Electric Druid FilterFX Project

Overview

The FilterFX project uses the Druid STOMPLFO to control a 12dB/oct state variable filter. A state variable filter is chosen because this filter design offers highpass, bandpass, and lowpass outputs

which gives us the greatest range of different effects. The LFO offers eight waveforms, including two random waveforms.

Furthermore, the PCB provides for addition of an expression pedal to allow the filter to be used like a wah pedal (but don't expect a replacement for your Clyde McCoy - this is a whole different thing!). The expression pedal can be used as well as the LFO modulation, or with Depth at zero, by itself. There is also a Sync input for the LFO which allows the LFO to be kept in time with an external clock from a sequencer, drum machine, or another LFO.

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 8 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 "FilterFX" title and "electric druid" logo the right way up, you'll see that the board is shaped like a "T". The top bar of the T has the Sync Input circuit, the vactrols, and the StompLFO chip. Below them in the middle of the T, we have the filter op-amp and capacitors. Next we have the toggle switches, with resistors between them, including the three resonance selection resistors which are inside a white box. Finally, at the foot of the board is the input/output op-amp and its associated resistors and capacitors.

Power protection diode

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

1N4148 Diode

Next do the 1N4148 diode. This protects the Sync input, and needs to be the right way around, so again check the orientation. It's the furthest left component on the PCB.

Resistors

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

- 47R resistor x 1 - top left, next to the 1N5817 you just did.
- 220R resistor x 1 - on the bar of the T, top far right
- 470R resistor x 3 - two in the centre between the vactrols, one far right between the switches
- 560R resistor x 1 - between the switches
- 1K resistor x 2 - both left of the lowest TL072 op-amp
- 2K2 resistor x 1 - on the bar of the T, top far right
- 4K7 resistor x 2 - both together at the edge of the board, centre-left
- 10K resistor x 6 - three together to the right of the lowest TL072 op-amp, one in the Sync Input circuit top left, one left of the STOMPLFO chip, one in the resonance resistors box between the switches.
- 20K resistor x 1 - between the switches, centred
- 100K resistor x 2 - between the switches, one is the resonance resistors box, one not
- 120K resistor x 2 - one below each vactrol
- 680K resistor x 1 - between the switches, in the resonance resistors box
- 2M2 resistor x 2 - one to the left of, and one above the lowest TL072

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 are identical. The two for the op-amps are on the centre line of the board with the notch pointed down, and the third one for the StompLFO is in the top-right corner of the PCB 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.

Regulator

The 78L05 +5V regulator REG1 is in the top left of the centre of the board, just below the electric druid logo. Be sure to line up the flat side and the curved side with the markings on the PCB. Don't mix it up with the similar-looking transistors.

Transistors

There are two 2N3904 transistors, both along the top of the board. TR1 is to the right of the right-most vactrol. TR2 is in the top-left in the Sync Input circuit. Both are marked with an "n" (for "NPN"). Again, be sure to line up the flat side and the curved side with the markings on the PCB.

Ceramic bypass capacitors

There are three 100n ceramic bypass capacitors, all at the top of the board. Two are beside the STOMPLFO chip, and one more one is by the 5V regulator. These are **not** the fat 100n film cap towards the bottom left. Don't mix them up! (although it'll still work if you do).

Film capacitors

There are four film caps in the FilterFX. You can use capacitors with either 0.2"/5mm or 0.3"/7.5mm lead spacing. You **can't** use capacitors with 0.1"/2.5mm lead spacing - those two holes are connected together!

- 10n (103, 0.01u) capacitor x 2 - centre left and centre right, either side of the TL072
- 100n (104, 0.1u) capacitor x 1 - bottom left
- 470n (474, 0.47u) capacitor x 1 - bottom right, by the smiley face :)

Electrolytic capacitors

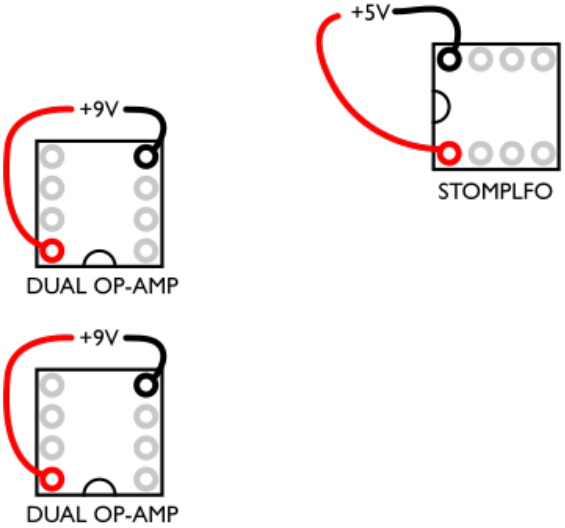
There are only three of these, but you need to watch the polarity.

- 47u capacitor x 2 - these are next to the two vactrols.
- 100u capacitor - top left, just left of the electric druid logo

Vactrols

There are two Xvive VTL5C3 optocouplers ("vactrols" or "optos"). They're the massive things in the middle and they're clearly marked on the silkscreen, so you can't miss 'em! Make sure you get them the right way around. The end of the vactrol marked LED has a + symbol, and the legs are close together on a 0.1"/2.5mm spacing. The end of the vactrol marked LDR or CELL has a wider spacing of 0.2"/5mm.

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. There should be 5V power across pins 8 and 1 of the STOMPLFO chip 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.

Ok, now we take a detour…the reason why will become apparent shortly.

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 pots and switches.

You can download the FilterFX drilling template from the Electric Druid website.

Note that the power input jack is a very tight fit if placed above the PCB as shown on the drilling template. It's possible to do, but the hole needs to be as close to the lid of the box as you dare.

If you don't fancy doing it all, for now you only need to drill the front panel holes for the pots, the switches, and the LEDs.

Potentiometers, Switches, and LEDs

Note that the pots, switches, and LEDs 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.

The best way to fit the pots, switches, and LEDs is to have the enclosure ready and drilled. The components can then be fitted into the PCB (but not soldered) and then the board fitted into the enclosure. Check the flat side of the LEDs matches the flat side on the silkscreen print. Once the LEDs are in the board, bend their legs outwards a little to stop them falling out of the PCB while you fit it. It's a good idea to tighten the pot and switch nuts down a bit to ensure they are properly located in the enclosure. Once you're sure everything's in place, it's a simple job to solder the connections, and you're safe in knowledge that when you next try to fit it everything will be be perfectly lined up with your enclosure.

The two switches have a groove in the threads which can be matched with the notch shown on the silkscreen. In theory the orientation of the switches shouldn't matter, but I admit I haven't dared to try doing it the wrong way around yet since desoldering the switches if it doesn't work would be very difficult!

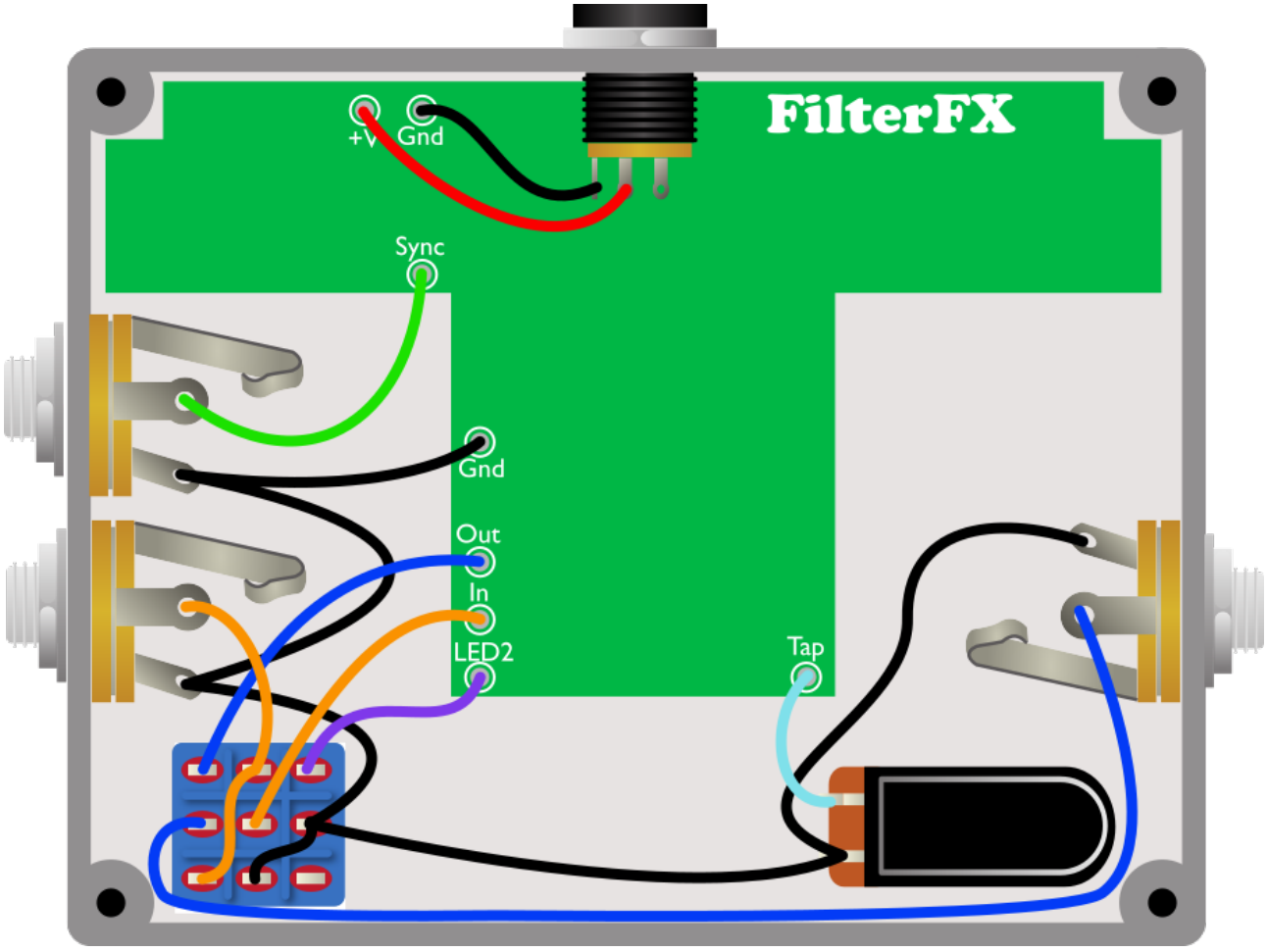
Install ICs

If the voltage check was ok, you can install the three chips; two dual op-amps, and the Electric Druid STOMPLFO chip.

The PCB is done! Well done!

Off-board wiring

The off-board wiring for the FilterFX is fairly complex - take your time and break it up into sections. In fact, you don't even have to do it all if you don't want to, but we'll come to that in a moment.

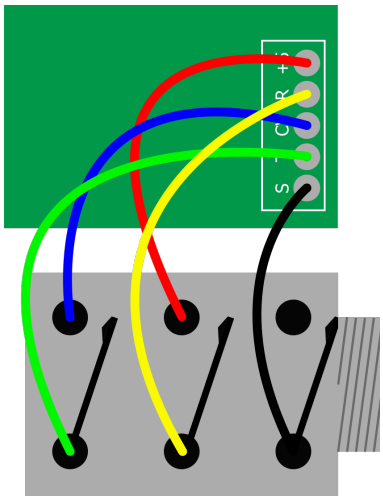


There's a power input and mono 1/4" jacks for the input, the output, and the sync input. The two switches are a 3PDT stomp switch to provide true bypass switching and control the bypass LED to show you the on/off status of the effect and a SPST momentary foot switch for the tap tempo.

Optional Sync Input

The Sync Input is optional. If you include it, it allows you to synchronise the LFO to incoming pulses from a sequencer, analog synth, or drum machine. The Sync input jack is shown on the diagram above, but if you don't need it, you can just ignore it and wire the ground direct from the input jack to the PCB.

Optional Expression pedal / CV input



The optional Expression/CV input allows you to connect an expression pedal or CV source to replace the Frequency control of the filter. A stereo/TRS jack with normally closed contacts should be used. The wiring required is shown on the left. Note the T,R,S pads connect to Tip, Ring, and Sleeve connections on the jack. +5 and CV provide the normally connections for Ring and Tip respectively.

If you don't want the Expression/CV input, there are two jumpers required, shown on the right. They connect the pads marked R/+5 and T/CV.

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 FilterFX pedal! It doesn't need any trimming, but the knobs cover a wide range and you'll need to learn how to adjust things for best effect and 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	Signal Diode	1N4148	1	
3	R23	1% Metal film resistor	47R	1	
4	R24	1% Metal film resistor	220R	1	
5	R16, R17, R18	1% Metal film resistor	470R	3	
6	R14	1% Metal film resistor	560R	1	
7	R1, R15	1% Metal film resistor	1K	2	
8	R25	1% Metal film resistor	2K2	1	
9	R20, R21	1% Metal film resistor	4K7	2	
10	R4, R6, R9, R10, R19, R22	1% Metal film resistor	10K	6	
11	R5	1% Metal film resistor	20K	1	
12	R7, R13	1% Metal film resistor	100K	2	
13	R11, R12	1% Metal film resistor	120K	2	
14	R8	1% Metal film resistor	680K	1	
15	R2, R3	1% Metal film resistor	2M2	2	
16	TL072, TL072, STOMPLFO	IC sockets	8-pin DIP	3	
17	REG1	+5V Regulator	78L05	1	
18	TR1, TR2	NPN Transistor	2N3904	2	
19	C7, C9, C10	Ceramic capacitor	100n	3	
20	C1	Film capacitor	100n	1	
21	C2, C3	Film capacitor	10n	2	
22	C4	Film capacitor	470n	1	
23	C5, C8	Electrolytic capacitor	47u	2	
24	C6	Electrolytic capacitor	100u	1	
25	V1, V2	VTL5C3 Optocoupler	VTL5C3	2	
26	VR1, VR2, VR3, VR4	10K Lin Potentiometers	10K Lin	4	
27	SW1, SW2	Salecom 3-way switch	T812	2	
28	LED1, LED2	Indicator LEDs		2	
28	Unmarked	Pot dust covers or plastic			
29	IC1, IC2	Dual audio op-amp	TL072	2	
30	uP1	PIC 16F18313	STOMPLFO	1	

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
- Mono 1/4"/6.35mm Input jack
- Mono 1/4"/6.35mm Output jack
- Optional stereo (TRS) 1/4"/6.35mm Expression pedal jack - must have switched contacts to allow normal operation.
- Optional Sync input jack - The PCB allows space for a 1/4"/6.35mm mono jack, but you may prefer a 3.5mm jack to allow easy synchronisation with a Eurorack synth, for example.
- Stomp switch, 3PDT for Bypass switching
- Stomp switch, SPST momentary, for Tap Tempo
- 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

Because of the current drain of this pedal, we don't recommend using batteries instead of a power adaptor. Ok, *maybe*, if you've got 9V rechargeables.

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.

Changing the LED resistors

Depending what LEDs you use, you may find the LEDs are too bright or not bright enough for your taste. The Bypass LED is powered from the 9V supply with a 1K series resistor (R15, bottom left below the 2M2). The LFO LED is powered from 5V with a 470R resistor (R16, furthest right of the resistors between the switches).

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. The two integrator capacitors C2 and C3 are especially important, since they're crucial to the filter's operation. The circuit is unlikely to be able to achieve self-oscillation at high resonance settings because of the differences in the optos, but close matching between the filter caps will help increase the possibility.

Transistors

Transistor choice is not critical. Any medium gain, low noise NPN device will work. The board expects a transistor with the EBC pinout, like the 2N3904. If you only have transistors with the alternative CBE pinout like the BC547, you can fit the transistor back-to-front.

Op-amps

Similarly, op-amp choice is not critical. Choose any 8-pin dual audio op-amp with the standard pinout. TL072, LF353, or MC1458 will all work. Many more audiophile options are also possible!

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 diode can be replaced with other small signal silicon diodes. 1N914 is a direct replacement and can be considered identical.

Vactrols/optocouplers

The VTL5C3 vactrol optocouplers can be replaced with either other types of vactrol (which might change response times and other behaviour - which could be good or bad, depending on your objectives) or could be replaced with home-made opto's made from and LDR and an LED stuck together and wrapped with something to keep the light out (heatshrink or even electrical insulation tape). R11 and R12 (both 120K) limit the dark resistance of the opto's to something reasonable, and you may find with other options that you need to change their values.

Ideas for potential upgrades or customizations

Adding CV inputs

Since the Electric Druid STOMPLFO chip operates using 0-5V control voltages like many other Druid chips, it is possible to add CV control of the Offset(Frequency), Rate, Depth, or Waveform controls.

Adding further expression pedal inputs

The circuit includes an expression pedal input to replace the filter's Frequency control, but a similar scheme can be used for the other controls if required.
A normally-closed stereo/TRS jack socket should be wired in, so the front panel control can be used when an expression pedal is not inserted.

The typical expression pedal wiring is for the Sleeve to be grounded, the Ring to carry the reference voltage, and the CV return on the tip. The diagram shows how this wiring relates to the PCB, the jack, and the pot.
However, this is not the only possible wiring, and your expression pedal may not be this way!