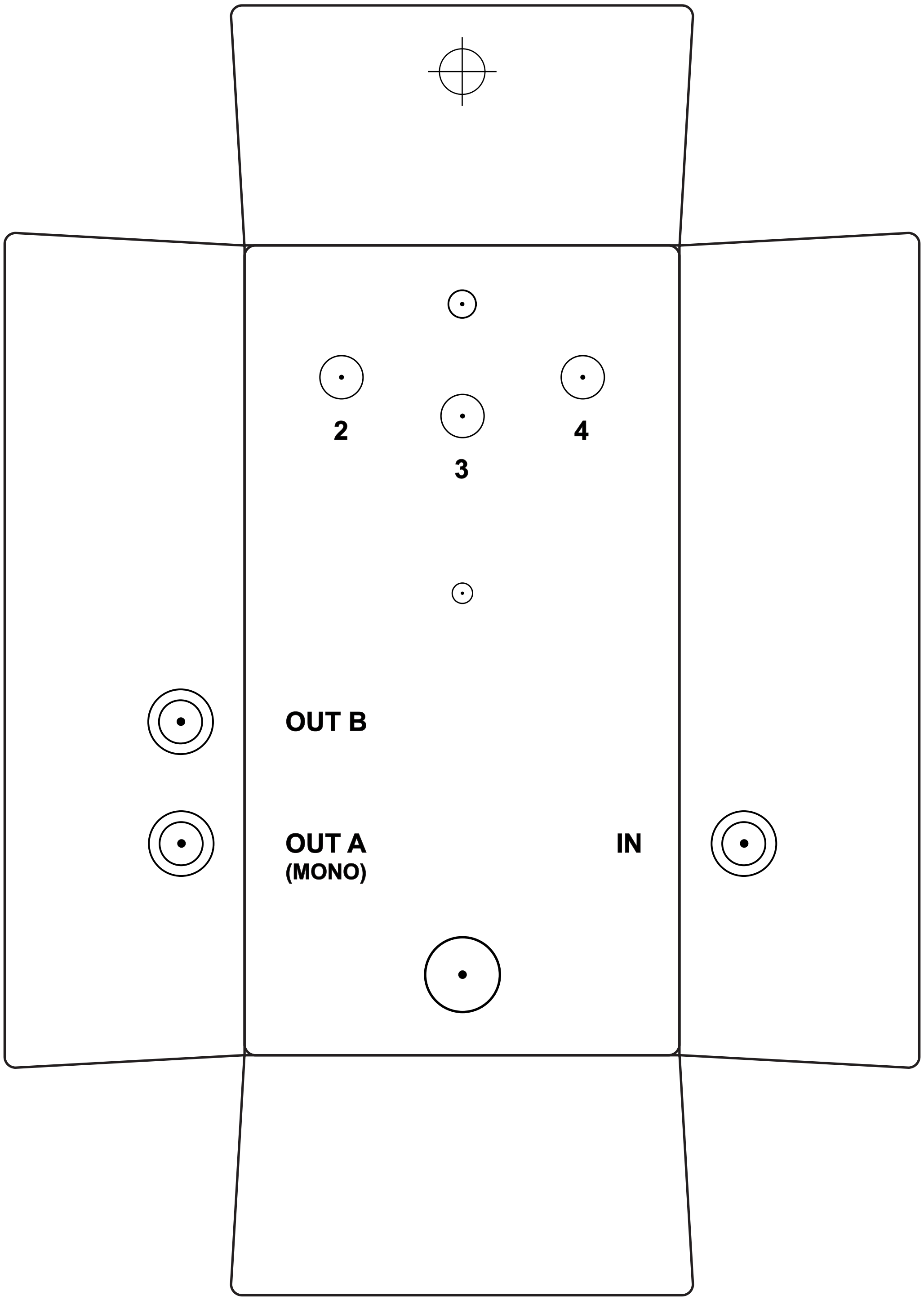
Schematic



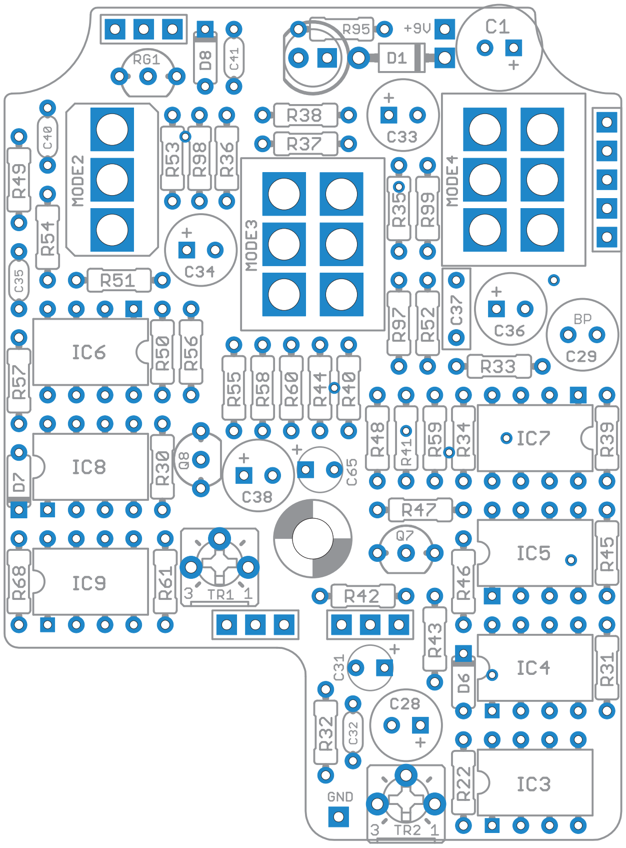
OUT_L
OUT_R
OUT_R_RING
IN
OPTO
OPT_IN
OPT_OUT
NE570
OPA2134
TL022
TL072
MN3102
MN3207
78L06
100kB TR1
100kB TR2

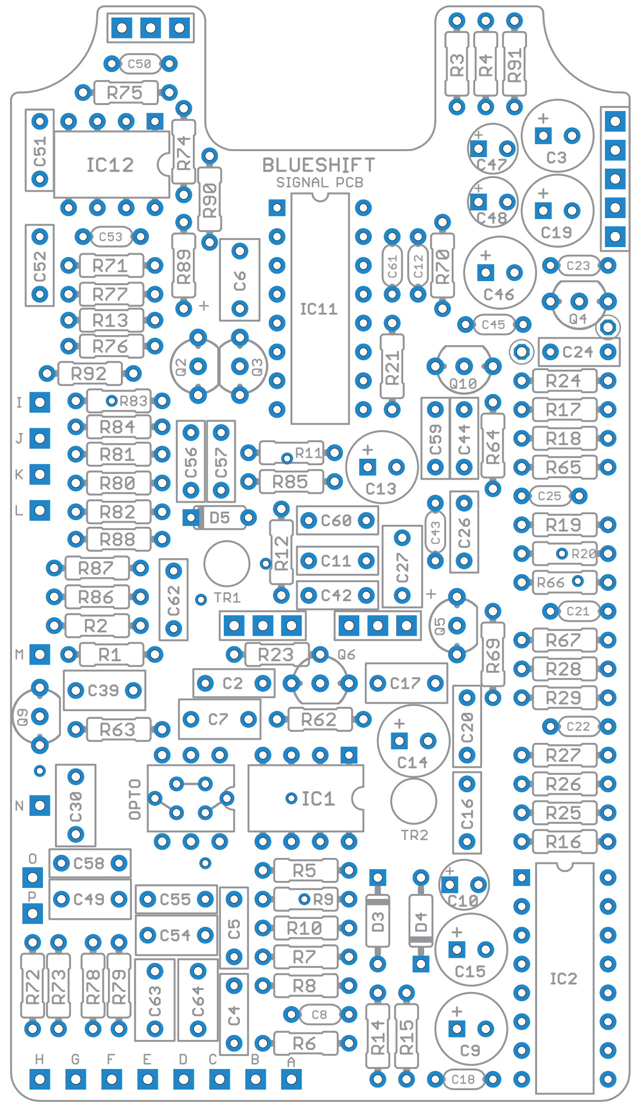
Drill Template

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. **The enclosure layout doesn't leave much room for error, so make sure you are very precise in your drilling!**



PCB Layout

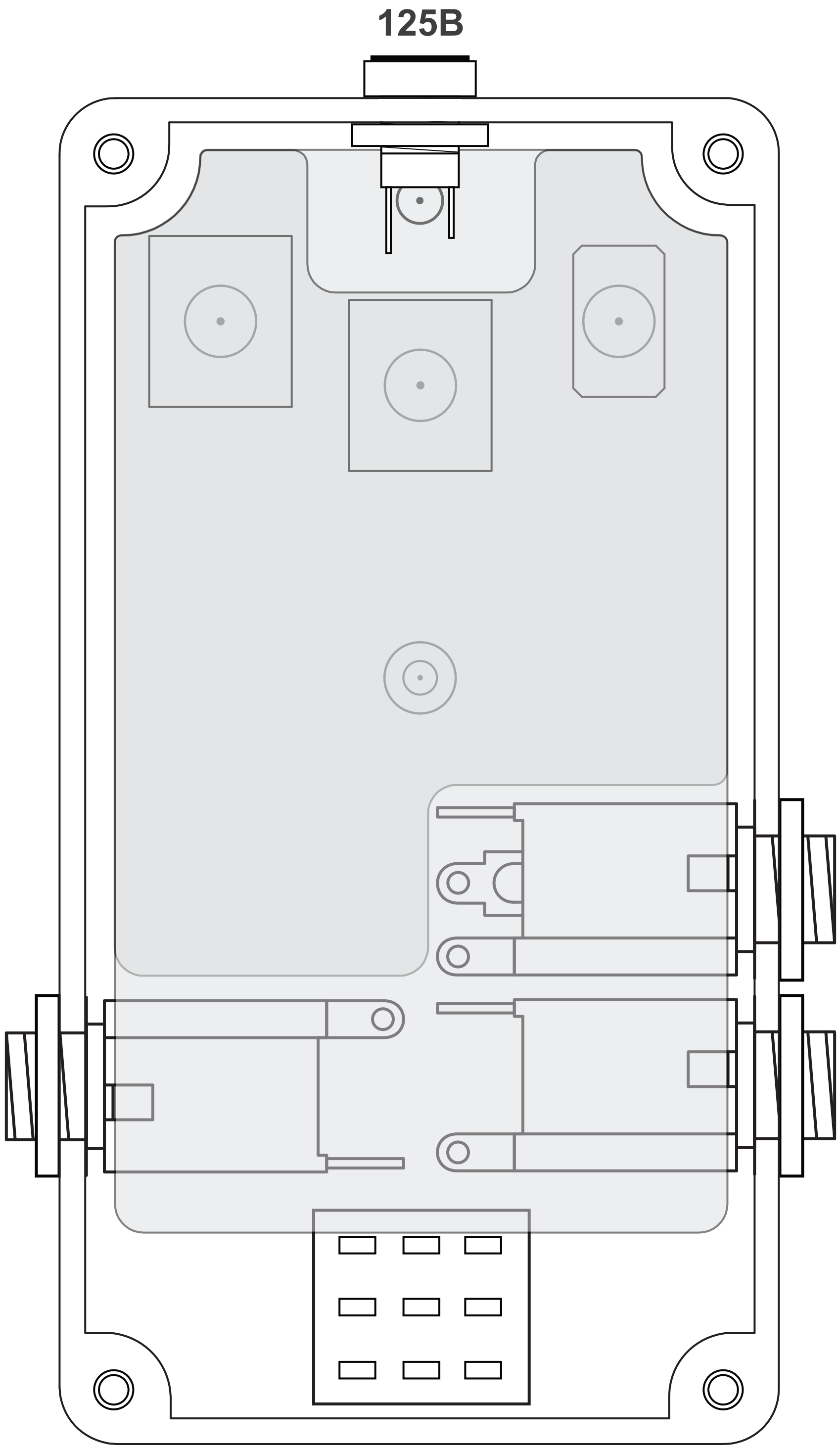




These are shown 50% bigger than life size. The PCB on the left (the clock + LFO board) measures 2.15” x 2.9”. The PCB on the right (the signal board) measures 2.15” x 3.79”. They are shown here with component sides facing up, but component sides will face down when installed in the enclosure.

Enclosure Parts Placement / Layout

This shows the placement of the jacks, switches, and other hardware, looking in from the bottom.



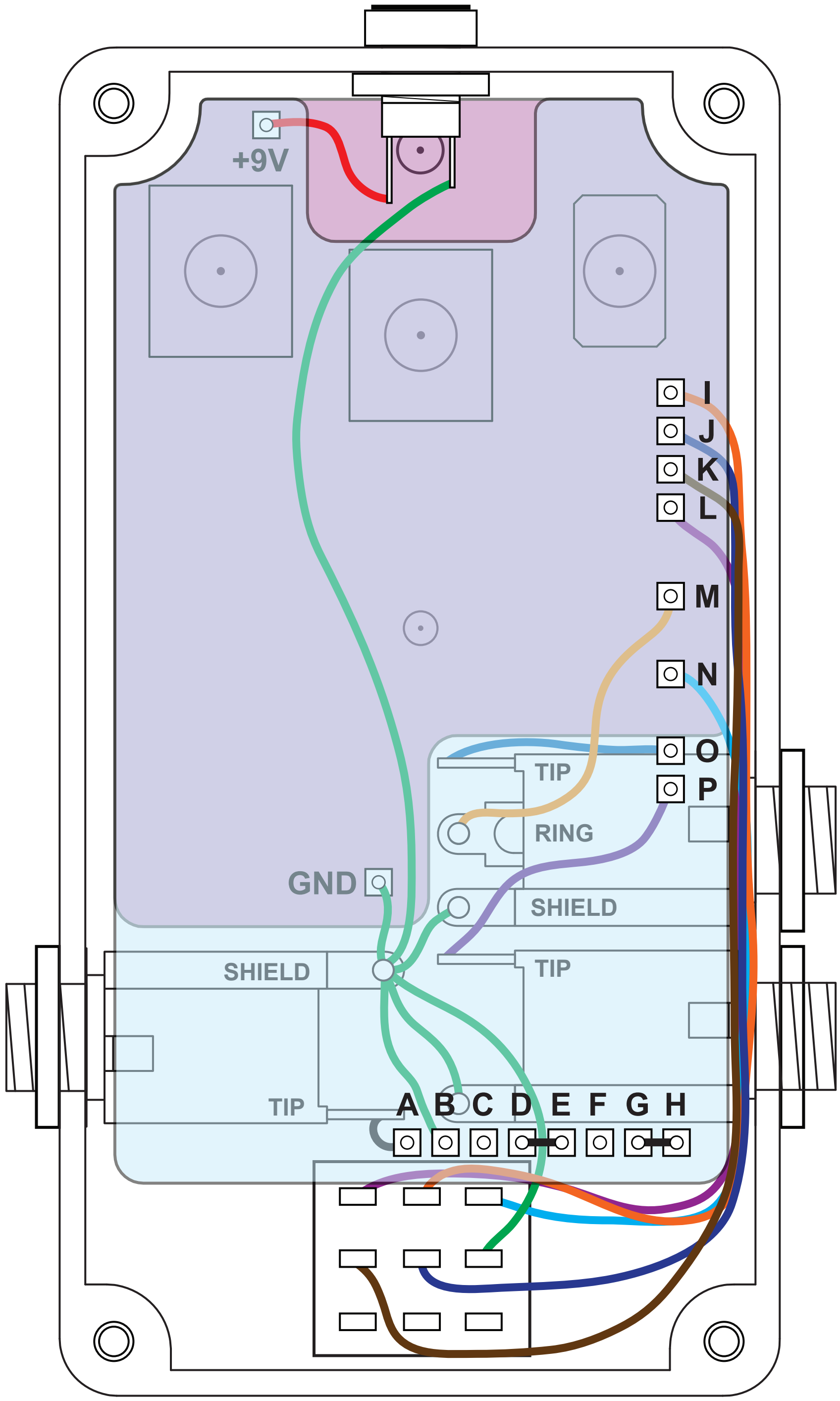
The **in/out jacks** are Switchcraft 111X for “In” and “Out A” and 112BX for “Out B”. Clones of these can be found pretty easily and will work fine—but they must be the black box-style enclosed jacks, not the open-frame type (Switchcraft 11 / 12B) or Neutrik NMJ series.

The **DC jack** is an unswitched miniature 2.1mm jack like this one from Small Bear Electronics.

Toggle switches are standard Taiway / Mountain Switch toggles (or equivalent) with at least 0.35” bushing.

No other hardware has been tested, so you’re on your own for measuring the fit if you use anything else!

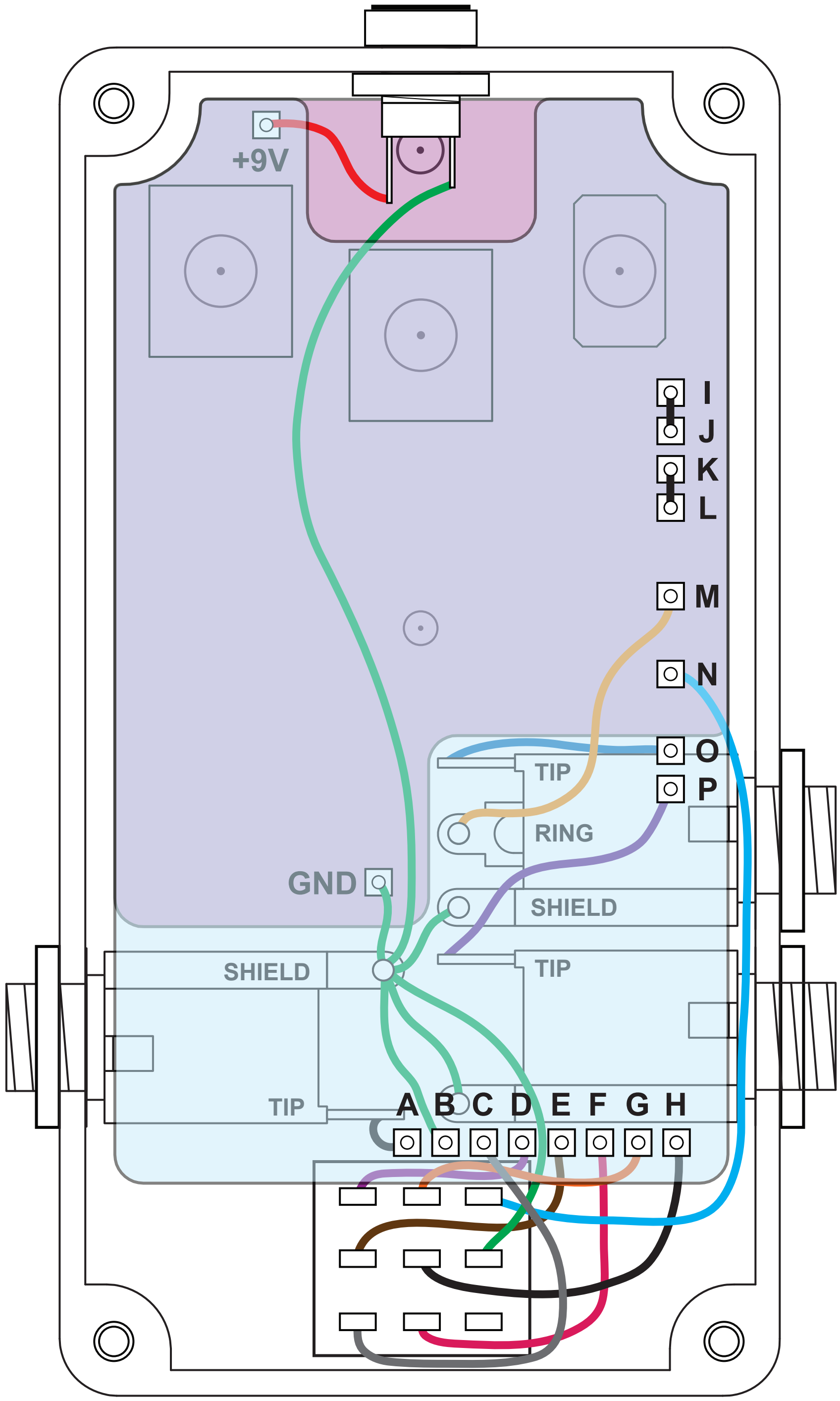
Wiring Diagram – Stock Bypass Version



In this wiring diagram, pads **C and F** are left unconnected. Pads **D & E** and **G & H** are jumpered.

Make sure to cross-reference this diagram with the table on page 15. The wires can be pretty hard to follow in the diagram since it is a small enclosure and there are a lot of layers.

Wiring Diagram – Minimalist Bypass Version



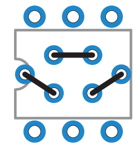
In this wiring diagram, pads **I & J** and **K & L** are jumpered.

Make sure to cross-reference this diagram with the table on page 15. The wires can be pretty hard to follow in the diagram since it is a small enclosure and there are a lot of layers.

Setting the bypass method

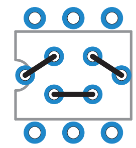
The bypass method must be set by using **jumpers on the optocoupler** and following the corresponding wiring diagram. The diagrams to the left show the jumper positions for each bypass mode. It's easiest to set these on the top (non-component) side of the board. The silkscreen on the PCB shows the jumpers in "Stock" mode.

STOCK



The **stock bypass** option mimics the signal path of the original circuit, with the signal passing through pre-emphasis and de-emphasis filters.

MINIMAL



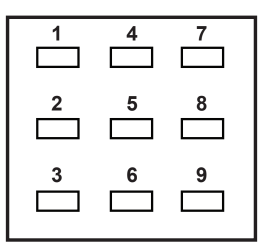
The **minimalist bypass** option splits the signal immediately after the first op-amp. The bypass is still buffered for the stereo output, but the signal only passes through that single op-amp buffer.

Since this is a stereo-splitting effect, I've intentionally not accounted for true bypass. Passive signal-splitting can be very problematic unless you know what's going on in your signal chain. With that disclaimer, you can probably wire it up as true bypass without too much trouble as long as you have a 4PDT stomp switch—but you're on your own if you do! I haven't tested this and it is beyond the scope of this documentation.

Since there are a lot of wires in multiple layers inside a small box, the wiring diagrams on pages 13 and 14 can be a little hard to follow. Use the table below and cross-reference it with the corresponding wiring diagram.

Pad	Description	Stock Bypass	Minimalist Bypass
A	Circuit input	Input jack tip	Input jack tip
B	Ground	Star ground point	Star ground point
C	Channel B buffer send	*No connection*	Switch lug 3
D	Channel B effect send	Jumper to pad E	Switch lug 1
E	Channel B switched output return	Jumper to pad D	Switch lug 2
F	Channel A buffer send	*No connection*	Switch lug 6
G	Channel A effect send	Jumper to pad H	Switch lug 4
H	Channel A switched output return	Jumper to pad G	Switch lug 5
I	Channel A pre-IC switch return	Switch lug 5	Jumper to pad J
J	Channel A pre-IC switch send	Switch lug 4	Jumper to pad I
K	Channel B pre-IC switch return	Switch lug 2	Jumper to pad L
L	Channel B pre-IC switch send	Switch lug 1	Jumper to pad K
M	Channel B ring	"B" output jack ring	B output jack ring
N	Optocoupler LED switched ground	Switch lug 7	Switch lug 7
O	Channel B output	"B" output jack tip	"B" output jack tip
P	Channel A output	"A" output jack tip	"A" output jack tip
+9V	Supply voltage (bottom board)	DC jack	DC jack
GND	Ground (bottom board)	Star ground point	Star ground point

The stomp switch lugs use the standard numbering system, shown in this diagram:



General Build Instructions

Build Order

When putting together the PCB, it's recommended that you do not yet solder any of the enclosure-mounted control components (switches) to the board. Instead, follow this build order:

1. Attach the **audio jacks**, **DC jack** and **footswitch** to the enclosure.
2. Remove all nuts and washers from the **toggle switches**. For each of the switches, thread three nuts firmly onto the shaft as far down as they'll go. The additional nuts serve as spacers so that the PCB is mounted at the correct height.
3. Firmly attach each of the **toggle switches** to the enclosure, taking care that they are aligned and straight.
4. 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.
5. Attach the **hex standoff** to the enclosure using a screw from the outside. If you have a countersink drill bit, you can countersink the hole from the outside and use the flat-head screw so that it's level with the enclosure. Otherwise, use a pan-head screw here.
6. Ensure that the **bottom (clock) PCB** has the female header sockets soldered in place and that they are at a straight 90° from the PCB. Fit this PCB onto the toggle switches as well as the hex standoff and the leads of the LED. Use a 6-32 x 3/8" panhead screw to attach the PCB to the standoff. The PCB should rest just slightly above the flat base of the toggle switches, but below the top of the lugs. If it doesn't fit, or if you need to bend things more than you think you should, double-check the alignment of the switches.
7. Once you feel good about everything, **solder the toggle switches from the top**.[2] By soldering them in place inside the enclosure, 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.
8. Wire everything on the bottom PCB according to the wiring diagram.
9. On the **top (signal) PCB**, it's easiest if you do not yet solder the male header pins. Instead, attach the header pins to the female sockets of the lower PCB. Then, put the top PCB in place, fitting the header pins inside the proper holes, and solder them in place. Once again, this helps "custom fit" the pins together so that there is no stress on the solder joints from slight misalignments.
10. Wire everything on the top PCB 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 from the top.

[2] **Note on soldering the toggle switches:** 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.

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 ICs. It may save you a lot of headaches later on. This PCB is particularly difficult since the pad, hole and trace sizes are smaller than most other DIY projects.

Checklist

I have tried to be as thorough as possible in this documentation, but the upshot is that by providing so much information, some of the most important details may be missed. Here are the high-level things to make sure of before turning it on for the first time:

- I have chosen a bypass method (page 15) and set the jumpers on the optocoupler accordingly.
- I double-checked the orientation of all of the ICs in their sockets.
- I'm using a stereo jack for Output B.
- I'm using female header sockets that are at least 0.325" (8.25mm) in height from the PCB.
- I triple-checked the wiring against both the diagram and the table on page 15.
- I double-checked the orientation of the LED before soldering it in place.

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!)