Marshall 'Blues Breaker'

Manual & Assembly Guide

Fore note;

It is recommended that you read the whole of this document (including Trouble-shooting) before commencing work on assembly of this kit.

Please DO NOT remove the seal from the components bag until you have done so!

Introduction

This document forms the assembly instructions and manual for the DIY stomp box Kit. It contains important information enabling you to build, test and use a classic audio effect processor based on a vintage and classic design, with relative ease, and at a fraction of the cost of buying new or 2nd hand, with your own customizations!

Although DIY stomp boxes are generally simple to build, a minimum knowledge of basic electronics is required in order to proceed. If you are in any doubt about your abilities with soldering, PCB assembly, use of test tools, or a basic understanding of Ohm's Law, it may help to improve those skills before taking on the challenge of building this kit! The extra knowledge will be rewarding, and enable you to further take on more DIY projects, and progress to become an expert!

In order to succeed, please take your time, don't rush, and it is highly recommended - no matter what level of expertise - to seek support from the vast amount of resources that now exist in the DIY Audio electronics community online, if not only to show off your work!

Background: Marshall 'Blues Breaker'

The Marshall Bluesbreaker is the popular name given to the Models 1961 and 1962 guitar amplifiers made by Marshall from 1964/1965 to 1972.

The Bluesbreaker, which derives its nickname from being used by Eric Clapton with John Mayall & The Bluesbreakers, is credited with delivering "the sound that launched British blues-rock in the mid-1960s".

This Pedal is a clone of the original using Integrated circuits rather than the original Valves as used in the Combo box. The sound character is of a warm, soft clipping character that resembles the original, distinctive tone.

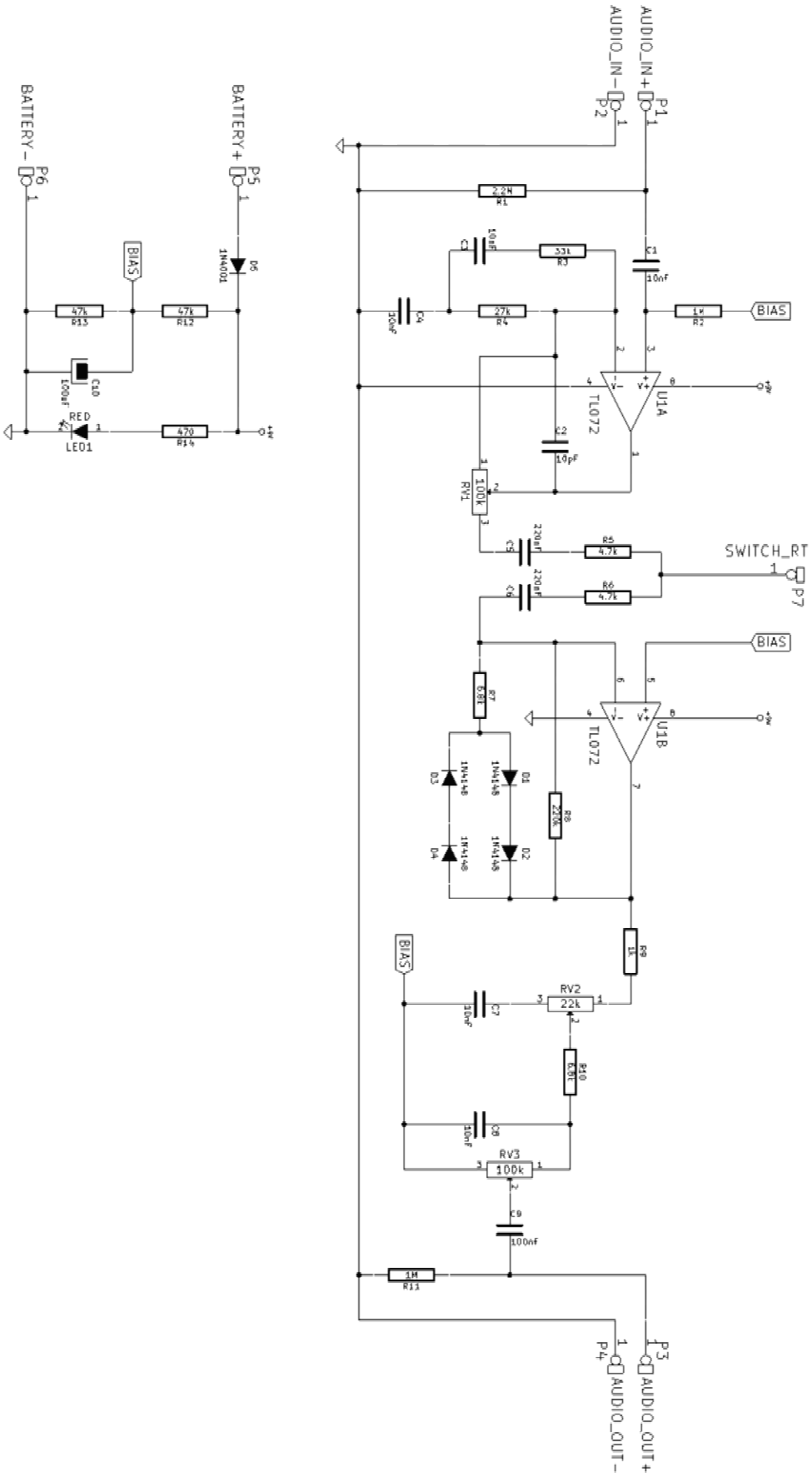
EMS Implementation: Marshall 'Blues Breaker'

The implementation in this kit is the original 1992 Blues breaker design produced by Marshall Corporation in its re-issue (the original combo box was Valve based). The design is exactly the same as the 1992 version with nothing added or removed. Only the Biasing arrangement (C10) slightly differs - the original having another 100uF across R12. This is not required and has no effect on the sound.

In addition, the original 1992 version has a 1N4007 rather than a 1N4001 for Battery reversal protection. The 1N4001 is more appropriate for this circuit, and has no effect on the sound.

There are many implementations of this pedal in the DIY community and 'mods' that can be performed to alter the sound - well worth checking out.

Fig 1: Schematic Diagram



Circuit Operating Principles

C1 functions as both a **1st Order HPF,** and **DC Blocking Capacitor** to filter the input and isolate the circuit. **R1** controls the input impedance of the circuit. We must Bias the input to the Operational Amplifier to 1/2 the Supply Voltage (explained later), and C1 effectively blocks this DC Bias from flowing back out of the input.

The Operational Amplifier IC **U1A** forms a simple Non-inverting Amplifier. A feedback loop formed by **R3, C3, R4, C4, C2** and the **GAIN** potentiometer **RV1** control the gain of this section. The gain is deliberately configured to cause the Amplifier to saturate, thus adding Harmonic Distortion from Amplifier Overloading. Note the GAIN also controls the gain of the second amplifier through limiting the input amplitude.

The function of **C4** in the feedback loop is to prevent the DC component (Bias) from being amplified, but it also gives this feedback loop some Frequency dependant filtering. Since the Reactance of C4 gets smaller with Higher Frequencies, the Gain of this stage will be smaller with Higher Frequencies, giving a Low-Pass response for the entire Amplifier. From the output from this Amplification stage is then passed through a Frequency network formed from **C5, R5, C6, R6**. The centre section is grounded when the Pedal is in Bypass mode - this can be ignored if required.

The second half of the Operational Amplifier **U1B** forms an Inverting Amplifier, the Feedback loop implementing a non-linear response via **D1, D2, D3 & D4**. This results in an Overdrive effect producing a unique sound characteristic when combined with the saturation from the first Amplifier section.

R9, R10, TONE, C7 & C8 form a **LPF** whose frequency response characteristic at the output is controlled by the TONE pot **RV2**, and the final output of this section is fed to the **VOLUME** control **RV3**, this being a simple potential divider to mix between the full output signal and ground. Finally, **C9** functions as another DC Blocking Capacitor. Since we originally biased the input, we must now remove this Bias before sending it to the output.

The final signal is then passed through the **FOOTSWITCH**, (see wiring diagram) allowing selection of either the original input signal (bypass), or the distorted signal.

The power supply begins with a rectifier diode **D5** (to prevent damage if the battery is connected the wrong way around). **R12** and **R13** provide the Bias level of exactly half the supply voltage, enabling the Operational Amplifiers to amplify the signal without a negative supply Voltage. **C10** is included to slow down the Bias transient, preventing an audible thump during power-up.

Finally a power **LED1** is added, this driven via the current limiting resistor **R14**. The Circuit can be powered from a single 9v Battery or **9v-12v DC Power Supply**

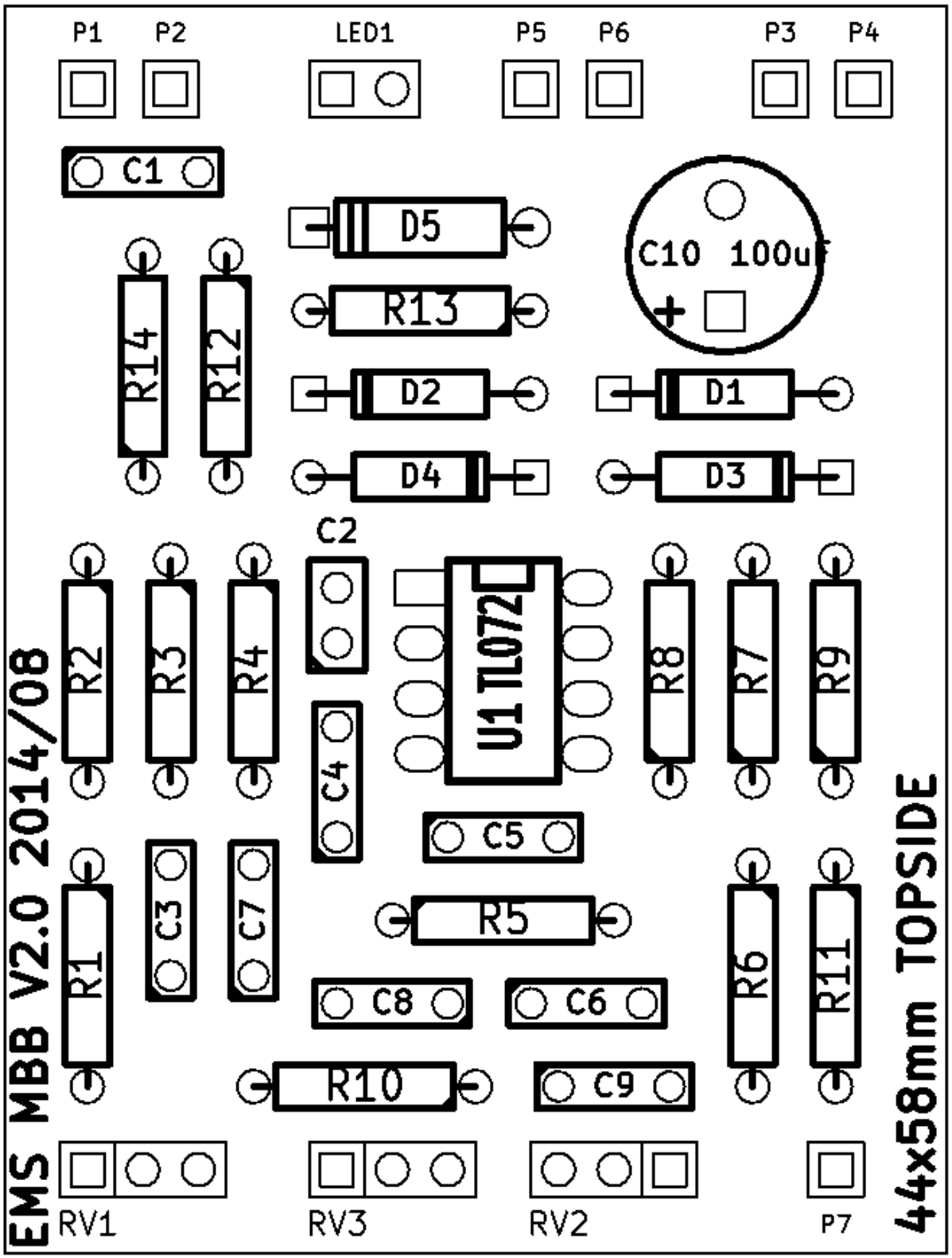
Bill of Materials

Component ID	Value	Type	Notes
R1	2.2MΩ	Resistor 0.25 Watt	
R2, R11	1MΩ	Resistor 0.25 Watt	
R3	33kΩ	Resistor 0.25 Watt	
R4	27kΩ	Resistor 0.25 Watt	
R5, R6	4.7kΩ	Resistor 0.25 Watt	
R7, R10	6.8kΩ	Resistor 0.25 Watt	
R8	220kΩ	Resistor 0.25 Watt	
R9	1kΩ	Resistor 0.25 Watt	
R12, R13	47kΩ	Resistor 0.25 Watt	
R14	470Ω	Resistor 0.25 Watt	Sets LED Current
C1, C3, C4, C7, C8	10nF	Polyester Capacitor	
C2	10pF	Ceramic Capacitor	
C5, C6	220nF	Polyester Capacitor	
C9	100nF	Polyester Capacitor	
C10	100uF	Electrolytic Capacitor	Check Orientation +
D1, D2, D3, D4	-	IN4148	Check Orientation +
D5	-	IN4001 Rectifier Diode	Check Orientation +
U1	-	TL072 Operational Amplifier	Check Orientation #
IC SOCKET	-	8 PIN Dual-in-Line (DIP)	Check Orientation #
PCB	-	EMS BLUES BREAKER	See Assembly
GAIN POT	100kΩ	Linear	Linear
TONE POT	22kΩ	Linear	Linear
VOLUME POT	100kΩ	Logarithmic Type (100K LOGB)	Logarithmic
May/May not be supplied in Kit depending on Version			
POWER SWITCH	SPST	Single Pole Single Throw	Radial Housing
KNOB x 3		Radial Mounting	Must fit POTS
FOOTSWITCH	DPDT	DPDT Footswitch (Knitter)	Check Orientation #
DC POWER JACK	-		Check Orientation +
AUDIO JACK x2	-	1/4 Inch Mono or Mono Switched	Check Orientation +
CASE	-	Aluminum	Die-cast Stomp Box
LED1	-	User defined	Check Orientation +
BATTERY	9v		Check Orientation +

PCB Connections

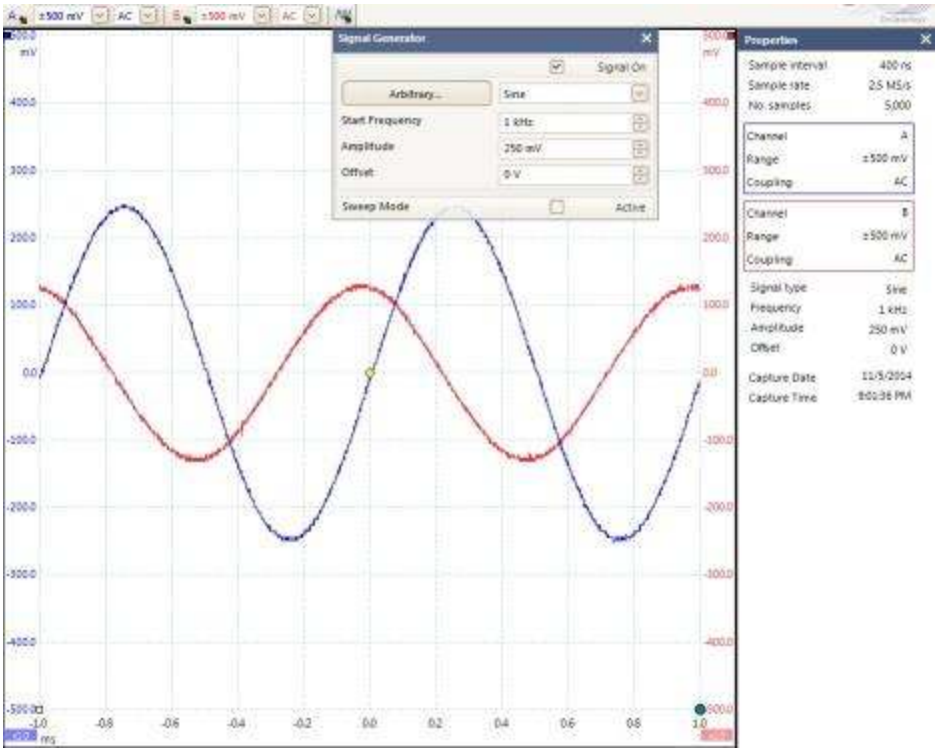
PCB Legend	Function	Notes
P1	AUDIO IN +	Also send to FOOTSW for Bypass
P2	AUDIO IN -	GND
P3	AUDIO OUT +	Send to FOOTSW for Effect On
P4	AUDIO OUT -	GND
P5	BATTERY +	Can use DC Input +VE
P6	BATTERY -	Can use DC Input -VE (GND)
P7	SWITCH RETURN	Connect to GND when OFF (can be **NOT CONNECTED**)
RV1, RV2, RV3	GAIN, TONE, VOLUME	Pin 1 is SQUARE Pad
LED1	POWER ON LED	+VE is SQUARE Pad

Fig 2: PCB Overlay

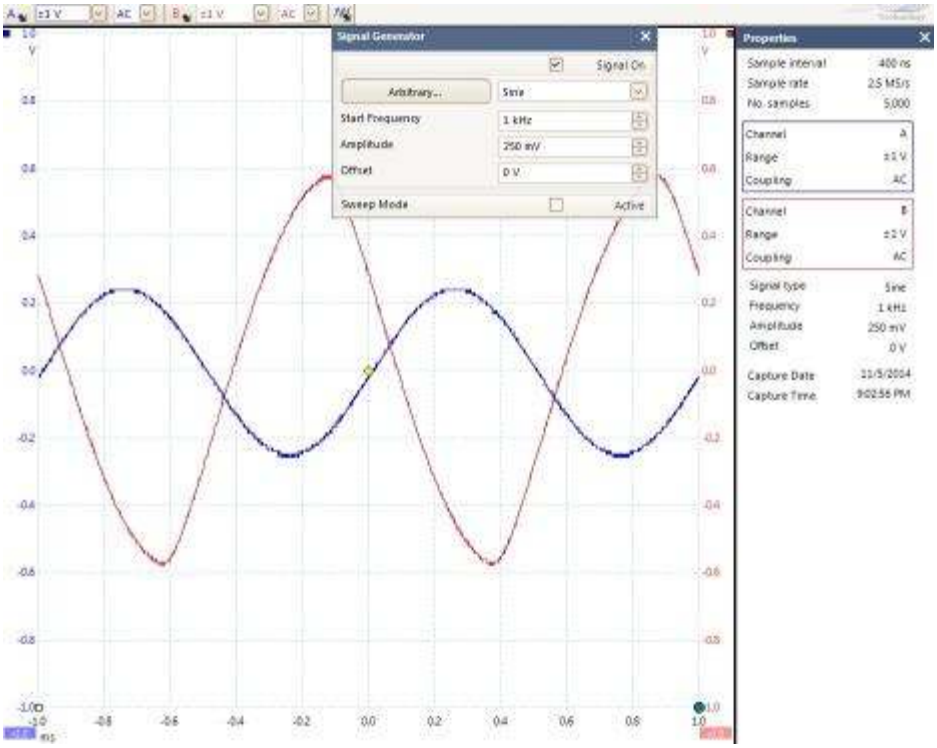


Note: P7 can be NOT CONNECTED. If required, use FOOTSWITCH to connect to GND when effect OFF.

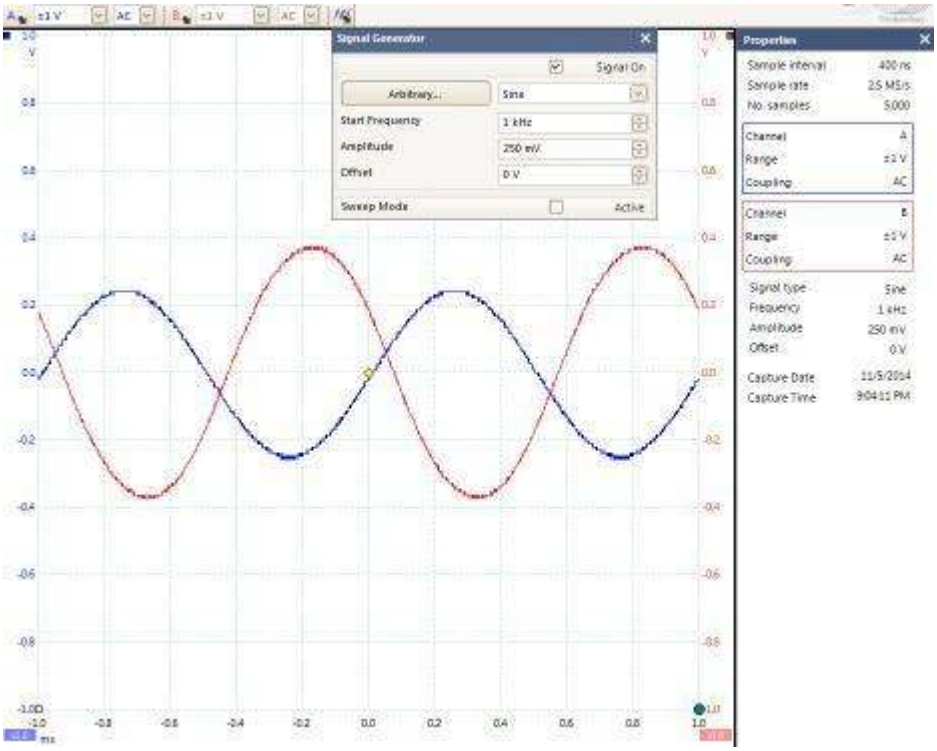
Fig 3: Test Waveforms



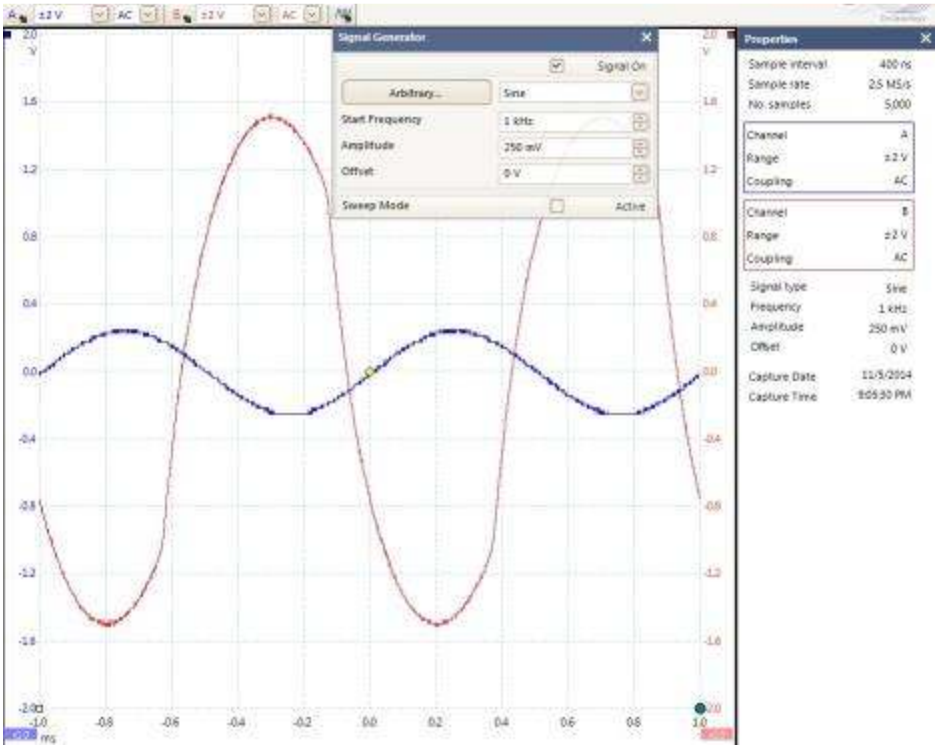
GAIN MIN (RV1 = MIN), TONE MIN (RV2 = MIN). (500mV P-P), Minimum Distortion



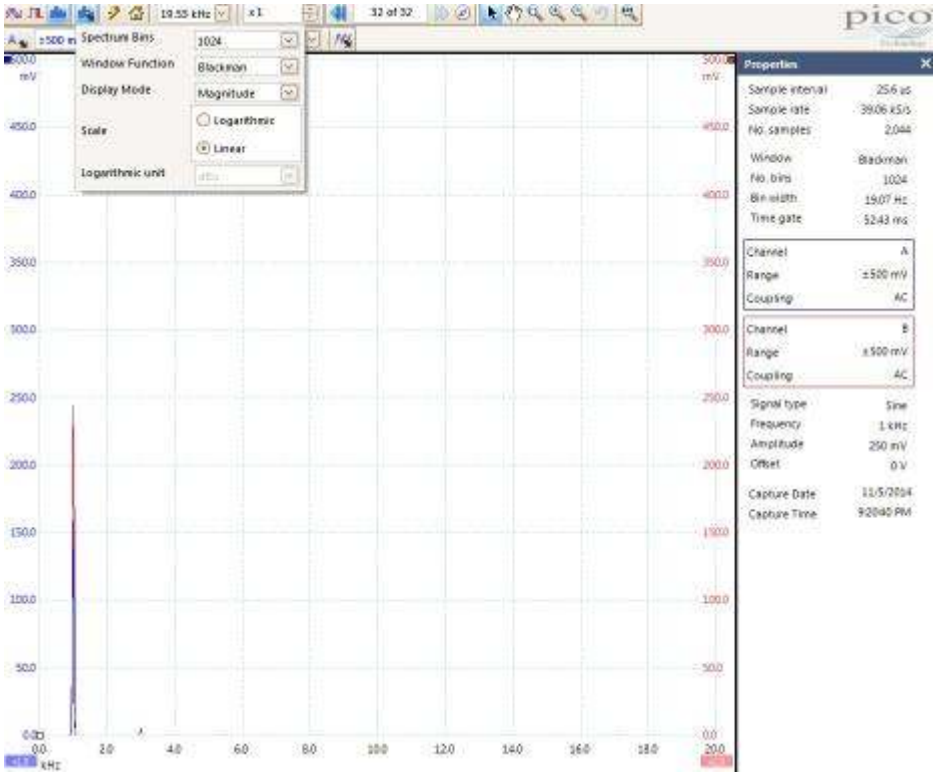
GAIN MAX (RV1 = MAX), TONE MIN (RV2 = MIN). (500mV P-P), Soft Clipping



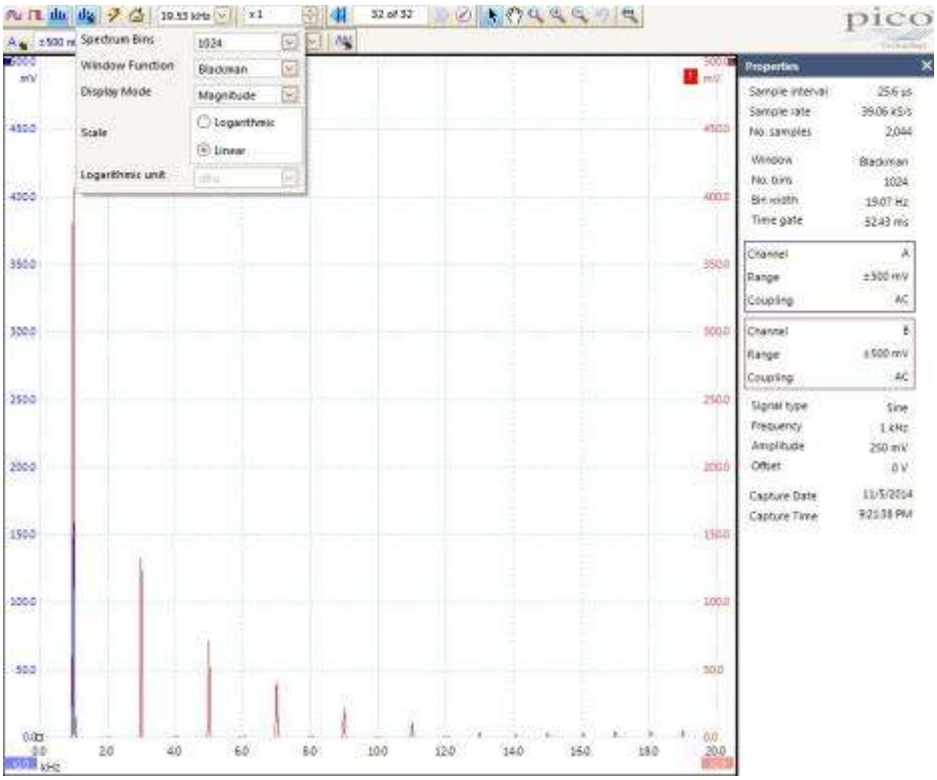
GAIN MIN (RV1 = MIN), TONE MAX (RV2 = MAX). (500mV P-P), Minimum Distortion



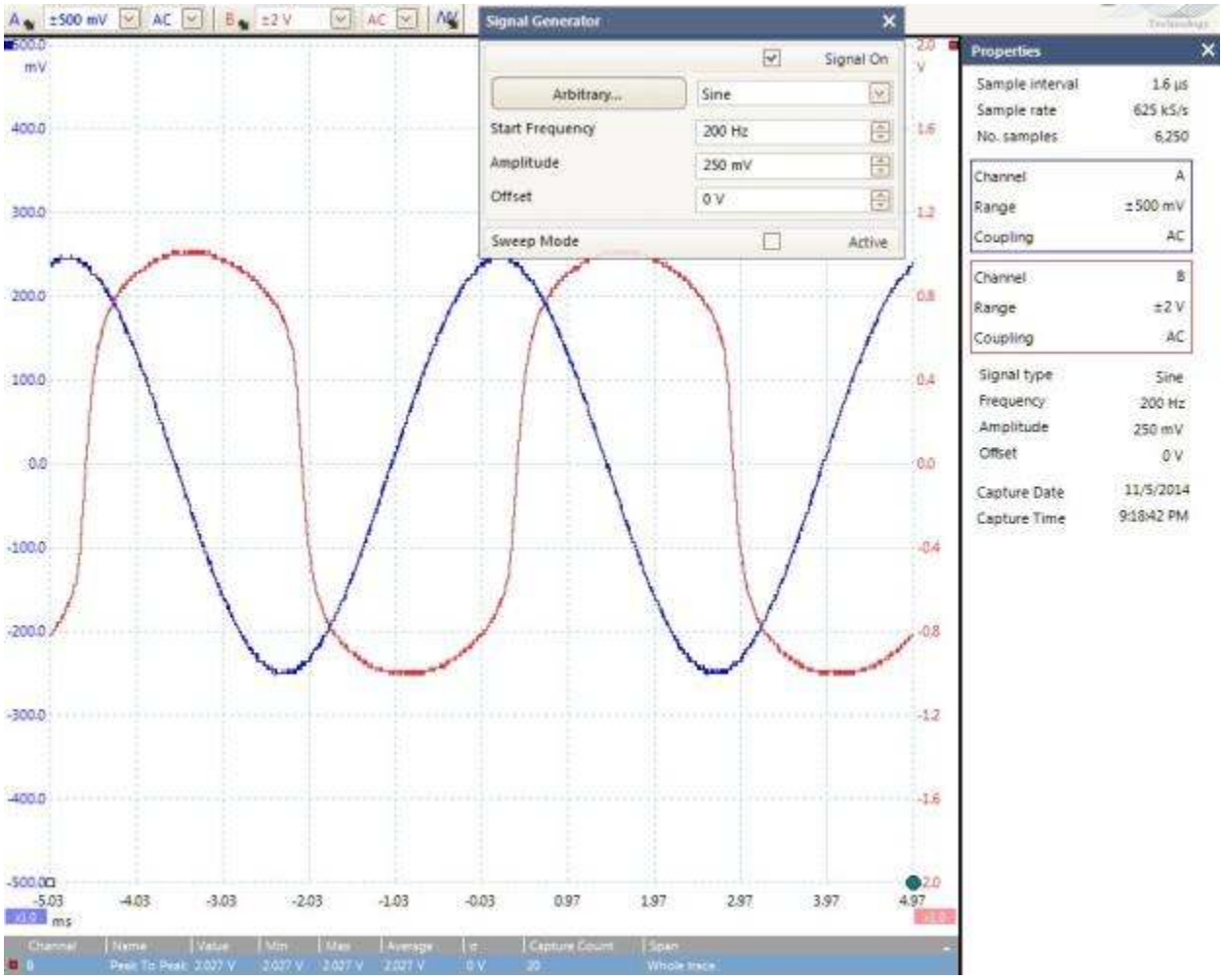
GAIN MAX (RV1 = MAX), TONE MAX (RV2 = MAX). (500mV P-P). Soft Clipping



AC 1kHz, Spectrum, 1024 bins, Blackman, Magnitude, Linear, RV1 POT = MIN, RV2 POT = MAX



AC 1kHz, Spectrum, 1024 bins, Blackman, Magnitude, Linear, RV1 POT = MAX, RV2 POT = MAX



Example of tube-like SOFT CLIPPING (typical usage scenario), (500mV P-P)

Assembly and Test Instructions

Equipment required;

1. Clean work area (table top) with Anti-static mat if available
2. Soldering Iron (15 - 30W)
3. Soldering Iron stand with damp (not wet) sponge
4. Soldering Iron tip cleaner (if available)
5. Bottlenose Pliers (2 pairs if available)
6. Tin-snips (Pliers)
7. Wire Strippers (correct size for chosen wire)
8. Safety Goggles (Essential)
9. 0.75mm or less Solder (Tin/Lead combination)
10. Anti-static Wristband or Earthed metal object (e.g. plumbing pipe)
11. Multi-meter (if available)
12. Oscilloscope (if available) a PC oscilloscope such as PicoScope is a worthwhile investment
13. Signal Generator (if available) often part of PC oscilloscope
14. DC Power Supply (Bench PSU, DC adapter, or Battery)
15. Print off of PCB Bill of Materials/Parts list
16. Print off of PCB Overlay from this manual
17. Print off of Schematic diagram from this manual
18. Print off of Wiring diagrams from this manual
19. Print off of pin-out from Data-sheet for ALL active components (e.g. IC's (Op-amp), Transistors, etc.). These are freely available online via a Part Number search e.g. "RC4558"

IMPORTANT

Before you start, ensure you have discharged any static Electricity from your body, by either touching an Earthed piece of bare metal (e.g. water pipe, back of PC, etc.) or using an anti-static wristband connected to ground. Failure to do so could destroy active components without any visible damage.

Make sure the work area is safe and clean, any stray cables are tucked away, and any potential hazards identified (e.g. stepping on soldering iron mains cable may cause a burn or damage!).

TAKE YOUR TIME during assembly. It is EASY to make mistakes and HARD to undo mistakes. Make sure you read each step fully and double-check or triple-check each step. A rush job is guaranteed to fail.

Maximize the resources available - print off this document if possible, and attempt to obtain all equipment listed. The correct tool for the job is important. There are many online resources available regarding soldering technique for example. Do not rely on information within this document ONLY. Research and practice first. A single mistake can ruin the project.

Make sure you NEVER touch anything POWERED up with the soldering Iron. This could result in an Electrical shock. NEVER SOLDER SOMETHING THAT IS POWERED ON.

Take time to "warm-up". Practice some soldering on an old piece of board or broken PCB to get used to the technique (even if you are an expert) before starting. This helps to maintain consistent quality control.

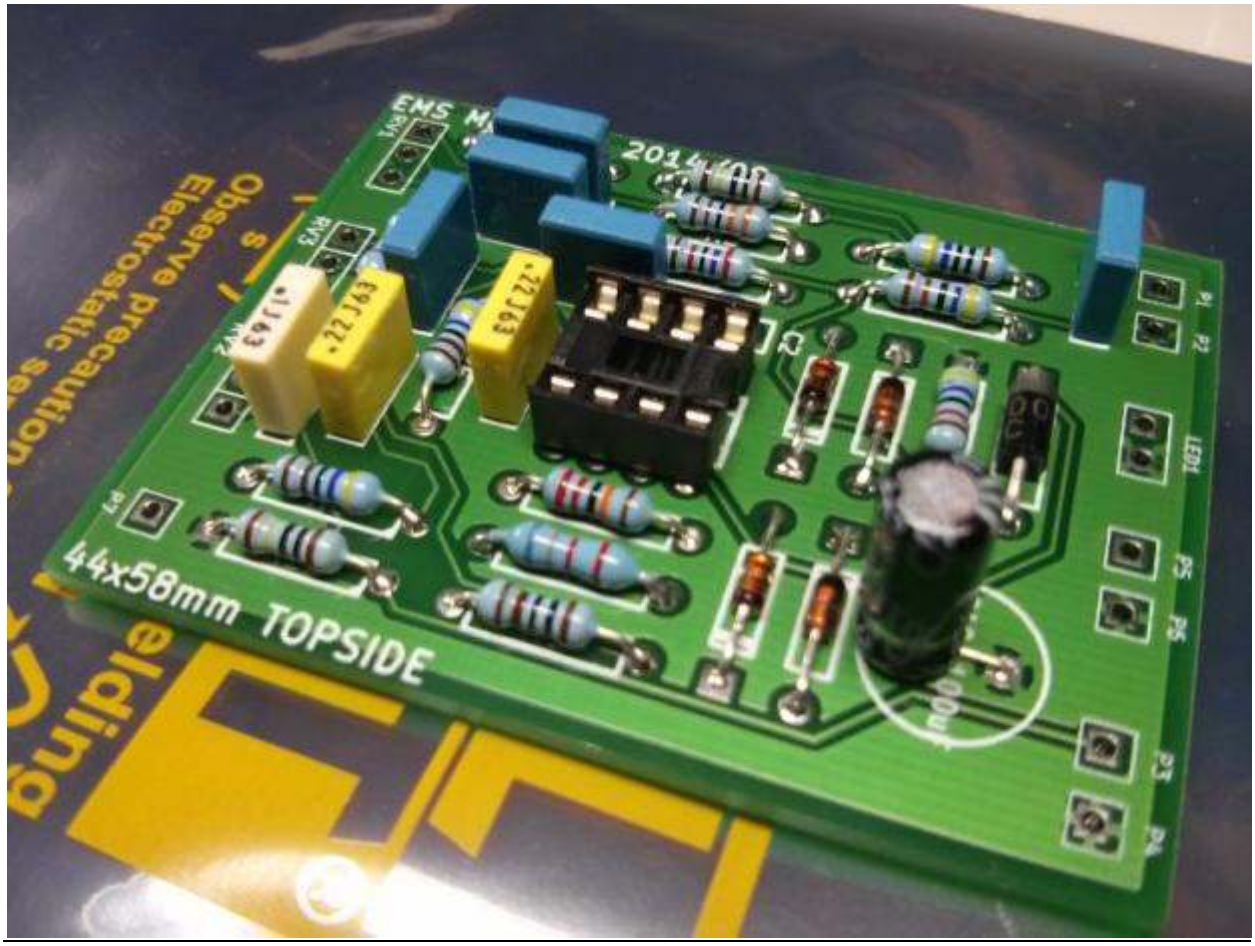
PLEASE READ THE ABOVE POINTS BEFORE PROCEEDING…

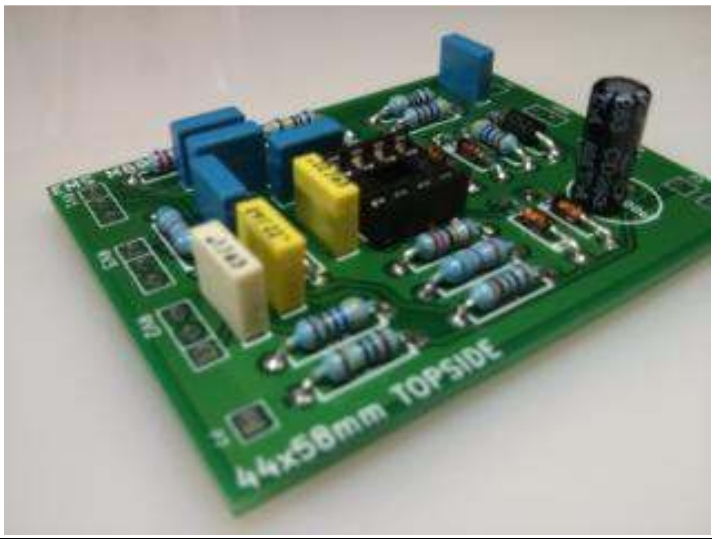
PCB Assembly Instructions

Method;

1) Determine which part refers to R1, D1, etc on the BOM. For Resistor/Capacitor Codes etc see Appendix. A multi-meter can also be used to measure the component and find the Value.
2) Put any IC's (e.g. the Op-Amp) to one side - a Socket will be provided and this is inserted last.
3) Label and organize components in terms of the Height of the body from the board. When assembling, you should with the Lowest (smallest) first, and work upward. E.g. it is likely the 1N1418 diode (or similar) will be the lowest component on the board, so organize so that this is the FIRST component to be installed.
4) Starting with the smallest (lowest) component, mount each component with the following procedure onto the PCB - TAKE YOUR TIME with each part;
   a) Find the position the component will go (e.g. R1 will be shown as a box with R1 on the PCB)
   b) Double check it is the CORRECT component, a mistake will be difficult to fix
   c) Determine if the component has to be placed a certain way around (polarized), this will be advised in the parts table (BOM). If it is an IC, ensure that Pin 1, is in the correct place for Pin 1 on the PCB. This is essential! If a polarized Capacitor check which end is +Ve.
   d) If a Resistor or Diode, place the component on top of the position to be placed, and try to gauge the width between the two connection points.
   e) Using the Bottle-nose pliers, for each side of the part next to the body, bend the legs at 90 degrees (if horizontal) to attempt to make the legs sit nicely into the PCB holes. i.e. achieve a "staple" like appearance to the part for a Resistor.
   f) Insert the component (top side) and push through the board, ensuring it is flush with the PCB.
   g) Turn the PCB over, and solder the component to the pads. Ensure you make a good solder joint - see Appendix
   h) If you cannot turn the board over without the part falling out, use tape to hold it in place.
   i) Whilst holding the excess lead and ensuring you are wearing safety goggles, use the Tin-snips to cut the leg exactly where it leaves the joint.
   j) If the joint is poor, with fresh solder, re-solder the joint to ensure it is of high quality
5) Move to the next Component in order of height.
6) As a general tip, Diodes (D1-Dn) will be the smallest, followed by Resistors (R1-Rn), followed by the IC Socket (U1-Un), followed by Capacitors (C1-Cn) however this is specific to the kit you are assembling.
7) Note: DO NOT INSERT THE IC yet! Only insert the IC into the socket at the last opportunity, e.g. when you have fully wired the board etc and are ready to screw the box together or test the circuit.
8) Once all the components have been, check it matches the overlay diagram (also picture of PCB)
9) Inspect the finished board for any bad joints (all joint should be shiny and in a Volcano shape - see Appendix).
10) Inspect the finished board for any solder splashes or stray lead that may cause a short circuit between points.
11) Lightly clean the PCB on both sides with a dry toothbrush or similar, to ensure it is clean and there is no stray matter.
12) Place the PCB back into the Anti-static (silver) bag and put to one side.
13) Clean the work area, including soldering Iron stand, and return the work area to the state when you began assembly - i.e. clean!

Fig 4: Assembled Example PCB





Note: P7 can be left NOT CONNECTED or should be routed to GND when the PEDAL is in BYPASS Mode using the FOOTSWITCH.

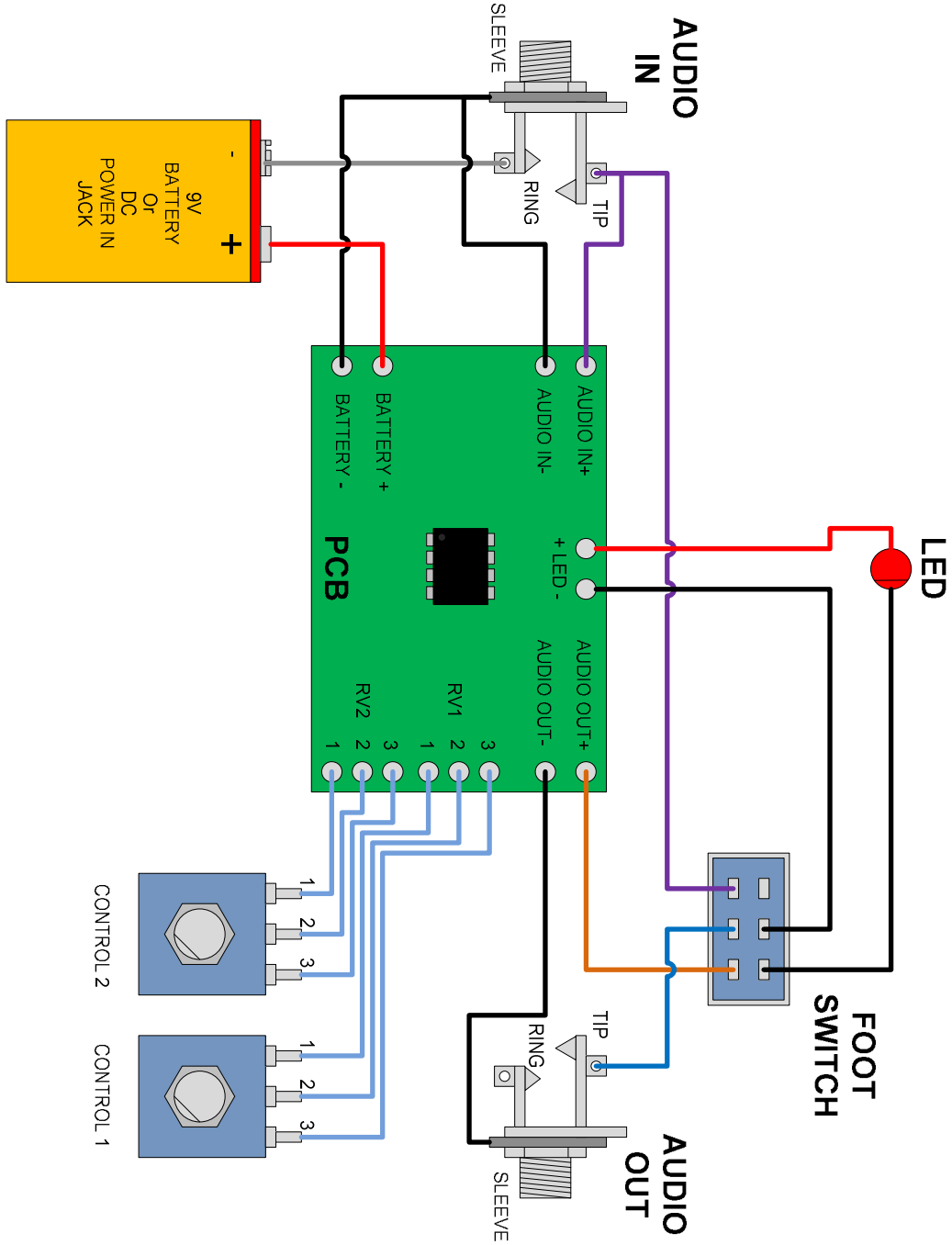
Hardware Assembly Instructions

Equipment list as in PCB Assembly Instructions.

Method;

1) Determine the wires you will require to connect up panel mount components (Potentiometers, Switches, Sockets, and LEDs. Use ONLY multi-core wire, single core will break when bent!
2) For each connection, prepare the wire using the following method;
   a) Determine the wire length and cut to that length. Remember to leave enough length for the PCB to flopped out of the case and for access, too short and it will be difficult to fix should there be any problems.
   b) For each end of the wire, use Wire-strippers to remove the insulation, exposing about 5mm on each end. Twist the conductors together to bind them together.
   c) Using the Soldering Iron, "Tin" the wire at the non-PCB end only. Instructions on how to "Tin" wire can be found online.
   d) Determine the correct position for the wire (P1, P2, etc.) by referring to the Wiring and/or Schematic diagram and PCB Overlay diagram.
   e) At the non-tinned end of the wire, insert into the PCB, then Solder in place.
   f) Move to the next Wire
3) Prepare the Panel Mount components (e.g. Audio sockets, Potentiometers, Foot-switch).
4) If a Panel Mount component is inserted into the case from the inside, you do not need to mount it to the case before soldering, otherwise, you must mount the component first.
5) CHECK THE ABOVE. It is a very common mistake to solder a Panel-mount component wires and then realize its wires must go through the front panel hole before assembly! Fixing this will be messy!
6) Solder each PCB wire in accordance with the wiring diagram, taking extra care to ensure it is the right wire.
7) Once all Panel-mount parts are soldered and mounted you may tighten the securing nut or screw.
8) For all Panel mount parts with a washer and nut, mount appropriately, using the Bottle-nose pliers (on both sides if you have two pairs) to tighten the nut. Do NOT use a spanner, too much force will break the nut or component.
9) Finally with all wiring assembled, insert the IC into the IC socket ensuring the orientation is correct.
10) It is recommended that you test the circuit at this point. Before powering up, double-check everything then refer to any Test page (if applicable) in this manual.
11) Once testing is complete, prepare the PCB for mounting in the enclosure.
12) In a small enclosure, the PCB may be wrapped in an insulating material (durable card, paper or foam) and simply placed in the bottom box.
13) For larger enclosures, a PCB mounting method is recommended such as a plastic clip or stand-off.
14) **Ensure that the PCB is INSULATED from any metal case!** If the solder joints become electrically connected via touching a metal case, the circuit will likely be instantly destroyed on power up.
15) Add any customizations you wish. For decals, painting or transfers (such as labeling panel controls) these should be done prior to assembly.
16) The pedal is now complete.

Fig 5: General Wiring Diagram (May vary depending on Model)



Please Check Schematic and PCB connections Table for Connections specific to relevant Model.

Note: This configuration uses the Sleeve of the Audio In jack to Switch the -Ve (Negative Power Connection) of the Battery. This can be modified as required (use a switch, or switched via DC power jack etc).

Trouble Shooting

General Guide;

1) Problems with Audio circuits can usually be solved by using an Oscilloscope and tracing through nodes at each step of the signal path from the input. E.g. Find the point where the signal enters a component, then move to the next, and check the signal is as expected at each point in the circuit to diagnose where the problem begins from Input to Output.
2) To aid diagnosis of the problem, it may be useful to download the manufacturer datasheet of the part (e.g. IC) that is used in the circuit. This will help determine that it is inserted the correct way around, and what pins perform which function of the part.
3) It is important that you pay attention to every single detail when working with electronics. If you are measuring using test equipment, ensure you have set the device to the correct range, and you understand exactly what the reading means (e.g. k, M, pF, mV, etc.)
4) You cannot measure Resistances or Capacitance when a component is soldered into a circuit since this will be affected by other components connected to it. In this case it may be necessary to check the color coding or numbers printed on the package to confirm it is the correct value and part type.
5) Check the Solder side of the PCB for any "dry joints" (not a proper connection), or shorts - where a stray piece of solder or lead is connecting two points on the PCB that shouldn't be connected.
6) Avoid removing components from the PCB unless absolutely essential, this can damage the tracks and destroy the PCB. If possible, find ways to avoid having to remove the leads of through-hole components, and if it is absolutely necessary to remove a part, take great care not to overheat the connection point (Pad) on the PCB since the track will lift from the board if over-heated.
7) No output can mean a lot of things, from a damaged IC (that could have been damaged by ESD (Electro-static Discharge) while handling, to simply not having the volume turned up.
8) Take great care and focus to slowly eliminate problems to attempt to find the source. E.g. check each wire and component is correctly placed/connected, as per the instructions - although tedious, this is the first thing that must be checked.
9) If using an adjustable output Current power supply, it is a good idea to set the current limit to the highest you can expect from the circuit prior to powering up (e.g. 30mA MAX for Stomp Box kits). This will prevent a high current (in the case of an incorrectly inserted component or short -circuit) destroying the components on your board if there is a problem (due to the current (power) limit.
10) The two inputs (+V and -V) to an operational amplifier should be exactly same in terms of voltage, if they are not, your op-amp is damaged.
11) Check supply voltages to IC's exist, and are at the correct voltage on the VCC (+V, VCC, VDD) and GND (-V, VEE, VSS) pins. Take care not to short out the legs of an IC when probing it live.
12) For NPN silicon transistors, you should never see a voltage difference of greater than 0.7V across the Base and Emitter, if this is the case, the transistor is damaged.
13) You can work out the current through any Resistor by measuring the Voltage across it, then dividing this by the Resistance value in Ohms - see Ohm's Law. E.g. 10V across a 10k Resistor is 0.1mA
14) If a buzzing or humming sound is heard at the output, the Power Supply has poor regulation. You will require a better (and ensure it is DC) power supply adapter, or simply use a Battery instead.

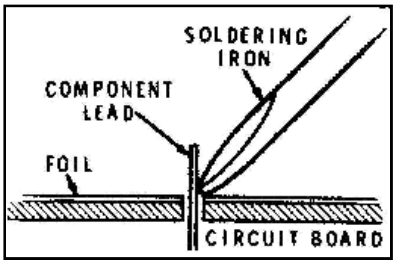
Appendix 1: Soldering Basics

SOLDERING BASICS

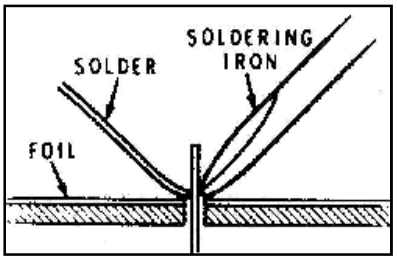
Tools Required:

- **A Soldering Iron**
- **The <u>right</u> solder**
- **A damp sponge**
- **Small sharp wire-cutters**
- **Blue Tack or Science Lab Stand**

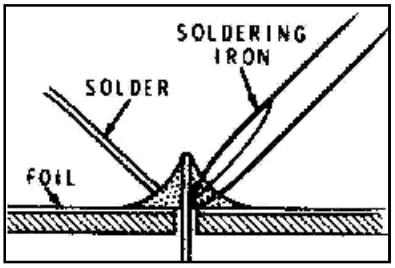
Soldering Method:



1. Place the Soldering Iron tip against both the Lead and the Circuit Board foil and allow to heat for 1-2 Seconds.



2. Then apply the Solder to the other side of the connection. It is important to let the heated lead and Circuit board foil melt the solder NOT the soldering iron.



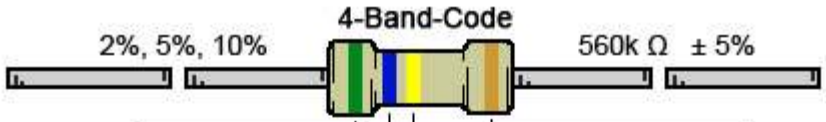
3. As the Solder begins to melt allow it to flow around the connection. Then pull away the Solder first, followed by the Soldering Iron.

4. As the joint cools ensure there is no movement that could create a dry joint. Finally snip the Lead flush being careful of any flying debris.

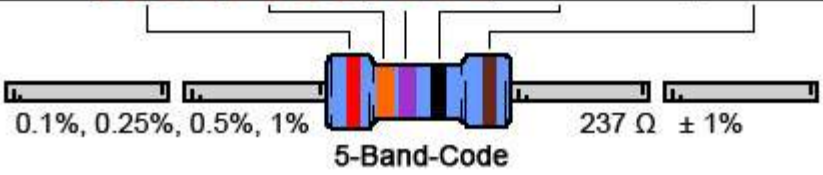
Soldering Key Points:

1. **All parts must be clean and free from dirt and grease.**
2. **Try to secure the work firmly.**
3. **"Tin" the iron tip with a small amount of solder once heated.**
4. **Clean the tip of the hot soldering iron on a damp sponge.**
5. **Heat all parts of the joint with the iron for under a second or so.**
6. **Continue heating, and then apply sufficient solder only, to form an adequate joint.**
7. **Remove and return the iron safely to its stand.**
8. **It only takes two or three seconds at most, to solder the average PCB joint (larger areas will take longer).**
9. **<u>DO NOT</u> move parts until the solder has cooled.**

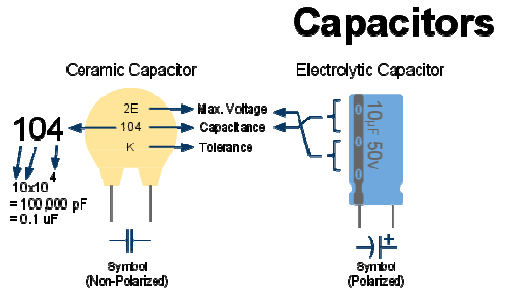
Appendix 2: Resistor Colour Codes



COLOR	1ST BAND	2ND BAND	3RD BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	± 0.5% (D)
Blue	6	6	6	1MΩ	± 0.25% (C)
Violet	7	7	7	10MΩ	± 0.10% (B)
Grey	8	8	8		± 0.05%
White	9	9	9		
Gold				0.1Ω	± 5% (J)
Silver				0.01Ω	± 10% (K)



Appendix 3: Capacitor Codes



Capacitance Conversion Values

Microfarads (µF)	Nanofarads (nF)	Picofarads (pF)
0.000001 µF	0.001 nF	1 pF
0.00001 µF	0.01 nF	10 pF
0.0001 µF	0.1 nF	100 pF
0.001 µF	1 nF	1,000 pF
0.01 µF	10 nF	10,000 pF
0.1 µF	100 nF	100,000 pF
1 µF	1,000 nF	1,000,000 pF
10 µF	10,000 nF	10,000,000 pF
100 µF	100,000 nF	100,000,000 pF

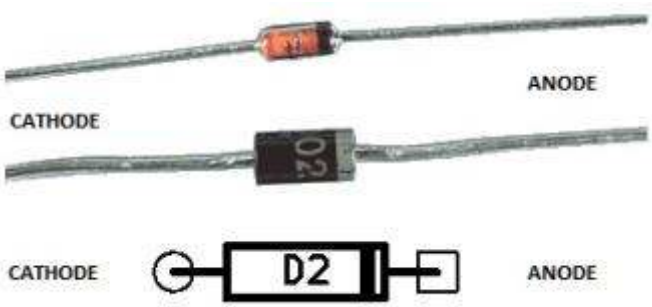
Max. Operating Voltage

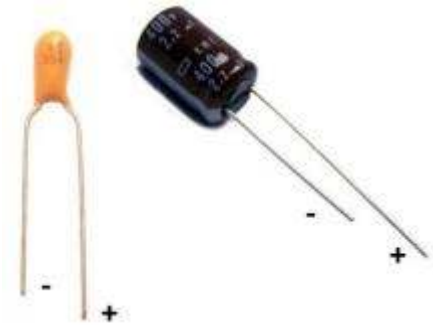
Code	Max. Voltage
1H	50V
2A	100V
2T	150V
2D	200V
2E	250V
2G	400V
2J	630V

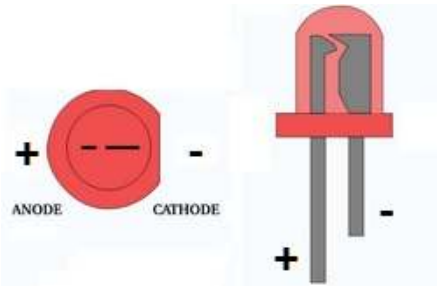
Tolerance

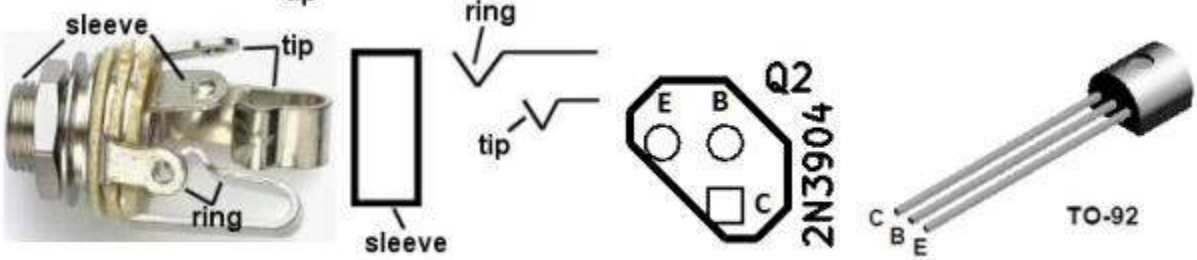
Code	Percentage
B	± 0.1 pF
C	±0.25 pF
D	±0.5 pF
F	±1%
G	±2%
H	±3%
J	±5%
K	±10%
M	±20%
Z	+80%, -20%

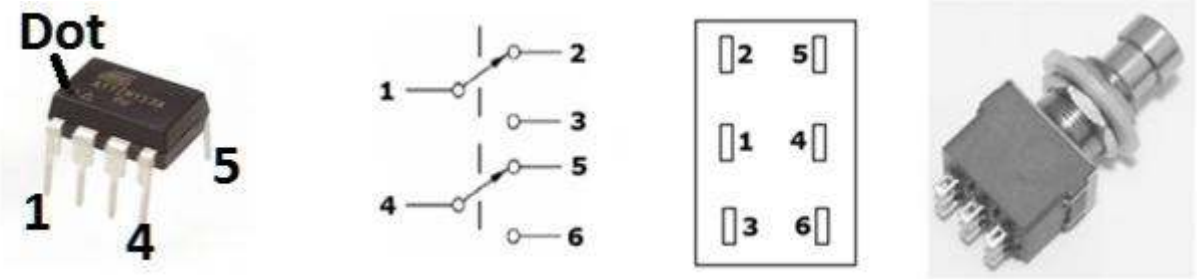
Appendix 3: Component Orientations











Appendix 4: General Assembly Guide

1. Firstly, organize your work area…

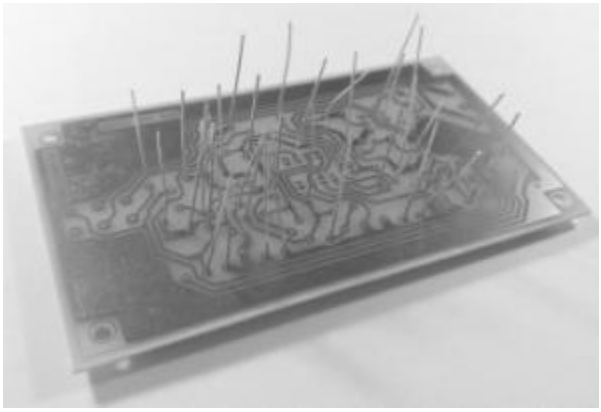
Recommended Tools are;

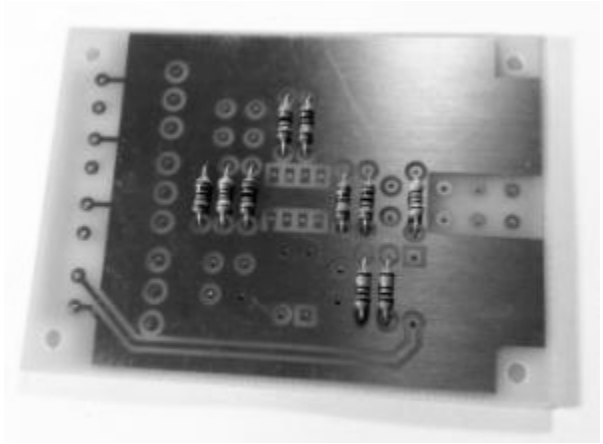
- 2x Bottlenose Pliers
- 1x Wire Cutters
- 1x Soldering Iron, Solder, Sponge, & Stand
- 1x Helping Hands (optional)
- 1x Soldering rework tool (optional)

Use a least a 25W Soldering Iron if possible, and observe handling precautions for Anti-static assembly (Ground yourself before assembly)

2. Place Resistors into Board. Component *bodies* go on the <u>top-side</u> of the Board. Bend the legs of the Components so they fit through the holes.

<u>IMPORTANT:</u> Be sure to check the *orientation* and correct position of Components by referring to the **<u>Component Layout</u>** illustration for Pedal, this is viewed from **<u>above</u>**).

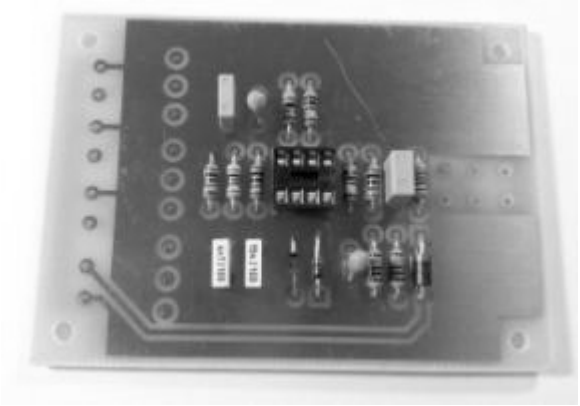




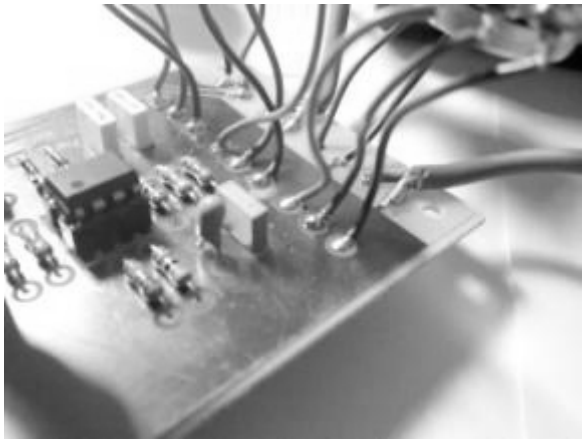
3. Solder in the Resistors and cut the legs with the Cutters. Make sure you use an appropriate soldering technique; clean the Soldering Iron regularly and ensure good contact to the leg and joint simultaneously.

<u>WARNING:</u> When cutting the excess legs, hold the excess part of the leg to avoid it flying off into your face or elsewhere. These can be very sharp!

4. Solder the remaining board components, starting with the lowest in body height first; E.g. Diodes, then the IC Holder, then the Capacitors. Do not insert the IC/Chip until ALL soldering is finished.



The completed board should look like this;



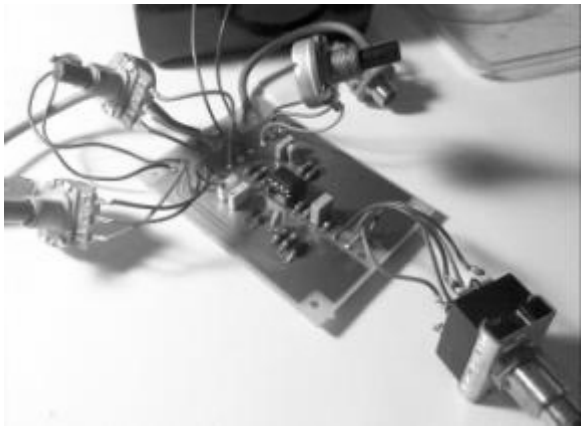
5. Now Solder the Wired Components

These include the Potentiometer Controls, Audio Jacks, Power Jack, and Foot-switch. Use *multi-core* wire, and solder it to **both** the top and bottom of the board as shown in the image.

This is essential to ensure that the top and bottom layers connect and to make a strong connection for the wiring.

6. Use *Shielded Cable* for Audio Connections to minimize noise (Shield = Ground).

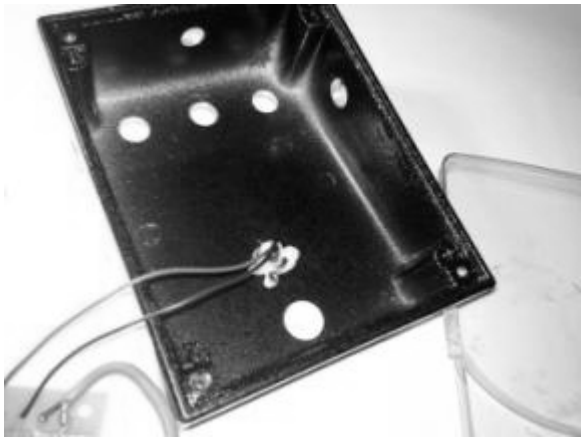
Be sure to connect signal cables correctly. *(+) goes to the Tip or Signal*, *(-) goes to the Ring or Gnd*. Double-check ALL connections before soldering them to avoid having to fix them later.



The board with Wired Components should look like this.

Aim for the cable lengths to be a close as possible to that required for the inside of the case.

IMPORTANT: Insert the IC/Chip only when ALL soldering has been completed.



7. Mount Wired Components

Depending upon your chosen design, fix the components securely into place. In this picture, the LED has been glued into the Pedal's pre-drilled case and connected directly to the board.

It is recommended that appropriate fixing hardware is used to ensure a robust design (e.g. an LED Bezel).

8. Fit all components *loosely*, until you have the basic layout of parts, and cable's tucked away nicely.

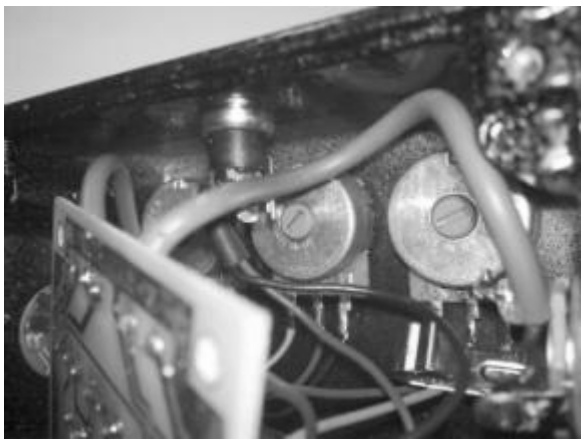
There should be plenty of space in your case, with no chance of exposed connections touching each other, or excessively bent or forced cables.

This image shows the completed hardware loosely mounted in the case.

9. Tighten fixings

All fixing should have the body held straight whilst tightening the nuts (preceded by a washer) with bottlenose pliers.

Some types of Potentiometer have flat side of the shaft enabling two bottlenose pliers to be used as shown in order to hold the body straight.



Mounted components should <u>not</u> be allowed to rotate when tightening.

Potentiometers should have their legs facing towards the bottom of the panel as shown.

<u>Make sure</u> that no exposed connections are touching. If this is likely, use rubber heat-shrink insulation or thick tape to isolate these.

This image illustrates holding and tightening the Audio Jacks from the inside;

10. Final Assembly

If you are placing the PCB directly into a metal case, be sure to *insulate* the PCB with thick card wrapped with tape, or other suitable insulating material in the case.

WARNING: If the PCB touches a metal surface, it will fail and may become damaged or even catch fire!

The image shows a tape over card insulation covering the base of this Pedal Box.

11. Attach Knobs and Aesthetics

When attaching knobs, position the shaft so that it is turned fully anti-clockwise, then attach the Knob (pointing in 7 o'clock direction).

Make sure the Knob isn't rubbing against the base, and tighten as much as possible without damaging to prevent slipping.

12. Completed Pedal

The Pedal is now complete and ready for testing.

Take *extra care* when testing. Do not test using your own expensive equipment until you are sure. Check the power supplies first, and if you have a Multi-meter, check solder joints and cabling for continuity. Always apply a *low-level signal* at first, and use a *low output level.* If the pedal does not work, check all wiring and component orientation, and then trace the signal path through the circuit to find the problem by referring to the schematic.

Online Resources

http://www.guitardistortioneffects.info/

http://www.schematicheaven.com/effects.htm

http://www.diyguitarist.com/DIYStompboxes/MyStompboxes.htm

Disclaimer and Intellectual Property

This kit is provided as is without any guarantees or warranty. In association with the product, EMS Electronics makes no warranties of any kind, either express or implied, including but not limited to warranties of merchantability, fitness for a particular purpose, of title, or of non-infringement of third party rights. Use of the product by a user is at the user's risk.

NO REPRESENTATIONS OR WARRANTIES, EITHER EXPRESS OR IMPLIED, OF MERCHANTABILITY, FITNESS FOR A SPECIFIC PURPOSE, THE PRODUCTS TO WHICH THE INFORMATION MENTIONS MAY BE USED WITHOUT INFRINGING THE INTELLECTUAL PROPERTY RIGHTS OF OTHERS, OR OF ANY OTHER NATURE ARE MADE WITH RESPECT TO INFORMATION OR THE PRODUCT TO WHICH INFORMATION MENTIONS. IN NO CASE SHALL THE INFORMATION BE CONSIDERED A PART OF OUR TERMS AND CONDITIONS OF SALE.

The Intellectual property of all material contained within this document and physical items supplied as "the kit" belong to EMS Electronics. By using this product you are accepting that you will;

1) NOT re-distribute this document - or any elements of it - online or in any other form without prior written permission from EMS Electronics
2) NOT resell this product - either complete or incomplete - for commercial purposes with prior written permission from EMS Electronics

By using this product, you are agreeing to both the Disclaimer and Intellectual property rules defined herein. If you do not agree, the product must not be used. EMS Electronics accept no responsibility for injury, loss or damage incurred by the use of the product.

EMS Electronics is the property and alias of Luke Benjamin Hastilow and any associates.