Electric Druid Hard Bargain Distortion

Overview

The Hard Bargain distortion is a simple-to-build hard distortion pedal that uses an unusual two-knob tone control to provide considerable tonal control over the final sound without huge complexity. The circuit is designed to allow a range from softer overdrive tones to mid-rangey sixties rock sounds and heavy distortion scooped-mids sounds. We hope that amongst this variety, you'll be able to find a tone that you like and that suits your playing.

Build Instructions

You're advised to have a read through of these instructions before starting work on the PCB. To keep these instructions reasonably brief, it is assumed that you know how to orientate common components.

Populate the PCB

The board should be populated in order from smallest components to tallest. The BOM on page 6 is arranged in this order, so start at the top and work your way down. You can tick off each line in the "Done?" column on the far right.

If you hold the PCB with the "hard bargain" text visible and the "electric druid" logo to the left, you'll see that the components are arranged in three rows. The top row is entirely film capacitors and resistors. The middle row is mostly electrolytic capacitors and the three op-amps. The bottom row is more film capacitors and resistors, but also includes the various diodes.

Power protection diode

Start by installing the 1N5817 diode in the bottom-left of the PCB. This protects the PCB against reverse voltage, so be sure to check the orientation carefully.

1N4148 Diodes

Next do the 1N4148 clipping diodes. They're to the right of centre in the bottom row. If you want to experiment with other diodes, leave this step out for now.

Resistors

Next come the resistors. We do them in value order, from the lowest to the highest.

- 47R resistor x 1 - bottom left, next to the 1N5817 you just did
- 560R resistor x 1 - bottom left, next to the 47R you just did
- 1K resistor x 5 - three on the top row, one in the centre, and one in the bottom right
- 1K5 resistor x 2 - above the left-most op-amp
- 2K2 resistor x 1 - next to the three clipping diodes
- 10K resistor x 3 - one on the top row next to the word "hard", one in the centre, and one on the bottom row underneath the right-most op-amp
- 33K resistor x 1 - farthest right on the top row
- 47K resistor x 3 - three together below the centre op-amp
- 100K resistor x 1 - bottom row, left hand end
- 220K resistor x 2 - both together on the top row, left hand end
- 1M resistor x 1 - bottom row below the centre op-amp
- 2M2 resistor x 2 - bottom row, below the right-most op-amp

Cup of tea and soldering check

When you've finished doing the resistors, stop and have a cup of tea and spend a few minutes looking over your solder joints and making sure everything's ok so far.

IC sockets

All three 8-pin DIP sockets for the op-amps are identical. They are on the centre line of the board with the notch pointing left. It helps to solder only a single pin or a couple of corner pins first, and then give the socket a check. If it's sitting correctly and orientated the right way around, you can solder the rest of the pins. If not, it's much easier to adjust it with only two pins soldered. Removing IC sockets from plated-through-hole PCBs like this one is difficult and not recommended.

Ceramic bypass capacitor

There is one 100n ceramic bypass capacitor in the power supply section in the centre row to the left of the left-most op-amp. This is ***not*** the fat 100n film cap in the bottom right corner.

Film capacitors

There are eleven film caps in the Hard Bargain.

- 100p (101, 0.1n) capacitor x 1 - top row, above the left-most op-amp
- 470p (471, 0.47n) capacitor x 2 - bottom row centre, and top row far right
- 2n2 (222, 2200p) capacitor x 1 - bottom row centre
- 3n3 (332, 3300p) capacitor x 1 - top row far left
- 22n (223, 0.022u) capacitor x 2 - both top row centre, either side of the word "hard"
- 33n (333, 0.033u) capacitor x 1 - top row, above left-most op-amp
- 100n (104, 0.1u) capacitor x 1 - bottom row far right corner
- 220n (224, 0.22u) capacitor x 1 - top row, between "hard" and "bargain"
- 470n (474, 0.47u) capacitor x 1 - bottom left, by the "Rev.2" text

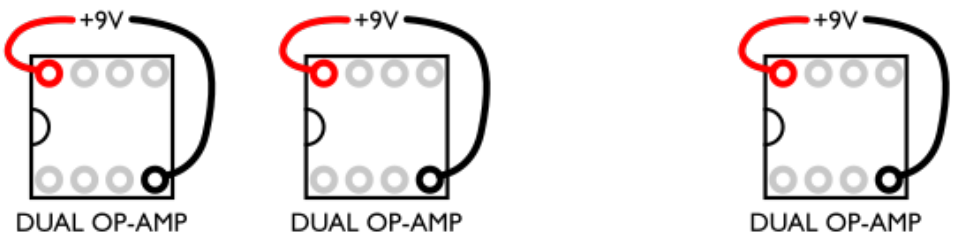
Electrolytic capacitors

There are only three of these, but you need to watch the polarity.

- 10u capacitor x 2 - these are both next to the right-most op-amp.
- 100u capacitor x 1 - centre left, next to the electric druid logo

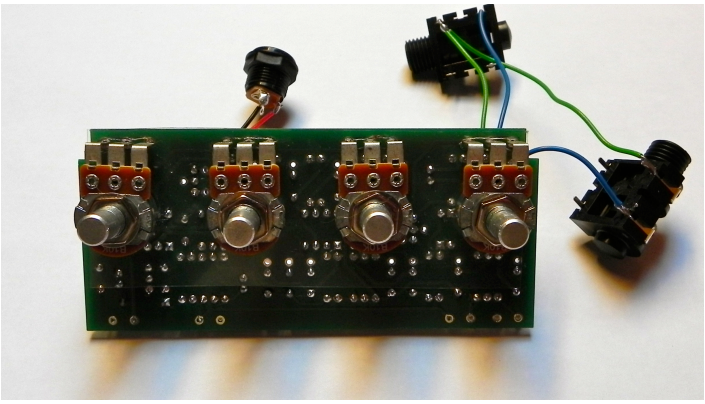
Second cup of tea and Power Test

Have a break. If you've got this far, you deserve it. Also, you need to be on top form for the next part - testing the power. At this stage, you can power the board up and check the voltages with a multimeter. Don't put the chips in yet. There should be 9V power across pins 4 and 8 of each op-amp socket.



Check the soldering over one last time, since after you fit the pots, it's a lot more difficult to get to some of the PCB.

Potentiometers



Note that the pots mount on the back (solder-side) of the PCB!

First, break the small anti-rotation tabs off the pots with pliers.

Something is required to prevent the pots from shorting out the back of the PCB. Many things work; all the way from expensive pot dust covers, to a couple of pieces of insulation tape stuck on the back of the pots, to a piece of cardboard stuck between the board and the pots. My current favourite solution is to cut a piece of stiff overhead transparency plastic and slide it between the PCB and the pots. This can be done after soldering the pots, but pot dust covers would need fitting now.

Install ICs

If the voltage check was ok, you can install the three op-amp chips.

The PCB is done! Well done!

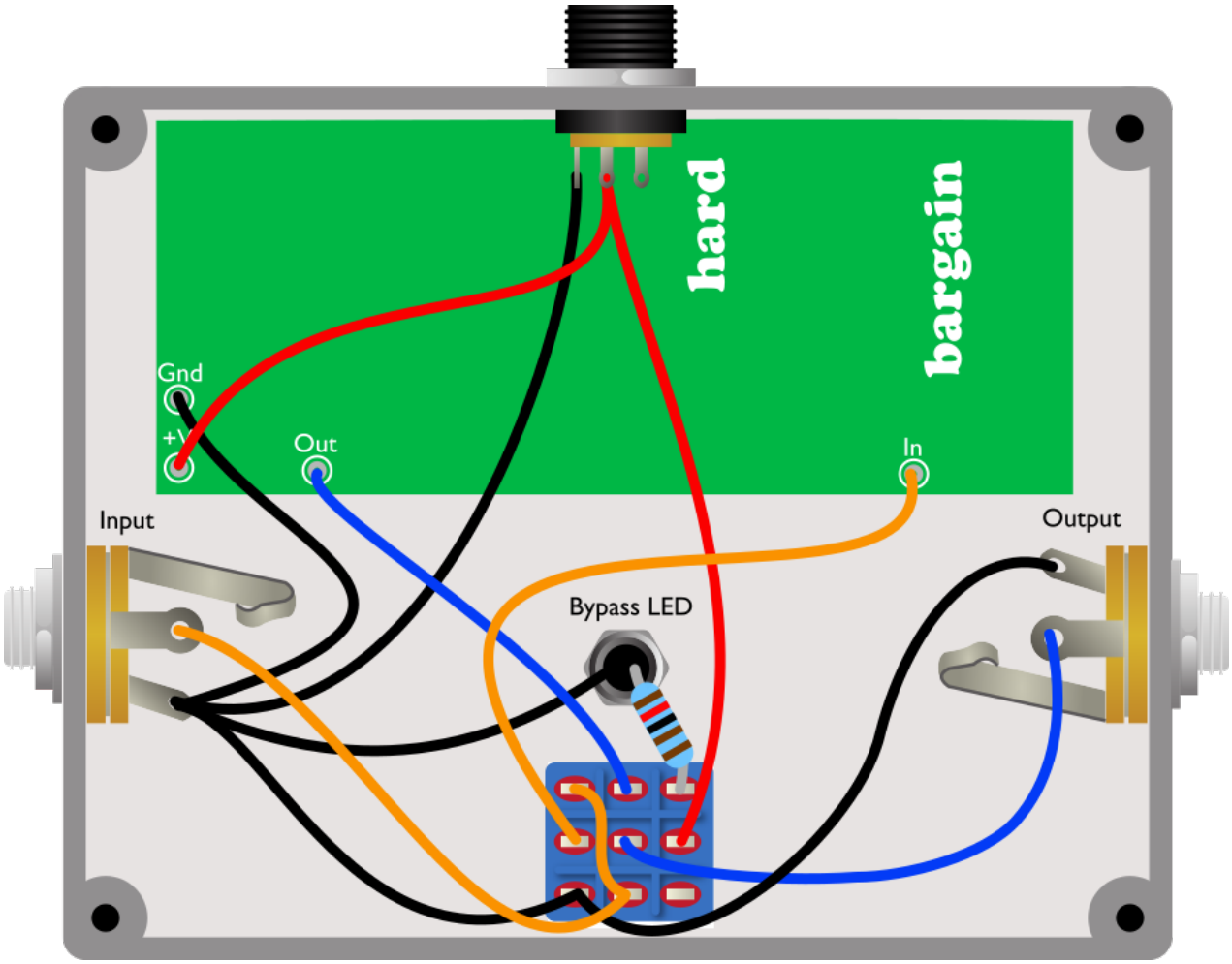
Drilling the enclosure

The PCB is designed to be mounted in landscape format in a Hammond 1590BB enclosure or equivalent. The board is held in place by the four pots.

You can download the Hard Bargain drilling template from the Electric Druid website.

Off-board wiring

The off-board wiring for the Hard Bargain isn't too complex.



Different requirements will need different wiring, and there are many ways to arrange things, but here's a basic 3PDT layout with indicator LED. The LED resistor will need adjusting for your particular LED.

There's a power input, mono 1/4" jacks for the input and output, and either a DPDT stomp switch, or a 3PDT stompswitch if you want an LED to show you the on/off status of the effect. For a DPDT, just ignore the right-most column of connections.

Adjustments and final testing

Ok, it's the moment of truth. Power it up and plug it in. With a bit of care and attention, you should now have a working Hard Bargain distortion pedal! It doesn't need any trimming, but the knobs cover a wide range and you'll need to learn how to adjust things to your personal taste.

You're done! Congratulations and enjoy your new pedal!

PS: We appreciate any corrections, feedback, suggestions, or thoughts you have about this pedal or any other Druid project. Please get in touch through the website. Thanks!

Bill of Materials

Order	Ref	Description	Value	Quantity	Done?
1	D1	Polarity Protection Diode	1N5817	1	
2	D2, D3, D4	Signal Diode	1N4148	3	
3	R1	1% Metal film resistor	47R	1	
4	R24	1% Metal film resistor	560R	1	
5	R4, R9, R10, R14, R17, R22	1% Metal film resistor	1K	6	
6	R18, R19	1% Metal film resistor	1K5	2	
7	R11	1% Metal film resistor	2K2	1	
8	R8, R15, R16	1% Metal film resistor	10K	3	
9	R7	1% Metal film resistor	33K	1	
10	R2, R3, R12	1% Metal film resistor	47K	3	
11	R23	1% Metal film resistor	100K	1	
12	R20, R21	1% Metal film resistor	220K	2	
13	R13	1% Metal film resistor	1M	1	
14	R5, R6	1% Metal film resistor	2M2	2	
15	IC1, IC2, IC3	IC sockets	8-pin DIP	3	
16	C2	Ceramic capacitor	100n	1	
17	C14	Film capacitor	100p	1	
18	C5, C8	Film capacitor	470p	2	
19	C9	Film capacitor	2n2	1	
20	C13	Film capacitor	3n3	1	
21	C10, C11	Film capacitor	22n	2	
22	C12	Film capacitor	33n	1	
23	C4	Film capacitor	100n	1	
24	C6, C15, C16	Film capacitor	220n	3	
25	C3, C7	Electrolytic capacitor	10u	2	
26	C1	Electrolytic capacitor	100u	1	
27	VR1, VR4	Drive and Level pots	100K Log	2	
28	VR2	Tilt Pot	100K Lin	1	
29	VR3	Mid Boost/Cut pot	10K Lin	1	
30	Unmarked	Pot dust covers or plastic			
31	IC1, IC2, IC3	Dual audio op-amp	TL072	3	

Additionally, you will need some/all of the offboard components listed on the next page.

Offboard components

Note that the BOM above doesn't include offboard components. These are a matter of taste, but the basics are listed below.

- Enclosure, PCB fits Hammond 1590BB or Eddystone 29830PSLA
- 2 x Mono 1/4"/6.35mm Input and Output jacks
- Stomp switch, 3PDT for Bypass switching
- Power Input socket, 2.1mm. Sockets with an external nut are much easier, since you can wire them and test the board without it in the enclosure.
- 4 x Knobs
- Effect on/off LED and series resistor for the brightness of your choice

Component choices and substitutions

Very few of the components in the circuit are especially critical and a unit built with non-ideal components will likely still work fine.

Resistors

In the interests of lowest noise, we recommend you use 1% metal film resistors.

Capacitors

Use good quality polypropylene or polyester film capacitors. The board allows either 0.2"/5mm or 0.3"/7.5mm lead spacing for the film capacitors.

Op-amps

Op-amp choice is not critical, but might have a marginal effect on the sound. Choose any 8-pin dual audio op-amp with the standard pinout. TL072, LF353, or MC1458 will all work. You can also use 'magic mojo' op-amps from your favourite snake-oil supplier.

Diodes

The power protection diode suggested is 1N5817. This diode is recommended because of its low voltage drop at the sort of currents the pedal draws. Others will work but may reduce headroom a little more.
The 1N4148 diodes can be replaced with other small signal silicon diodes. 1N914 is a direct replacement and can be considered identical. You can also experiment with other more esoteric clipping options like LEDs or diode-wired MOSFETs.

Ideas for potential upgrades or customizations

Changing the centre frequency

With the parts marked on the PCB, the centre frequency of the Tilt and Mid Boost/Cut controls are both set to 800Hz. It is possible to build the pedal with a 640Hz centre frequency instead by altering the following components. This alters the position of the mid-range scoop/peak and the centre of the Tilt control. Despite the human ear being particularly sensitive in this range, I can best describe the effect as "very subtle"!

- R15 and R16: Change from 10K to 13K
- R18 and R19: Change from 1K5 to 1K2
- C12: Change from 33n to 47n
- C13: Change from 3n3 to 4n7

Different diode choices

The 1N4148 clipping diodes can be replaced with other diodes. Pretty much anything goes here. You can try LEDs, power rectifier diodes, Zener diodes, germanium diodes, diode-wired MOSFETs, whatever you like. Be aware that these different diodes have different clipping levels, so some will clip more signal (=harder distortion) but give a lower output level, whereas others clip at a higher level, which leaves more of the waveform intact (=slightly less distortion) but gives you more output volume. Diodes that provide a higher level output might allow further clipping to occur in the EQ stages that follow when the signal is heavily boosted. Whether this is a fault or a feature is a matter of taste at that point.

Power supply level

The pedal will run fine at 12V or higher. You could try running it on 18V without any changes. The extra headroom will reduce the distortion to some extent, since the op-amp producing most of the gain will no longer clip quite as hard. The overall volume level is still limited by the clipping diodes, so this will not alter, unless you alter the clipping diodes too.