PROJECT NAME

AMETHYST

BASED ON

BOSS® DM-2 Delay

BUILD DIFFICULTY

Advanced

EFFECT TYPE

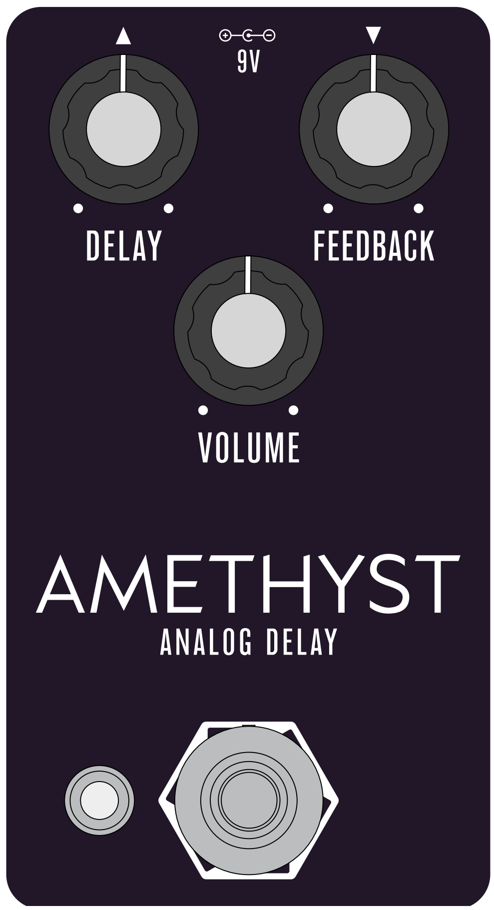
Analog delay

DOCUMENT VERSION

1.0.2 (2025-01-05)

PROJECT SUMMARY

One of the earliest analog delay effects to use a bucket brigade chip, and still widely considered among the best analog delays ever designed.



IMPORTANT NOTE

This documentation is for the **kit** version of the project. If you purchased the PCB by itself, please use the PCB-only version of the documentation instead. The circuit is the same, but the instructions are completely different due to the specialized parts and assembly methods used in the kit.

TABLE OF CONTENTS

INTRODUCTION

If this is your first pedal, welcome to the hobby and thank you for choosing Aion FX. You've just joined a community of over 100,000 people around the world with a passion for building homemade noise machines using obsolete electronics technology, and we're glad to have you!

If you've done this before, it's great to see you again and we're confident you'll find this build experience an enjoyable one.

Aion FX kits are designed to empower anyone to build a high-quality pedal, no matter the skill level. The pedalbuilding hobby has traditionally had a steep learning curve, but don't be overwhelmed—we've done all the hard work for you. All you need to do is follow these instructions and you'll be on your way to transforming your tone.

There are a few things to go over before you get started.

- **You're going to have to get your hands dirty**—there's no way around it. Nothing here comes preassembled, and you'll have to learn the skills to put it all together. This document will walk you through everything you need, but be prepared to learn a few things along the way.
- **This will take time**. Plan on about two hours start to finish. It may take even longer if it's your first time building. Don't rush it. If you find yourself getting frustrated or overwhelmed, take a break and come back in a couple of hours or the next day.
- **No direct technical support is offered.** There are several DIY forums and Facebook groups with thousands of members who enjoy troubleshooting and teaching. But please be sensitive to the fact that the staff at Aion FX is minimal, and every minute spent helping individuals in private is time that can't be spent on new project development.
- **There is no implied guarantee of a final product.** Aion FX provides the ingredients and the recipe, but you are responsible for putting everything together to make it work. We've tried to make the process as clear and accessible as possible, but it must be expressly stated that purchasing the kit is not a guarantee that you will end up with a working pedal.

It's recommended to read through all of the instructions before you start, particularly if you've never built a pedal before. If you familiarize yourself with the entire process ahead of time and you know what the goal looks like, each step will make more sense.

Now, on to the fun stuff!

PACKING LIST

This is a list of all the parts that are included with the kit, grouped by value. For a list of all the parts based on their PCB part numbers, please see pages 29-30.

If you find that any parts are missing or damaged, please fill out the Missing Parts form.

Film Capacitors

NAME	QTY
1n	1
2n2	1
6n8	3
33n (0.033)	1
39n (0.039)	1
47n (0.047)	1
82n (0.082)	1
100n (0.1 or “µ1J100”)	1
220n (0.22)	2
1uF	3

Electrolytic Capacitors

NAME	QTY
1uF	3
10uF	5
47uF	1
220uF	2

MLCC Capacitors

NAME	QTY
100pF (marked “101”)	5
330pF (marked “331”)	2
100n (marked “104”)	2

Transistors

NAME	QTY
2N5088	4

Resistors

NAME	QTY
22R	1
470R	1
10k	21
18k	1
22k	3
47k	5
100k	4
470k	1
1M	1

Diodes

NAME	QTY
1N5817	1
1N914	1
1N4739A	1

Trimmers

NAME	QTY
10k (marked “103”)	1
22k (marked “223”)	1
1M (marked “105”)	1

PACKING LIST (CONT.)

ICs

NAME	QTY
JRC4558D	1
v3205SD	1
v3102D	1
v571D	1
8-pin socket	4
14-pin socket	1

Other

NAME	QTY
LED bezel	1
LED, white	1
DC jack	1
Input/output jack	2
Mounting nut, jack, 0.54"	4
Outer washer, jack, 0.6"	2
Lock washer, jack, 0.5" (thin)	2
Enclosure	1
Enclosure screws	4
PCB, main circuit	1
PCB, footswitch	1
PCB, input/output/DC	1

Potentiometers

NAME	QTY
50kA *(see note)*	1
50kB	1
1MB	1
Dust cover	3
Knob	3
Mounting nut, potentiometer, 0.44"	3
Lock washer, potentiometer, 0.5"	3
Outer washer, potentiometer, 0.475"	3

Note: The Feedback pot was changed from 50kB to 50kA in January 2025. Kits purchased before this time will have two 50kB pots.

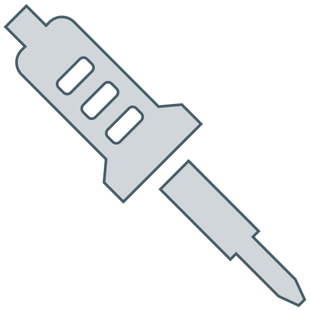
Switches

NAME	QTY
Stomp switch, 3PDT	1
Mounting nut, stomp switch, 0.6"	2
Lock washer, stomp switch, 0.6"	1
Dress nut, stomp switch, 0.77"	1

Wiring

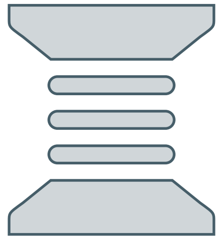
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 122mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



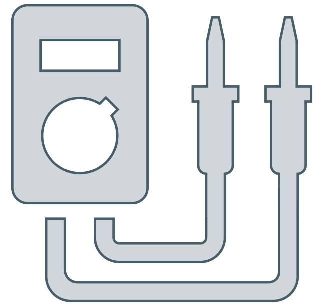
SOLDERING IRON

Temperature-adjustable is recommended. The optimum soldering temperature is 700-725° F (371-385° C) for leaded solder, or 750° F (400° C) for lead-free.



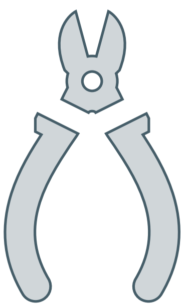
SOLDER

Preferably 63/37 or 60/40 leaded solder. Lead-free is more difficult to use, so if that's the only type you can get, it's best to watch tutorials that are specific to lead-free solder.



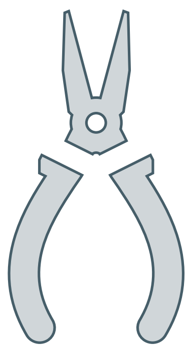
DIGITAL MULTIMETER (DMM)

Most cheap ones in the $10-30 range are fine for what we're doing. Make sure it has audible continuity testing (i.e. it beeps at the lowest resistance) and transistor hFE measurement.



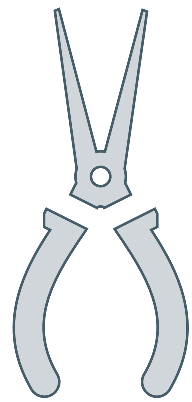
WIRE SNIPPERS

Also called nippers or wire cutters. The Hakko CHP-170 is the best you can get for less than $10.



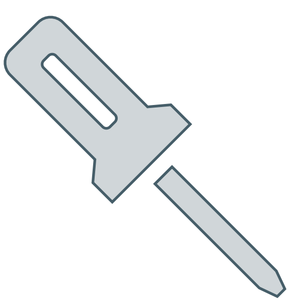
FLAT-NOSE PLIERS

Many general-purpose uses, but particularly tightening the nuts of pots, switches and jacks. Quicker than changing out sockets on a ratchet.



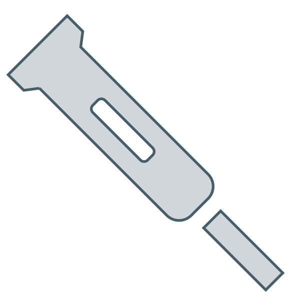
NEEDLE-NOSE PLIERS

These are used for bending leads on components and other general uses. Use the smaller type with a tip that's approximately 0.05" (1.25mm) wide.



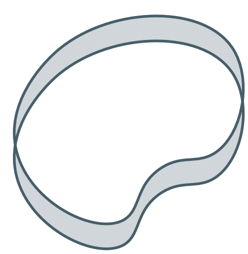
SCREWDRIVER (PHILLIPS)

Used for the enclosure screws. Get a powered driver if you'll be building a lot of pedals!



FLAT SCREWDRIVER (SMALL)

This is used for tightening the set screws on the knobs. The tip should be no more than 0.1" (2.5mm) wide.



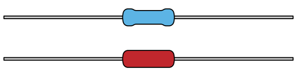
RUBBER BAND

Yes, a plain old rubber band. This is used to tighten the dress nut to avoid scratching or denting it (which can happen with metal tools).

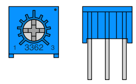
COMPONENT IDENTIFICATION

If you've never built a pedal before, you'll need to know what all the components are. These are shown actual size. (Not all of these types of components may be part of this kit.)

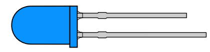
RESISTOR



TRIM POTENTIOMETER

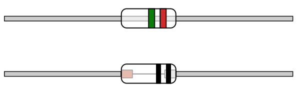


LED



SILICON DIODE

GERMANIUM DIODE



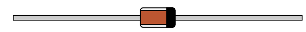
These are very fragile, so be careful when handling them.

RECTIFIER DIODE

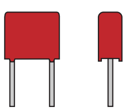
Some Schottky diodes also look like this.

SCHOTTKY DIODE

ZENER DIODE

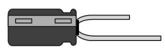


FILM CAPACITOR



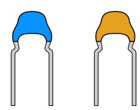
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



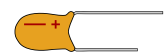
Polarized. The negative side is marked.

MLCC



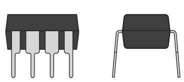
Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR



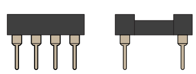
Polarized. The positive side is marked.

IC OR OP-AMP

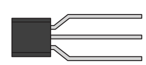


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET

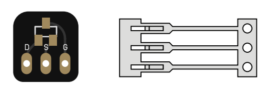


TRANSISTOR OR JFET



Some voltage regulators also look like this.

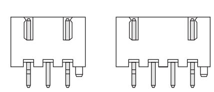
TRANSISTOR WITH ADAPTER



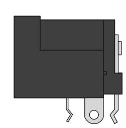
The pins will be soldered to the adapter during kit assembly.

WIRE ASSEMBLY

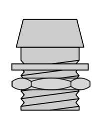
WIRE ASSEMBLY HEADER



DC JACK



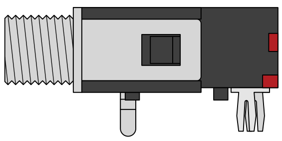
LED BEZEL



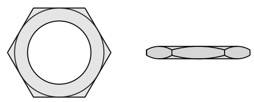
HARDWARE IDENTIFICATION

The hardware comes unassembled, so you'll need to sort & identify each of the pieces. The diagrams below are actual size, so you can set them against the printed page to identify them if needed.

I/O JACK

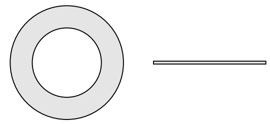


MOUNTING NUT



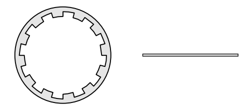
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



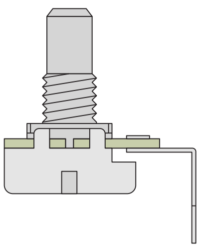
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

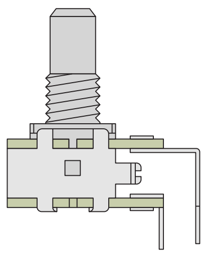


DIAMETER: 0.5" / 12.7mm

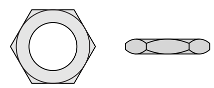
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

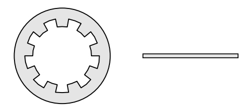


MOUNTING NUT



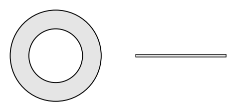
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



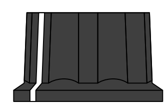
DIAMETER: 0.5" / 12.7mm

OUTER WASHER

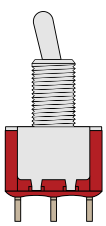


DIAMETER: 0.475" / 12mm

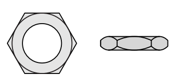
KNOB



TOGGLE SWITCH

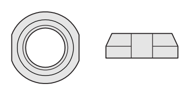


MOUNTING NUT



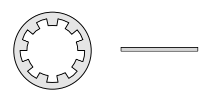
DIAMETER: 0.36" / 9.1mm

DRESS NUT



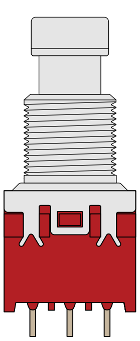
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

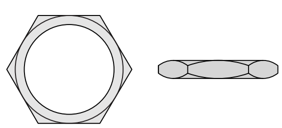


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

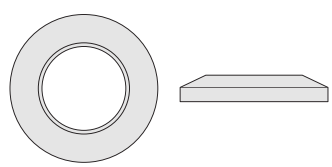


MOUNTING NUT



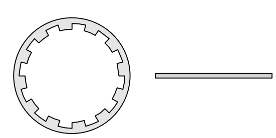
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER

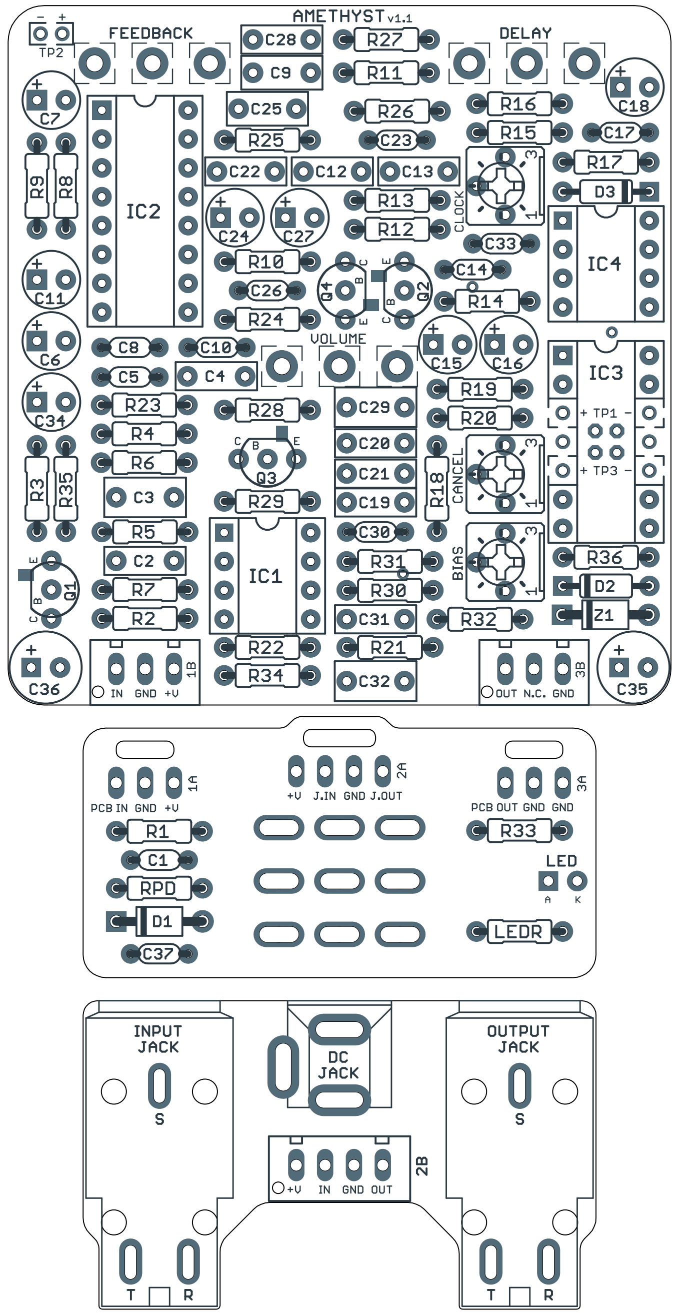


DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

Now it's time to start building!

The first thing you need to do is snap apart the PCBs into 3 separate boards (if needed) and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the PCBs shown to the right.

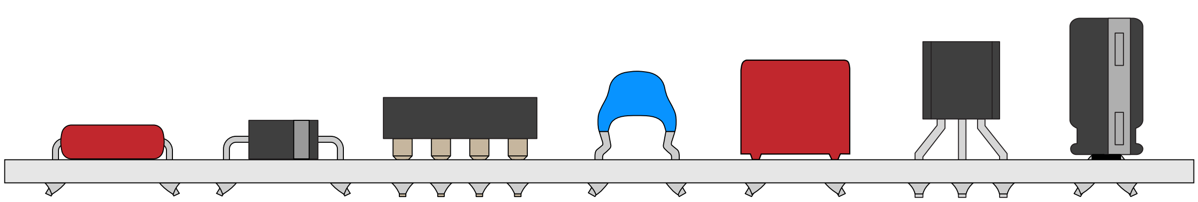


The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest so that when the PCB is upside-down, everything is making contact with the work surface and is held in place.

Generally speaking, you should populate the components in this order:

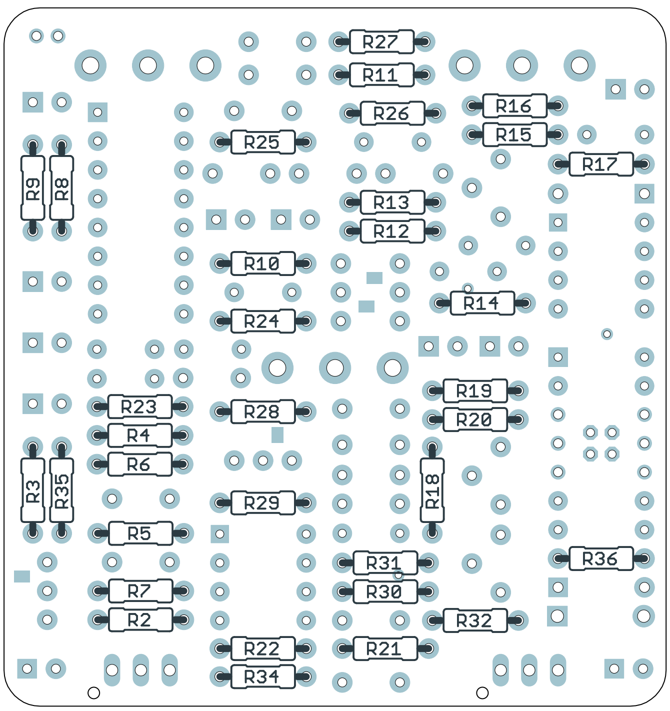
1. Resistors
2. Diodes
3. IC sockets
4. MLCC capacitors
5. Film capacitors
6. Transistors
7. Electrolytic capacitors

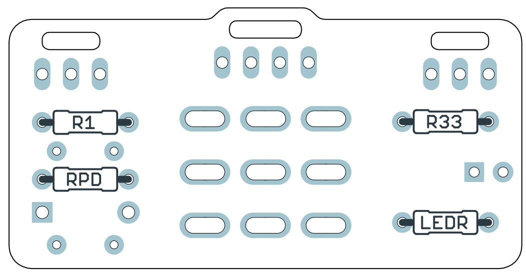
Not all of these component types are included in each kit, so skip them if they aren't applicable. Some types of film capacitors are taller than electrolytics, so those can be done last.



RESISTORS

PART	VALUE	PART	VALUE	PART	VALUE	PART	VALUE
R1	10k	R11	10k	R21	10k	R31	47k
R2	470k	R12	10k	R22	10k	R32	470R
R3	10k	R13	10k	R23	10k	R33	100k
R4	10k	R14	100k	R24	10k	R34	10k
R5	47k	R15	10k	R25	10k	R35	10k
R6	47k	R16	18k	R26	10k	R36	22R
R7	10k	R17	22k	R27	22k	RPD	1M
R8	10k	R18	100k	R28	47k	LEDR	22k
R9	10k	R19	100k	R29	47k		
R10	10k	R20	10k	R30	10k		

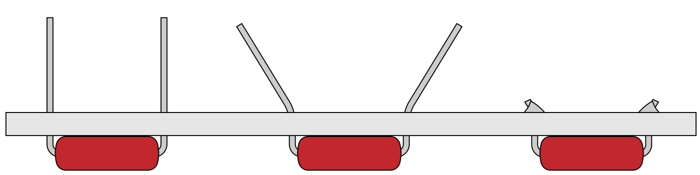




Be careful when installing the 10k resistors. There are 21 of them, and R16 is 18k which is easily mistaken for 10k. It's recommended to find the one 18k resistor first and set it aside before soldering the 10ks.

Likewise, R36 is 22R and there are three 22k resistors (R17, R27 and LEDR). The "R" is easily mistaken for a "K". Make sure that you correctly identify the one 22R before soldering any of them, and use a multimeter if you are uncertain.

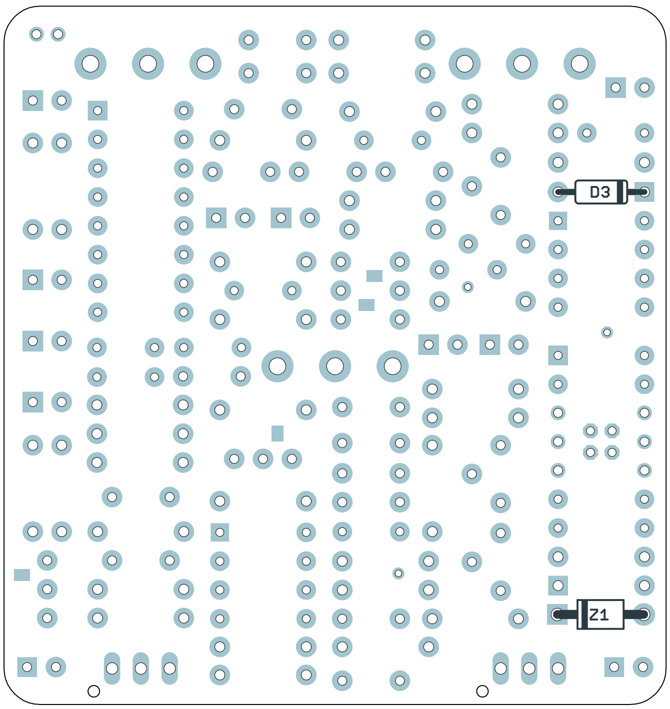
Using the parts list above, populate the resistors by pushing them through the holes and bending the leads outward at an angle to hold them in place. Resistors are not polarized, so they will work in any direction. Turn the board upside-down to keep the components held in place while you solder.

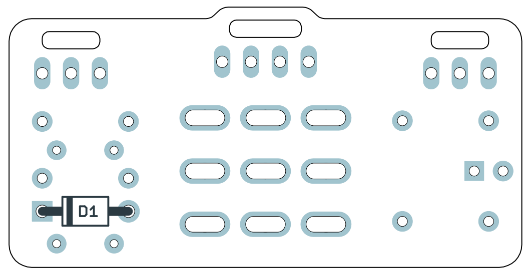


Don't try to do all of the resistors at once. You'll want to stop periodically flip the board and solder everything, then cut the leads using the wire snippers to make room for more.

DIODES

PART	VALUE
D1	1N5817
D3	1N914
Z1	1N4739A



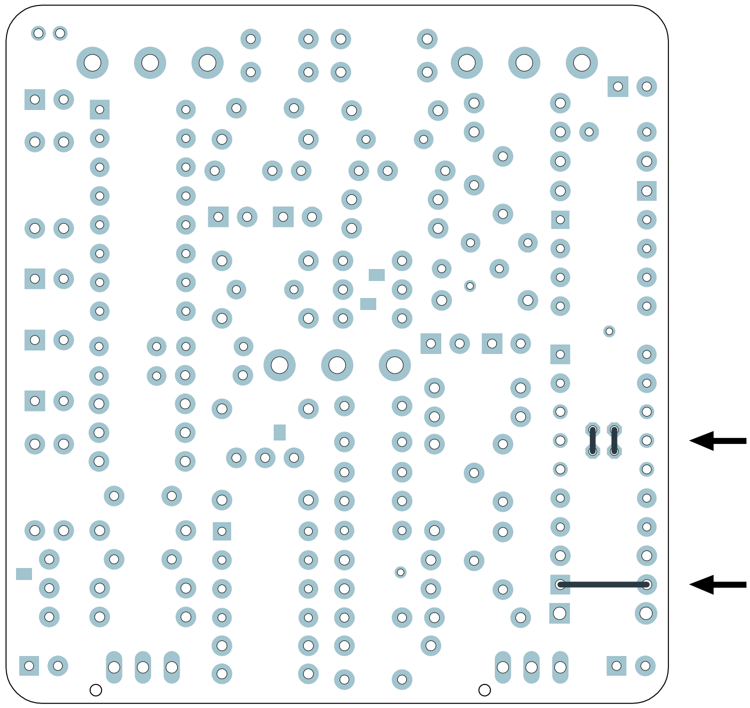


Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

The value can be difficult to read, so they can alternately be identified by appearance. D1 (1N5817) is black. D3 (1N914) is the two smaller orange and black one. The larger orange and black one is Z1, the 1N4739A. D2 is not used in the kit.

When trimming the leads after soldering, set aside the two leftover leads from the 1N914 diode. We will use these in the next step.

JUMPERS



Before we get to the ICs, we need to solder three small jumper wires.

The first is in place of D2. In this circuit, only D2 or D3 is used depending on the type of BBD, and the other is jumpered, meaning that a wire is used in place of a component to short the two pads together. Bend a leftover resistor or diode lead into approximately the width of a resistor (0.4”) and install it as a normal component.

The other two jumpers are located underneath IC3. These jumpers configure the PCB for the type of BBDs that are used in the kit. Each of the two top pads needs to be connected to the corresponding bottom pad as shown in the diagram above.

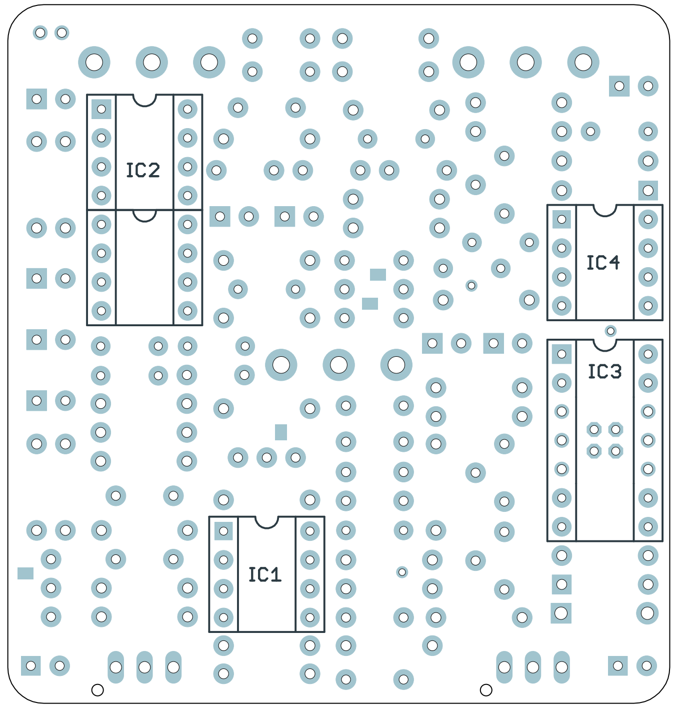
It’s easiest to use the leftover leads from the 1N914 diode in the previous step. The leads from these diodes are thinner than those of the resistors, so they are easier to form into the right shape. Bend them into a tight “U” shape and then solder them to the PCB like normal components.

Note that the jumper pads are much smaller than normal pads, so use caution when soldering. Since these pads carry power and ground, a short between the two adjacent jumpers would likely cause permanent damage to the circuit.

If you want to be extra careful, you can use your multimeter in continuity test mode to ensure that there is no connection between the two vertical pairs of pads before moving onto the next step.

SOCKETS & ICS

PART	VALUE
IC1	JRC4558D
IC2	v571D
IC3	v3205SD
IC4	v3102D



Next up are the IC sockets. You can't bend the leads of the sockets as with the other components, so they won't stay in on their own until they are soldered. Flip the PCB over and use gravity to hold them in place.

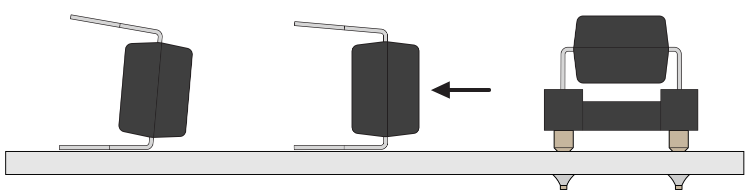
Note that IC2 uses two 8-pin sockets instead of one 16-pin socket.

IC3 is a non-standard configuration, with only 8 pins in a 14-pin-sized package. The middle three rows of the socket will be left empty once the IC is installed, but all 14 pins should be soldered as normal. Two of the rows will be used later as test points for calibration.

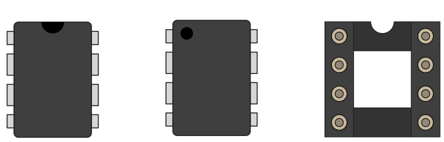
Installing the ICs

Don't insert the ICs into the sockets just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

The legs of the ICs are bent outward slightly during manufacturing, so they'll need to be bent back inward before they can be inserted into the sockets. It's easiest to do this by laying the IC legs against the table and bending the body itself so all four legs on the side are straightened out at once. Then, flip it and do the other side.

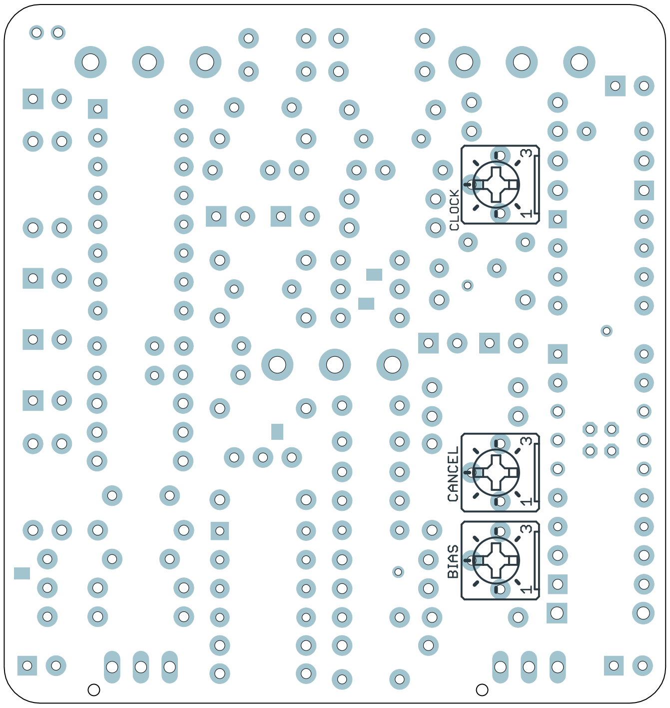


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRIMMERS

PART	VALUE
CANCEL	10k trimmer (marked "103")
BIAS	22k trimmer (marked "223")
CLOCK	1M trimmer (marked "105")



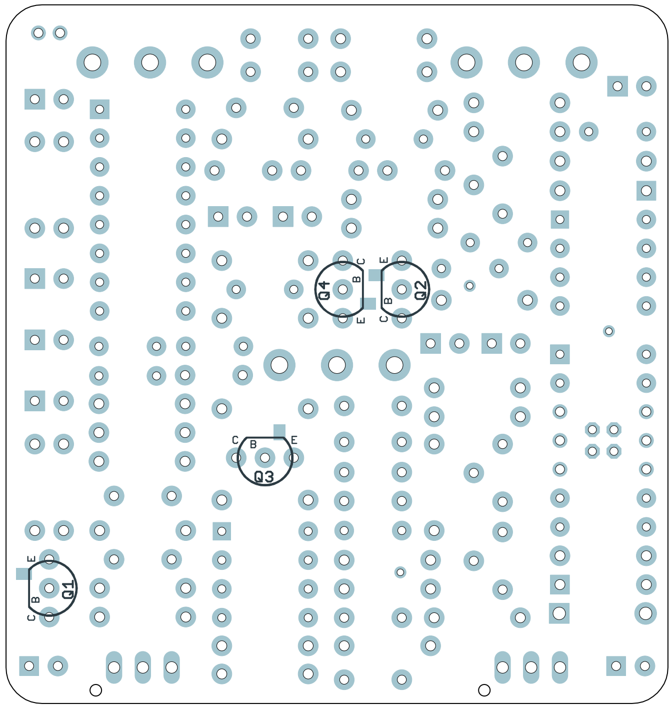
Next, we'll do the trimmers. The value code is located on the side. The 10k trimmer will read "103", the 22k trimmer will read "223", and the 1M trimmer will read "105". This code is etched or engraved rather than printed, so it's easiest to read with direct light reflecting off the side.

Setting the trimmers

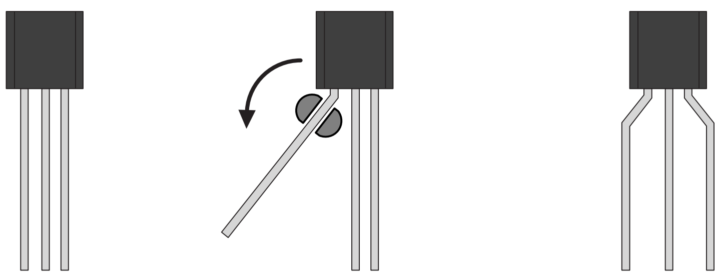
These trimmers will be used for the calibration process on pages 25 and 26 and should not be adjusted after calibration is completed. For now, set all three of them them to the 50% position (12:00 noon). Note that since they are rotated 90 degrees on the PCB, the 12:00 position means that the indicator points directly to the left.

TRANSISTORS

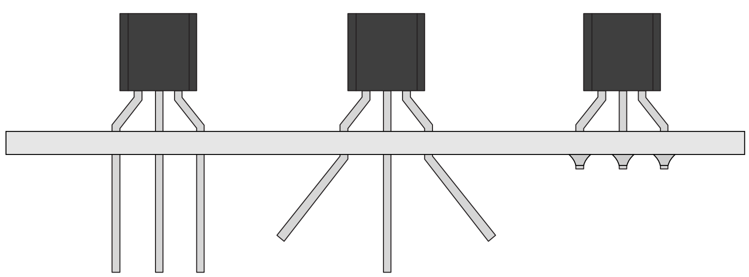
PART	VALUE
Q1	2N5088
Q2	2N5088
Q3	2N5088
Q4	2N5088



Now we'll do the four transistors. For each, if the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



Once installed, bend the two outer legs to hold it in place on the board. Then, solder them in place and clip the leads.



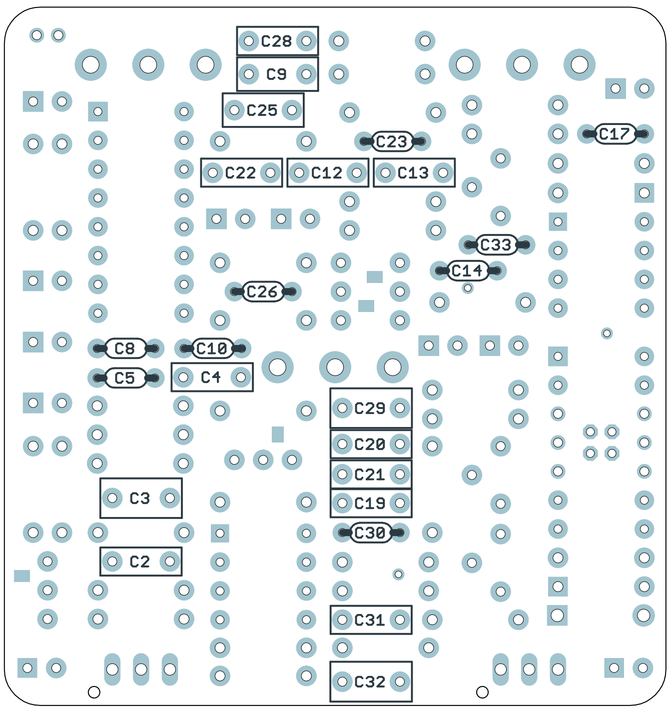
CAPACITORS (NON-POLARIZED)

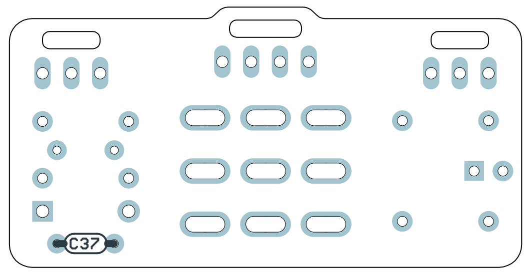
PART	VALUE
C2	47n (0.047)
C3	1uF
C4	6n8
C5	100pF MLCC
C8	100n MLCC
C9	220n (0.22)

PART	VALUE
C10	100pF MLCC
C12	6n8
C13	82n (0.082)
C14	330pF MLCC
C17	100pF MLCC
C19	2n2

PART	VALUE
C20	33n (0.033)
C21	1n
C22	39n (0.039)
C23	330pF MLCC
C25	220n (0.22)
C26	100pF MLCC

PART	VALUE
C28	100n (0.1)
C29	1uF
C30	100pF MLCC
C31	6n8
C32	1uF
C37	100n MLCC





C1 and C33 are not used in the kit and will be left empty on the PCB.

After the sockets come the box film and MLCC capacitors. These are all several different heights, so it's recommended to do them shortest to tallest. Bend the leads at an angle to hold them in place.

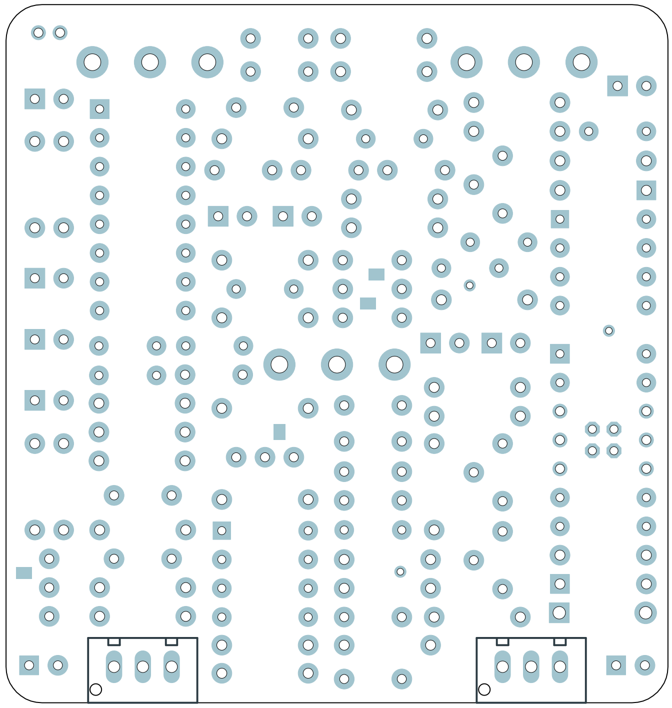
MLCCs and box capacitors are not polarized, so they will work in any direction, but to keep things neat, it's best to put them all facing the same way.

Note: Depending on the type, the box film capacitors may have their value printed on either the top or the side. Usually the red ones have it printed on the side while the blue or gray ones have it on the top.

C28 is usually red, but may read "µ1J100" on the top rather than the side.

All of the MLCCs except C8 and C37 are blue MLCC capacitors taped to cardboard. For these, the value will be written on the cardboard. C8 and C37 (100n MLCCs) are the only yellow ones and are most easily identified by color.

WIRE HEADERS

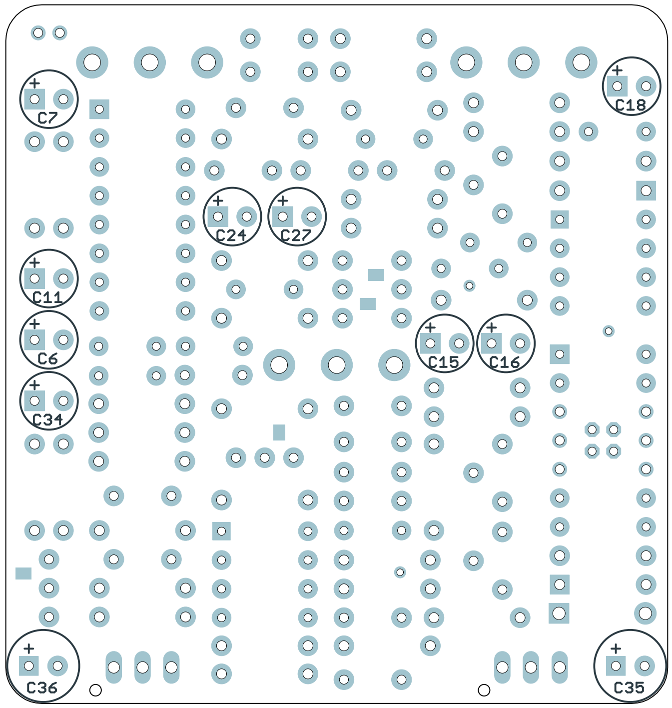


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

There's also a 4-pin header on the I/O board that we will do in a later step.

CAPACITORS (POLARIZED)

PART	VALUE
C6	10uF
C7	10uF
C11	10uF
C15	1uF
C16	10uF
C18	1uF
C24	10uF
C27	1uF
C34	47uF
C35	220uF
C36	220uF

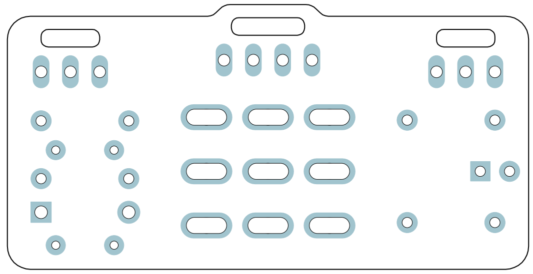


Populate the electrolytic capacitors. These are the tallest components so we save them for last. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

These are the last of the on-board components. Now is the time to go back to page 13 and insert the ICs into the sockets.

FOOTSWITCH PCB

PARTS
3-strand wire assembly (2)
4-strand wire assembly



Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

There will be one longer assembly with 4 wires and two shorter ones with 3 wires. The longer one goes in the middle and the shorter ones go on the left and right sides. The wire assemblies should then be soldered to the footswitch board as shown.

STEP 1

First, thread the wire through the strain-relief slots, with the blue side facing outward and the PCB's previously-installed components facing up.

For now, pull it through as far as it can go.

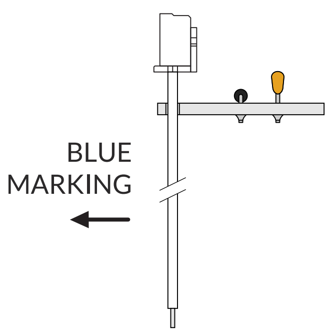
STEP 2

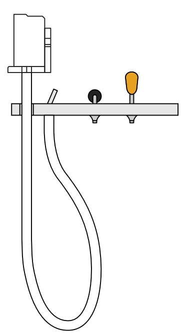
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

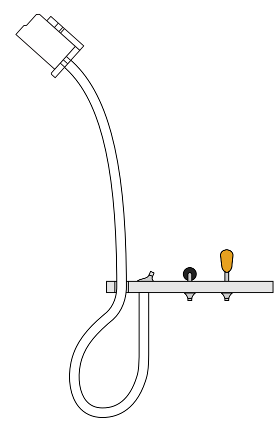
On the top side of the PCB, bend the exposed wires backward so it holds the wire in place. Pull the header back up through the slot partway.

STEP 3

Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



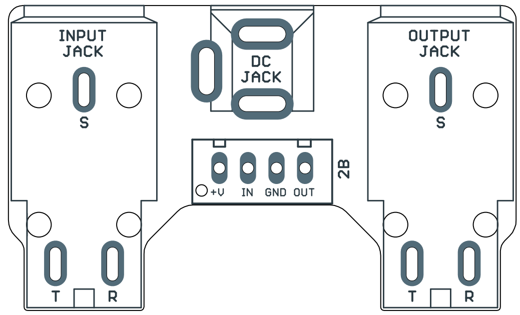




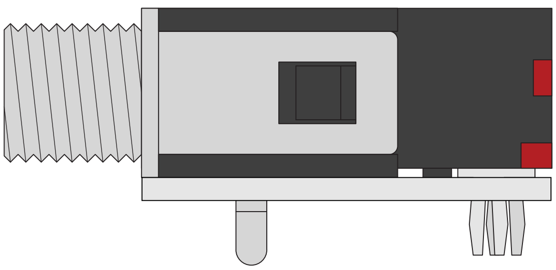
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

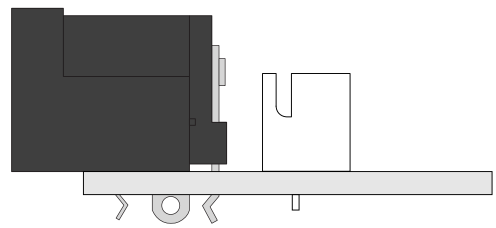
INPUT/OUTPUT PCB

PARTS
Input & output jacks
DC jack
Wire header



Almost done! Get the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.

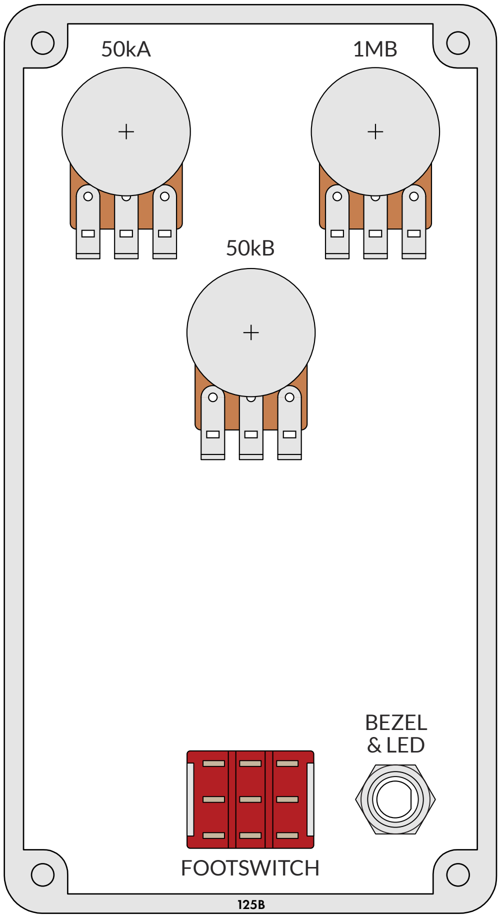




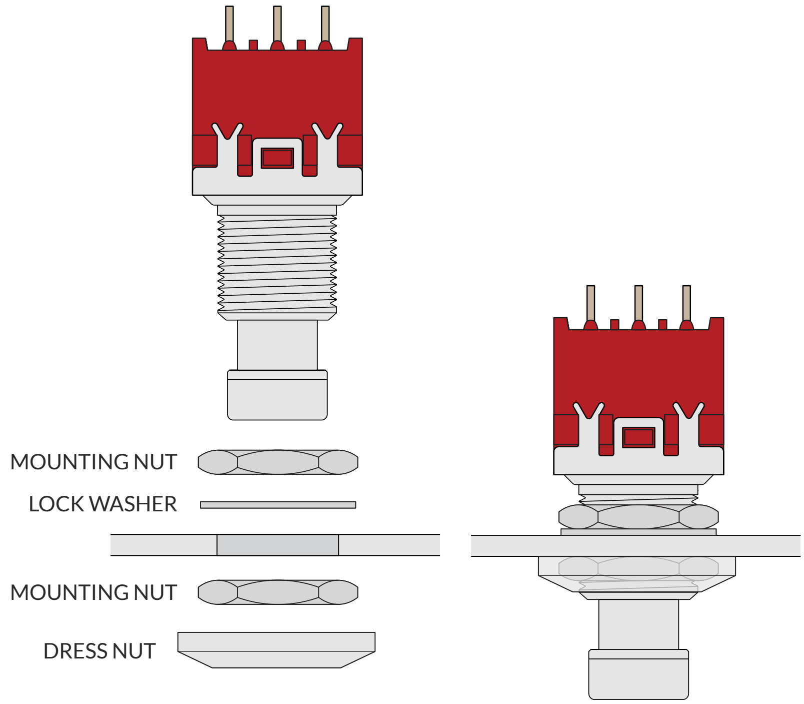
After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

ENCLOSURE LAYOUT: PANEL MOUNTS

Attach the hardware to the enclosure as shown. (The I/O board is done in a later step.)



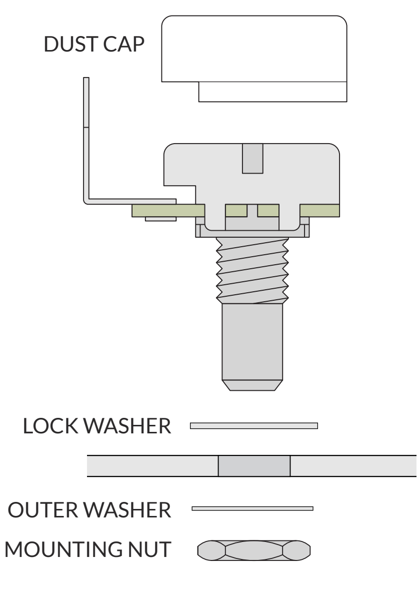
FOOTSWITCH

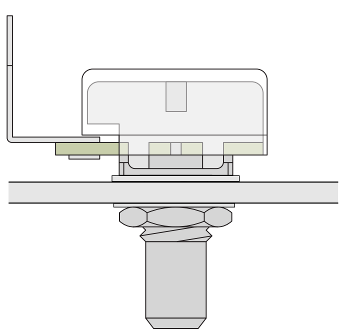


The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Avoid using metal tools on it or you run the risk of scratching or denting it.

POTENTIOMETERS

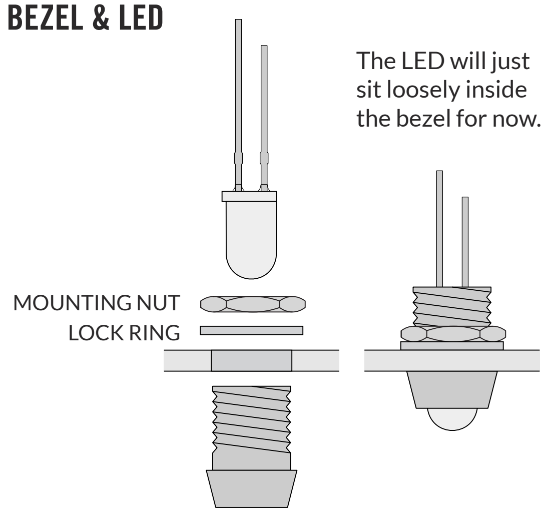
Feedback: 50kA
Delay: 1MB
Volume: 50kB





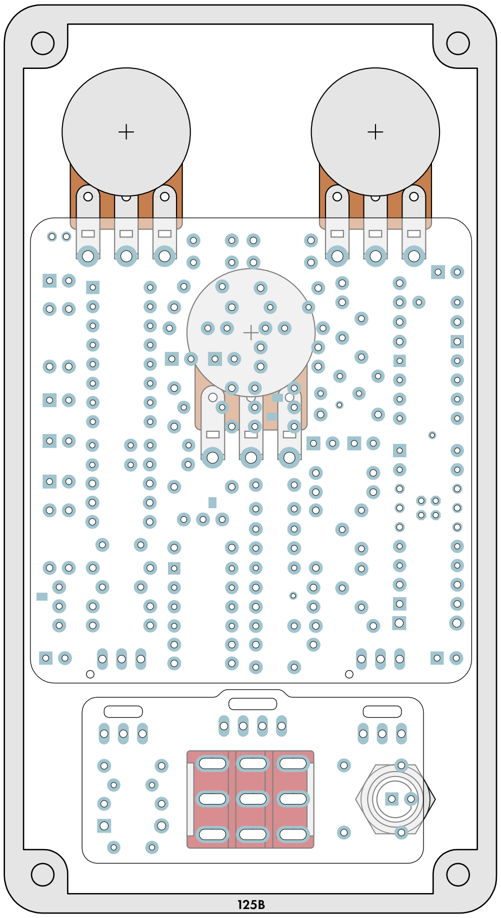
BEZEL & LED

The LED will just sit loosely inside the bezel for now.



You'll need to hold the bezel in place when tightening the nut. Be aware that the bezel is fairly sharp. Try using a rubber band for grip instead of just pressing your finger against the bottom.

ENCLOSURE LAYOUT: MAIN & FOOTSWITCH PCBS



After all the components are affixed to the enclosure as shown on the previous page, place the main PCB on top of the potentiometers as in the diagram to the left.

You may need to adjust the position of the potentiometers slightly if they are not aligned straight.

Once all of the pins are through and the PCB is laying flat, solder each of the pins from the top, being careful not to touch any of the surrounding components with the soldering iron.

After you've finished soldering the pots, **clip the leads as close as you can to the main PCB**. This is important because the input/output PCB overlaps them and you need to avoid any of the components shorting.

Next, do the same thing with the footswitch board—the 3PDT footswitch and the LED.

Before soldering, double-check to **make sure the flat side of the LED is facing to the right**, as shown in the diagram, and that the short leg is coming through the pad on the right. It won't work if it's turned the other way.

Why solder everything inside the enclosure before testing it?

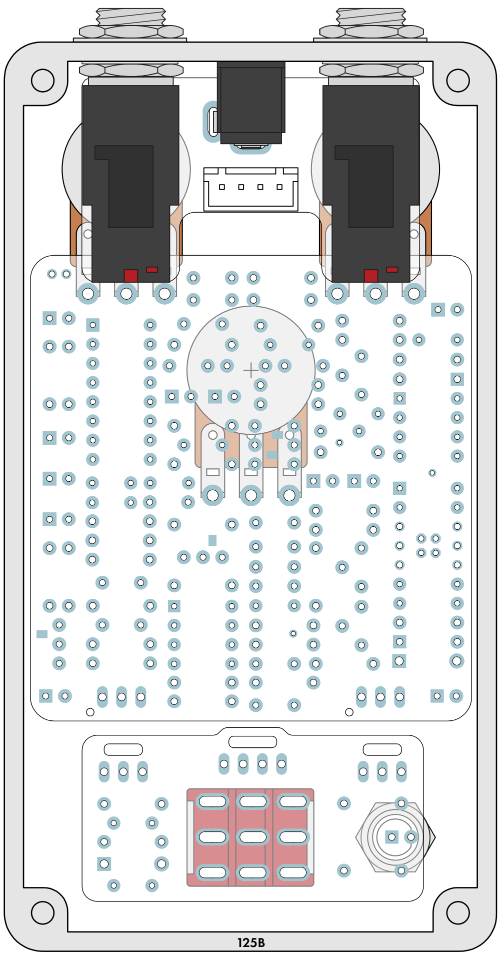
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting, with no desoldering required—so with these kits, it's actually much easier to "box it before you rock it".

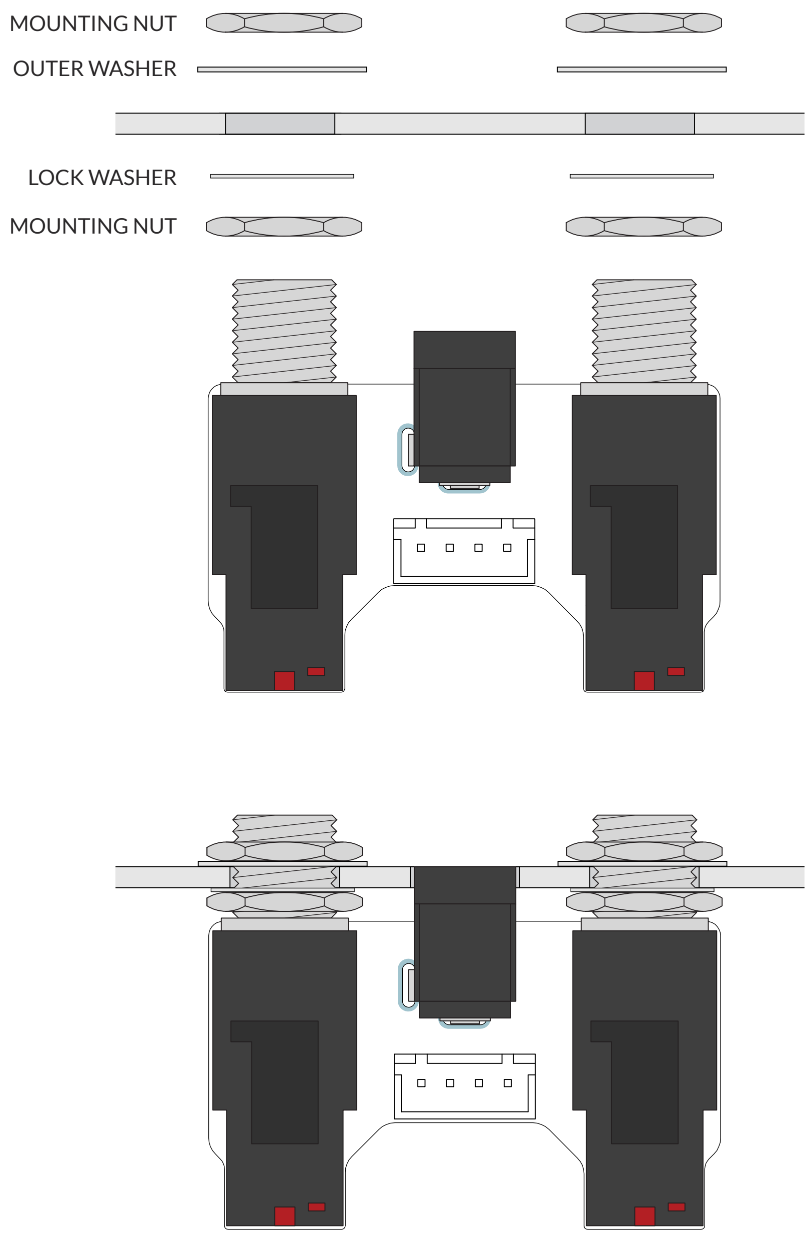
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

Affix the input/output PCB to the north-facing panel of the enclosure as shown.

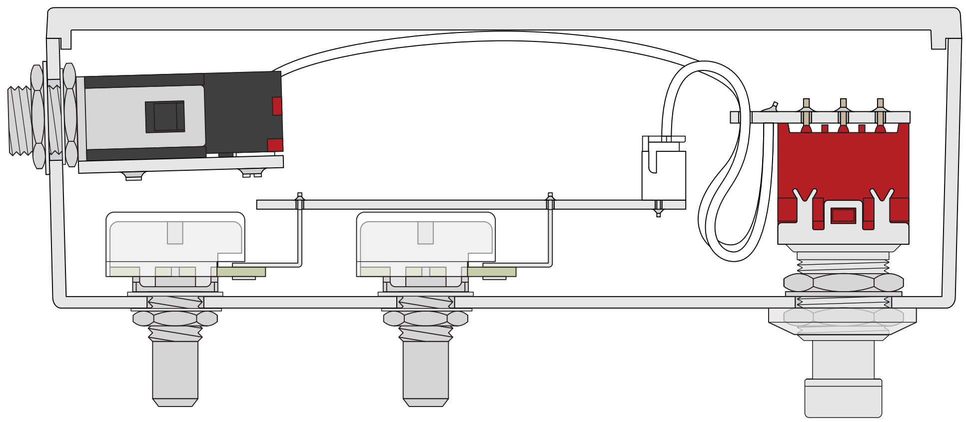
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





TESTING & ASSEMBLY

After everything is in place, just plug the 3 wire assemblies into their respective headers and make sure they're secure. Here is a cross-section of the inside of the completed pedal.



At this point, you have completed the full circuit as far as the electrons are concerned. Plug in a 9-volt supply and test it out with a guitar and an amplifier.

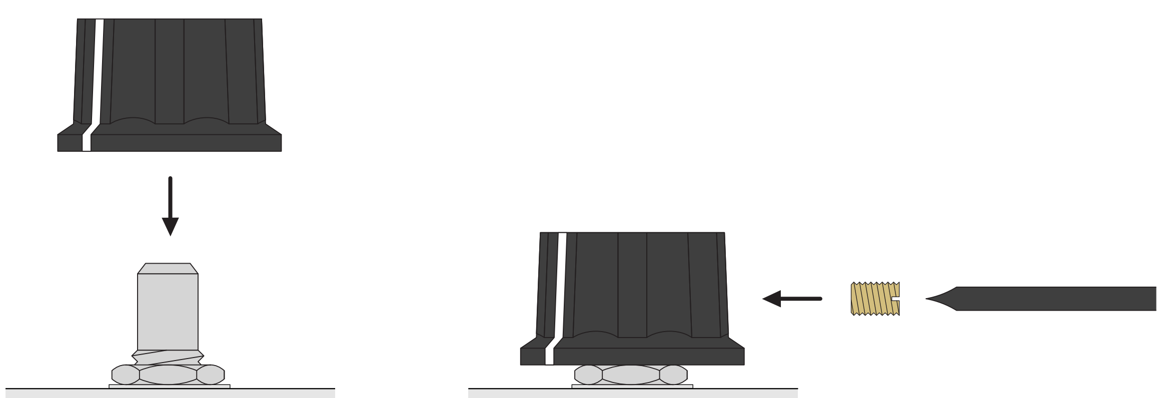
Test the bypass switch a few times, then leave it in effect mode and listen while you turn the knobs. Note that it has not been calibrated yet, so we're only testing to make sure it passes signal so we can proceed to the next step. We'll evaluate the sound as part of the calibration process.

Installing the knobs

Even though we're not finished, it's helpful to have the knobs installed for the calibration process. Turn the potentiometer shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

Be careful not to over-tighten or you may damage the set screw. But if it's not tight enough, the knob will be more likely to fall off or lose its alignment with the markings on the enclosure.



CALIBRATION

The Amethyst circuit has three different trimmers that need to be adjusted for best performance and minimal noise. While the calibration is best done with professional test equipment, it can be done by ear as well, and we have included instructions for both.

The calibration is relative to a specific supply voltage, so when following these steps it's recommended to power the pedal with the exact supply you plan on using in your rig. If you use a bench supply or something else, the difference in supply voltage may cause the calibration to change.

Before you begin either method, make sure all three trimmers are set to the center (12:00) position.

Calibrating by ear

If you don't have an oscilloscope available, you can get 90% of the way there or better just by using your ears, and if you compared it to a professionally calibrated unit, you might not notice a difference. DIYers have been building clones of the DM-2 circuit for decades and most of them did not use an oscilloscope, so it's not a requirement—it just means it's a more subjective process rather than objective.

Clock frequency

Set the Delay knob to maximum, the longest delay setting. Feedback and Volume can be set anywhere. Adjust the CLOCK trimmer until you hear a high-pitched squeal or whine, and then back off slightly until it goes away.

BBD bias

Set Feedback to minimum and Delay to halfway. Volume can be set anywhere. Adjust the BIAS trimmer until the delay signal has the lowest amount of audible distortion across the full range of the Delay control. In our experience, this will typically be around 1:00 with the type of BBDs used in the kit.

If you don't hear any delay signal at all, turn it up until you measure around 6 to 6.5V on pins 3 and 4. Sometimes it's higher than 1:00 on the trimmer, and if it's too low then the IC won't pass signal.

Clock cancellation

Set Delay to maximum and Feedback to minimum, then adjust the CANCEL trimmer between 12:00 and 2:00 and see if you hear any difference in background noise with no input signal. If not, just leave it at around 1:30. This is usually close to the optimal setting. Further fine-tuning requires test equipment.

Now, test all three knobs across the full range. If you notice any distortion, artifacts, or squealing, you'll need to go through the calibration steps again.

Calibrating with an oscilloscope

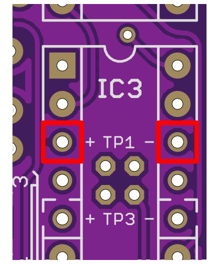
If you have access to an oscilloscope, you can look at the waveforms and fine-tune the bias visually rather than by ear. By following the calibration procedure from the factory service manual, which we've reproduced in the following pages, you're guaranteed the best possible results.

If you're more of a visual learner, here is an in-depth video showing the calibration of a Way Huge Aqua Puss clone. The procedure and target specifications are identical to the DM-2, only the part names are different. You'll just need to mentally translate a few of the part names and positions to this project.

Clock frequency

This step requires a multimeter with frequency measurement, sometimes called a frequency counter.

1. Set the Delay knob to maximum, the longest delay. Feedback and Volume can be set anywhere.
2. Connect the multimeter to the pads marked “TP1”, which is on the right side of the PCB immediately below pins 2 and 7 of IC3. The positive lead should connect to the left side and the negative lead to the right side. You can insert leftover resistor leads into the empty IC sockets and attach the probes to them while testing.
3. Adjust the CLOCK trimmer until you read a frequency of 6.8kHz.

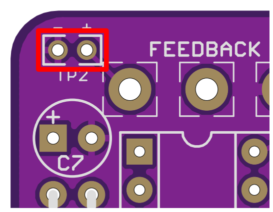


If you have an oscilloscope but not a frequency counter, you can use a scope to visualize the clock signal waveform on TP1. Adjust the CLOCK trimmer until one full square-wave cycle is 146μs (in other words, the time between the start of one positive cycle to the start of the next positive cycle).

BBD bias

For this step, you will need a signal generator and an oscilloscope. For the generator, you can use an application like ARTA, or even a phone app as long as you verify that the output level matches the spec.

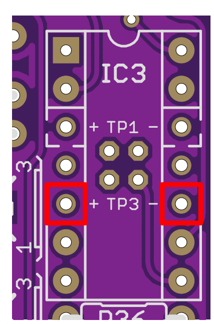
1. Set the Feedback to minimum, Delay to halfway, and Volume anywhere.
2. Set the signal generator for a 200 Hz sine wave at 0dBm. Connect this to the input jack of the effect, making sure it’s in effect mode.
3. Connect the oscilloscope to the pads marked “TP2”, which are located in the upper-left corner of the PCB. You can solder leftover resistor leads to these pads to make them easier to reach if everything is already installed to the enclosure. Note that the positive pad is on the right, the reverse of TP1 & TP3.
4. Adjust the BIAS trimmer until the waveform has the minimum amount of distortion or clipping, aiming for symmetry on the top and bottom halves. If you don’t see any clipping on the waveform, increase the level of the test signal until it appears.



Clock cancellation

For this step, you will need an oscilloscope.

1. Put the effect in bypass mode to ensure there is no input signal.
2. Set Delay to maximum and Feedback to minimum. Volume can be set anywhere.
3. Connect the oscilloscope to the pads marked “TP3”, which is on the right side of the PCB immediately above pins 3 and 6 of IC3. The positive lead should connect to the left side and the negative lead to the right side. You can insert leftover resistor leads into these empty sockets and attach the oscilloscope probes to them while testing.
4. Adjust the CANCEL trimmer until the two clock signals have converged with the closest amount of overlap.



FINAL ASSEMBLY

Now, test all three knobs across the full range. If you notice any distortion, artifacts, or squealing, you'll need to go through the calibration steps again.

If everything calibrated OK, you're all done! Affix the back panel with the four screws provided and make some music.

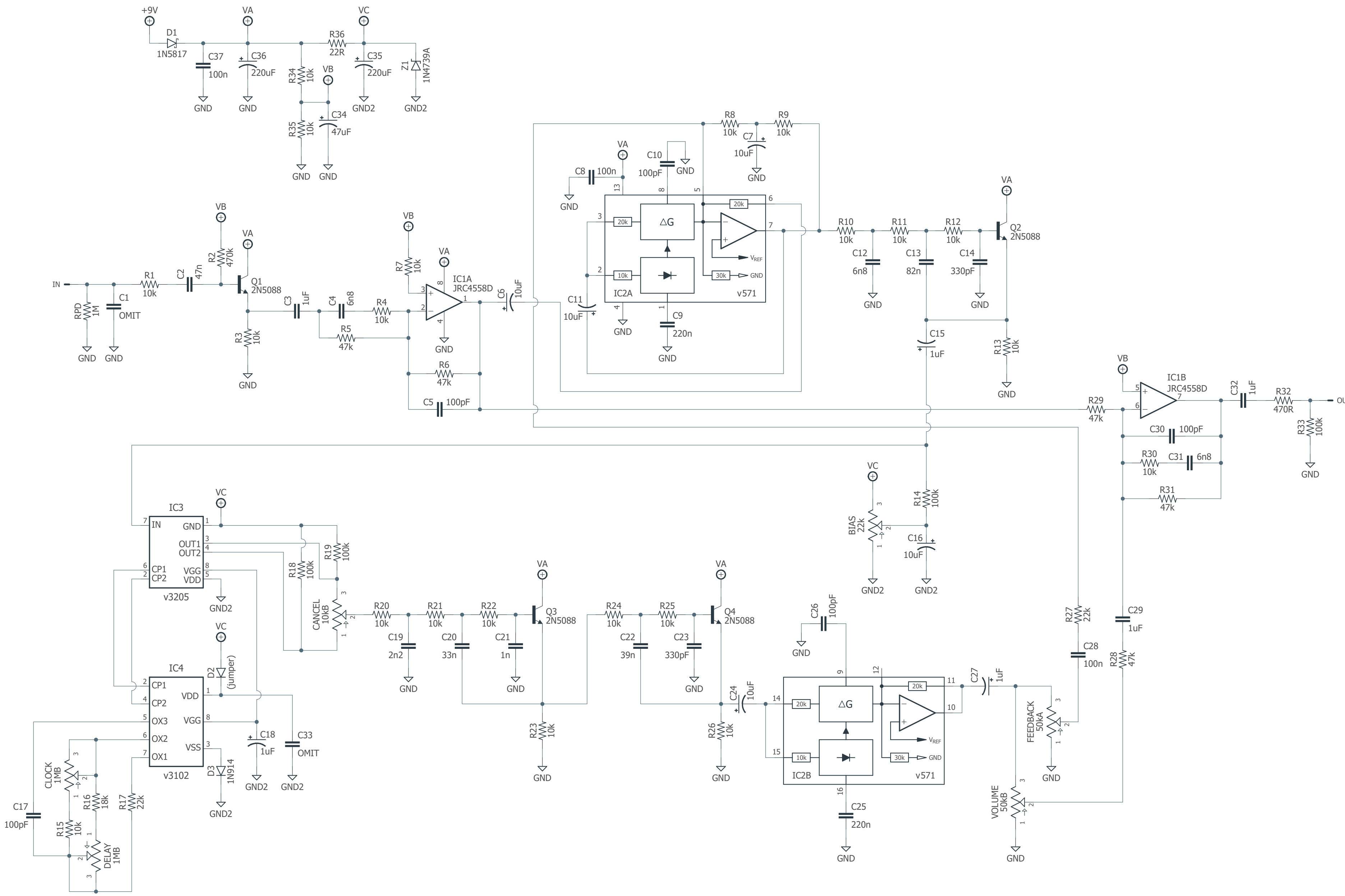
If doesn't work, or doesn't sound quite right after calibration, don't be discouraged. See page 31 for troubleshooting info.

USAGE

The Amethyst has three controls:

- **Delay** (called "Repeat Rate" on the DM-2) sets the delay time, shortest at minimum and getting longer as you turn it up. The rotation direction is reversed from the original DM-2, but consistent with most modern delays.
- **Feedback** (called "Intensity" on the DM-2) sets the number of repeats. Due to anti-alias filters, each successive repeat has degraded treble content, resulting in an ambient wash that doesn't conflict with the dry signal. The upper end of the range can get into self-oscillation or infinite repeats.
- **Level** (called "Echo" on the DM-2) sets the volume level of the delay signal. The dry signal is unaffected and always unity gain.

SCHEMATIC


+9V
D1
1N5817
C37
100n
VA
C36
220uF
R36
22R
VC
C35
220uF
Z1
1N4739A
R34
10k
VB
R35
10k
C34
47uF
GND
GND2
IN
RPD
1M
C1
OMIT
R1
10k
C2
47n
R2
470k
Q1
2N5088
R3
10k
C3
1uF
C4
6n8
R4
10k
R5
47k
R7
10k
IC1A
JRC4558D
C6
10uF
R6
47k
C5
100pF
C8
100n
C10
100pF
R8
10k
R9
10k
C7
10uF
ΔG
20k
10k
30k
V_REF
IC2A
v571
C11
10uF
C9
220n
R10
10k
R11
10k
R12
10k
Q2
2N5088
C12
6n8
C13
82n
C14
330pF
C15
1uF
R13
10k
IC1B
JRC4558D
C32
1uF
R32
470R
OUT
R33
100k
R29
47k
C30
100pF
R30
10k
C31
6n8
R31
47k
BIAS
22k
R14
100k
C16
10uF
IC3
IN
GND
OUT1
OUT2
CP1
CP2
VGG
VDD
v3205
R19
100k
R18
100k
CANCEL
10kB
R20
10k
R21
10k
R22
10k
Q3
2N5088
C19
2n2
C20
33n
C21
1n
R23
10k
R24
10k
R25
10k
Q4
2N5088
C22
39n
C23
330pF
R26
10k
C24
10uF
C26
100pF
IC2B
v571
C25
220n
C27
1uF
FEEDBACK
50kA
VOLUME
50kB
R27
22k
C28
100n
C29
1uF
R28
47k
IC4
CP1
CP2
OX3
OX2
OX1
VDD
VGG
VSS
v3102
D2
(jumper)
D3
1N914
C18
1uF
C33
OMIT
C17
100pF
CLOCK
1MB
R15
10k
R16
18k
DELAY
1MB
R17
22k

FULL PARTS LIST

Resistors

PART	VALUE
R1	10k
R2	470k
R3	10k
R4	10k
R5	47k
R6	47k
R7	10k
R8	10k
R9	10k
R10	10k

PART	VALUE
R11	10k
R12	10k
R13	10k
R14	100k
R15	10k
R16	18k
R17	22k
R18	100k
R19	100k
R20	10k

PART	VALUE
R21	10k
R22	10k
R23	10k
R24	10k
R25	10k
R26	10k
R27	22k
R28	47k
R29	47k
R30	10k

PART	VALUE
R31	47k
R32	470R
R33	100k
R34	10k
R35	10k
R36	22R
RPD	1M
LEDR	22k

Capacitors

PART	VALUE
C2	47n film
C3	1uF film
C4	6n8 film
C5	100pF MLCC
C6	10uF electro
C7	10uF electro
C8	100n MLCC
C9	220n film
C10	100pF MLCC

PART	VALUE
C11	10uF electro
C12	6n8 film
C13	82n film
C14	330pF MLCC
C15	1uF electro
C16	10uF electro
C17	100pF MLCC
C18	1uF electro
C19	2n2 film

PART	VALUE
C20	33n film
C21	1n film
C22	39n film
C23	330pF MLCC
C24	10uF electro
C25	220n film
C26	100pF MLCC
C27	1uF electro
C28	100n film

PART	VALUE
C29	1uF film
C30	100pF MLCC
C31	6n8 film
C32	1uF film
C34	47uF electro
C35	220uF electro
C36	220uF electro
C37	100n MLCC

Transistors

PART	VALUE
Q1	2N5088
Q2	2N5088
Q3	2N5088
Q4	2N5088

Diodes

PART	VALUE
D1	1N5817
D3	1N914
Z1	1N4739A

ICs

PART	VALUE
IC1	JRC4558D
IC2	v571D
IC3	v3205SD
IC4	v3102D

Sockets

PART
DIP-8 socket (4)
DIP-14 socket

FULL PARTS LIST, CONT.

Trimmers

PART	VALUE
CANCEL	10k
BIAS	22k
CLOCK	1M

Potentiometers

PART	VALUE
Delay	1MB
Feedback	50kA or 50kB
Volume	50kB

Switches

PART
3PDT stomp

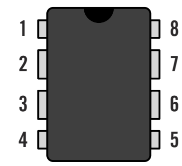
TROUBLESHOOTING INFORMATION

If you finish building the kit and find that it doesn't work right, we've written a separate in-depth Troubleshooting Guide that applies to all of our kits. The main troubleshooting process is covered there. Here you will find information specific to this kit that will help with that process.

Voltages

The following voltages are taken from our prototype unit using a **9.6V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything more than +/-0.5V from the listed voltages, it's a good indicator of an issue, and the exact voltages can help narrow it down.

Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right. Transistors and JFETs have their pins labeled on the PCB.



All voltages taken with Delay at maximum (longest delay) and Feedback at minimum.

IC1

PIN	VOLTAGE
1	4.67
2	4.67
3	4.67
4	0
5	4.67
6	4.67
7	4.67
8	9.35

IC2

PIN	VOLTAGE
1	1.75
2	1.85
3	1.85
4	0
5	1.85
6	1.85
7	3.03
8	1.85

PIN	VOLTAGE
9	1.85
10	3.18
11	3.18
12	1.85
13	9.35
14	1.84
15	1.85
16	2.32

IC3

PIN	VOLTAGE
1	0
2	4.58
3	6.29
4	6.23
5	8.85
6	4.56
7	5.42
8	8.31

IC4

PIN	VOLTAGE
1	8.85
2	4.56
3	0.63
4	4.58
5	4.71
6	4.67
7	4.70
8	8.32

Q1

PIN	VOLTAGE
C	9.35
B	4.15
E	3.74

Q2

PIN	VOLTAGE
C	9.35
B	3.00
E	2.43

Q3

PIN	VOLTAGE
C	9.35
B	6.11
E	5.53

Q4

PIN	VOLTAGE
C	9.35
B	5.50
E	4.92

SUPPORT

Aion FX does not offer direct support for these projects beyond the provided documentation. Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error or that the included components are non-functional.

Where to get help

The three best places to ask for help are the DIY Stompboxes forum, the DIY Stompboxes Facebook group, and the r/diypedals subreddit. These communities have more than 150,000 members between them and they are very accommodating to new builders.

When posting a troubleshooting request, always include the following:

1. A thorough description of the problem you are experiencing
2. A photo of the inside of the pedal
3. A list of all the measured voltages of each of the pins, described on the previous page

While we cannot offer direct, private support, you may send a link to your public troubleshooting thread to Aion FX using the contact form on the website. There is no guarantee that we will be able to join the discussion and help solve your problem, but this improves the chances.

It benefits the whole community if the troubleshooting process is public because then people who have the same issue in the future may come across it when searching. And if you do get help, remember to pay it forward! The best way to learn new skills is to help others. Even if you've only built one pedal, you have more experience than someone who is brand new, so you have something to offer.

RESALE TERMS

These kits may be used for commercial endeavors in any quantity unless otherwise noted. It's okay to sell individual builds locally or online, or even to offer a service to build pedals based on these kits.

No direct attribution is necessary, though a link back is always greatly appreciated. The only usage restriction is that you cannot "goop" the PCB or otherwise obscure the source. In other words: you don't have to go out of your way to advertise the fact that you use Aion FX kits, but please don't go out of your way to hide it. The guitar effects industry needs more transparency, not less!

LEGAL INFORMATION

DOCUMENT REVISIONS

1.0.2 (2025-01-05)
Changed Feedback pot from 50kB to 50kA so that there is a greater range of control before the self-oscillation begins.

1.0.1 (2024-07-01)
Added note about 22R vs. 22k resistors.

1.0.0 (2024-06-01)
Initial release.