



# Electric Druid Flangelicious Flanger Project

(Using either 4KNOBFLANGE or MULTIFLANGE chips)

<b>Overview</b>	<b>2</b>
<b>Build Instructions</b>	<b>2</b>
<b>Populate the PCB</b>	<b>2</b>
<i>1N4148 Diodes</i>	<i>2</i>
<i>Resistors</i>	<i>2</i>
<i>Cup of tea and soldering check</i>	<i>3</i>
<i>Power protection diode</i>	<i>3</i>
<i>IC sockets</i>	<i>3</i>
<i>Regulator</i>	<i>3</i>
<i>Transistor</i>	<i>3</i>
<i>Ceramic bypass capacitors</i>	<i>3</i>
<i>Film capacitors</i>	<i>3</i>
<i>Electrolytic capacitors</i>	<i>4</i>
<i>Preset resistors (Trimmers)</i>	<i>4</i>
<i>Second cup of tea and Power Test</i>	<i>4</i>
<i>Potentiometers</i>	<i>4</i>
<i>Install ICs</i>	<i>5</i>
<b>Off-board wiring</b>	<b>6</b>
<b>Adjustments and final testing</b>	<b>6</b>
<b>Bill of Materials</b>	<b>8</b>
<b>Offboard components</b>	<b>9</b>
<b>Component choices and substitutions</b>	<b>9</b>
<b>Ideas for potential upgrades or customizations</b>	<b>10</b>
<b>Adding CV inputs</b>	<b>10</b>
<b>Adding Expression pedal inputs</b>	<b>10</b>
<b>Adding more switches and options</b>	<b>10</b>

## Overview



The Flangelicious flanger project comes in two flavours, depending on which FLANGE chip you fit - either the 4 KNOB FLANGE or the MULTIFLANGE. Additional tonal variation is possible by building the pedal with either a 256-stage or a 1024-stage BBD. All four options use the same PCB and components.

The 4 KNOB FLANGE is a fairly standard four knob flanger, as the name suggests. It offers Manual (Flanger frequency), Depth, Rate, and Resonance (Feedback) controls. You can get a wide range of classic flanging sounds from this pedal, from deep slow whooshes to rippling wobbles. With the Depth at zero, you can use the Manual control to set 'static flanger' sounds.



Serving Suggestion

The MULTIFLANGE is a more experimental design. It offers Rate, Depth, Waveform, and Resonance controls. In place of the the sinewave LFO of the 4 Knob Flanger, this offers 7 different waveforms and a 'static' position, where the flanger follows the Rate knob. It can also switch between top-down or bottom-up flanging.

## 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 "flangelicious" and "electric druid" logos the right way up, you'll see that the components are arranged in three rows. The top row is passive components. The centre row is for the ICs, with a few other components. The bottom row is more passive components. Underneath the bottom row are the connections for off-board components.

### 1N4148 Diodes

Start with the two 1N4148 diodes. These need to be the right way around and point in opposite directions, so be careful. They're on the left in the centre row.

### Resistors

Next come the resistors.

- 1K resistor x 1 - bottom row, far left
- 1M resistor x 1 - top row, far left

- 10K resistor x 6 - two on the top row, four on the bottom row
- 100K resistor x 9 - six on the top row, three on the bottom row
- 47K resistor x 7 - four on the top row, three on the bottom row
- 24K resistor x 2 - one on the top row, one on the bottom row, both in the centre
- 12K resistor x 1 - bottom row, on the right
- 3K3 resistor x 1 - bottom row, on the right
- 560R resistor x 1 - bottom row, far left

## 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.

## Power protection diode

Install the fat black 1N4002 diode in the bottom right corner of the PCB. This protects the PCB against reverse voltage, so be sure to check the orientation carefully.

## IC sockets

All four 8-pin DIP sockets are arranged the same way around down the centre of the PCB. It helps to solder only 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 bottom right corner next to the power protection diode. 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 transistor.

## Transistor

The 2N3904 transistor TR1 is in the centre row, far left. 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 power supply bypass capacitors, one beside the FLANGE chip, and two more one either side of the 5V regulator you just fitted. These are **not** the 100n film caps on the bottom row far left. Don't mix them up!

## Film capacitors

There are quite a few of these. Take your time.

- 100n (104, 0.1u) capacitor x 2 - bottom row, far left
- 1n (102, 0.001u) capacitor x 1 - bottom row, centre left
- 150p (150) capacitor x 2 - one on the top row, one on the bottom, both centre left
- 33n (333, 0.033u) capacitor x 1 - top row, far left

- 470p (470) capacitor x 2 - one on the top row, one on the bottom, both in the centre
- 3n3 (332, 0.0033u) capacitor x 2 - one on the top row, one on the bottom, both in the centre
- 470n (474, 0.47u) capacitor x 3 - one in the centre row, far left, the other two on the top row and bottom row, both right of centre.

## Electrolytic capacitors

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

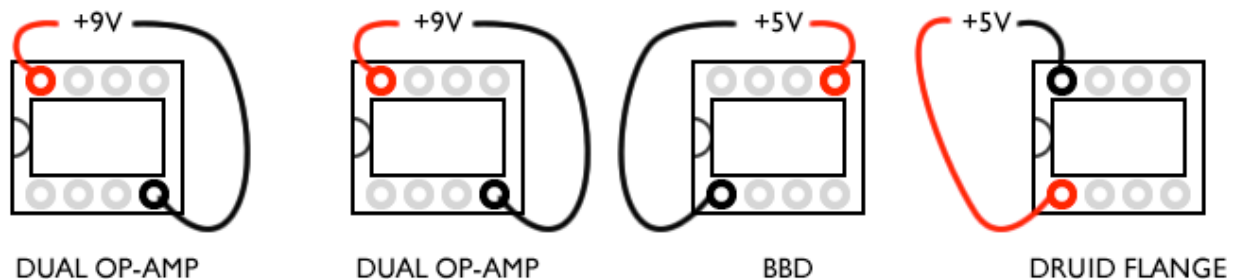
- 47u capacitor - bottom row, far right
- 47u capacitor - top row, left
- 1u capacitor - bottom row, right of centre

## Preset resistors (Trimmers)

There are two trimmers, the 100K Resonance trimmer (marked "100K Res") and the 10K clock bleedthrough trimmer (marked "10K Bal"). The resonance trim is in the centre row, to the left of the op-amps. The clock balance trimmer is top-right, next to the electric druid logo.

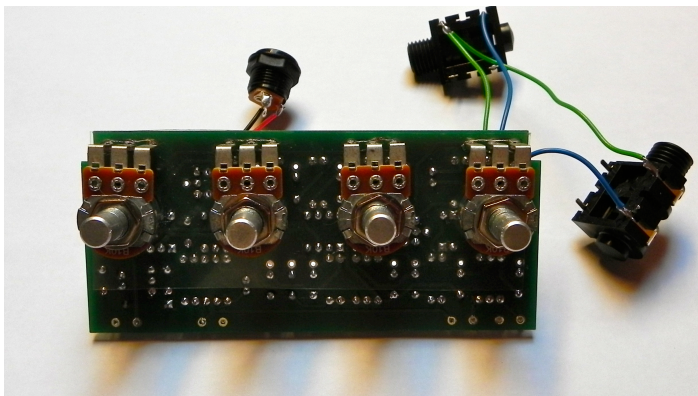
## Second cup of tea and Power Test

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. There should be 9V power across pins 4 and 8 of each op-amp socket. There should be 5V power across pins 1 and pin 5 of the BBD socket and between pins 8 and 1 of the FLANGE 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.

## 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 fit it between the PCB and the pots. If you make holes in it for the legs of the pots to pass through, they hold it in place once soldered and it can't fall out. This is cheap (steal the plastic from work), simple (get a grown-up to help you with the scissors), and effective (none have blown up yet).

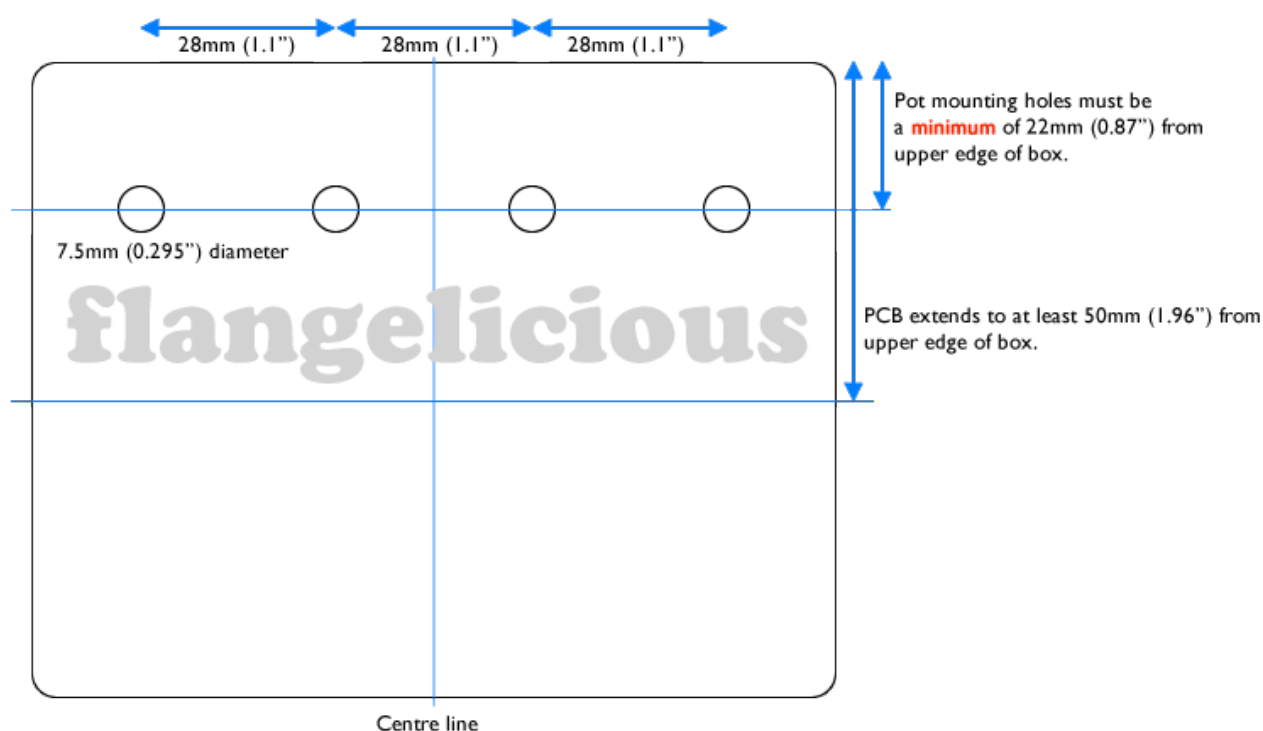
## Install ICs

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

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 pots.

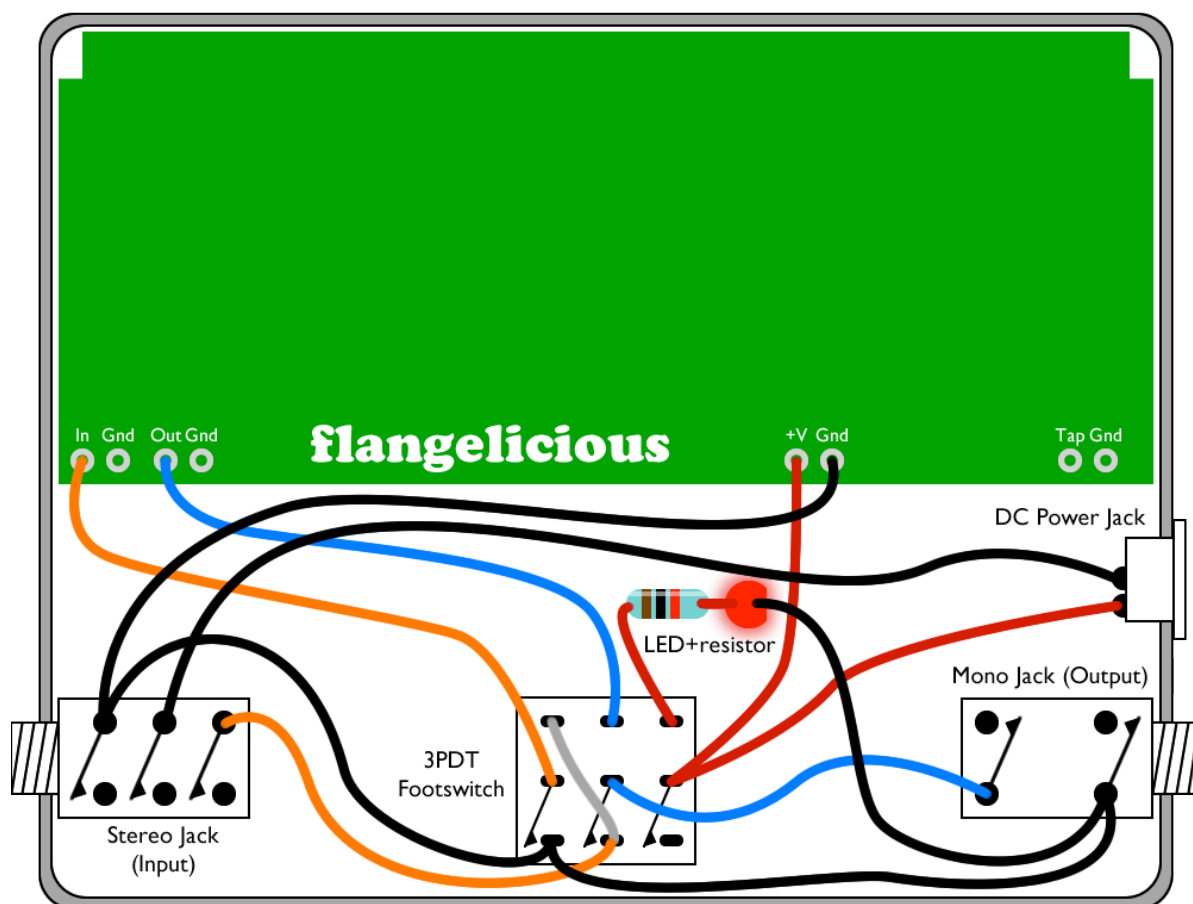


## Off-board wiring

This is the same for all pedals, pretty much.

There's a power input, a stereo 1/4" jack which serves as the input and powers the pedal up when something is inserted, a mono 1/4" jack for the output, and either a DPDT stomp switch, or a 3PDT stomp switch if you want an LED to show you the on/off status of the effect.

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.



## 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 flangerlicious pedal! Let's get it tuned up.

There are two trimmers on the PCB that need adjusting. The first is the resonance trim ("Res"). Turn the resonance knob on the far right up to maximum. It's possible that the pedal will start to howl at you, but it's more likely that nothing much will happen. Tweak the resonance trim until the flanger starts to oscillate, and then back it off just a little. Or not, if you prefer. Up to you.

The other trimmer is the clock balance / bleedthrough trim ("Bal").



If you have an oscilloscope (or a friend who does!) you can put the 'scope probe on the test point (marked "TPI" in the centre of the board) and adjust the trim for minimum clock bleedthrough. If you don't have an oscilloscope, you can do the same with a multimeter on a AC voltage range - you should be aiming for 100mV or less.

You can also adjust the trim by ear. With no input signal (turn the guitar down to zero) turn the amp up loud until you can hear the background noise, and then adjust the trim to minimize the noise.

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

*PS: We appreciate any 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, D2	Signal Diode	1N4148	2	
2	R1	1% Metal film resistor	1K	1	
3	R2	1% Metal film resistor	1M	1	
4	R3, R10, R12, R17, R28, R29	1% Metal film resistor	10K	6	
5	R4, R6, R13, R14, R15, R19, R22, R23, R26	1% Metal film resistor	100K	9	
6	R5, R7, R8, R16, R21, R24, R25	1% Metal film resistor	47K	7	
7	R9, R11	1% Metal film resistor	24K	2	
8	R18	1% Metal film resistor	12K	1	
9	R20	1% Metal film resistor	3K3	1	
10	R27	1% Metal film resistor	560R	1	
11	D3	Rectifier Diode	1N4002	1	
12	TL072, TL072, BBD, FLANGE	IC sockets	8-pin DIP	4	
13	REG1	+5V Regulator	78L05	1	
14	TR1	NPN Transistor	2N3904	1	
15	C14, C17, C19	Ceramic capacitor	100n	3	
16	C1, C2	Film capacitor	100n	2	
17	C3	Film capacitor	1n	1	
18	C4, C10	Film capacitor	150p	2	
19	C5	Film capacitor	33n	1	
20	C6, C8	Film capacitor	470p	2	
21	C7, C9	Film capacitor	3n3	2	
22	C11, C12, C15	Film capacitor	470n	3	
23	C13	Electrolytic capacitor	1u	1	
24	C16, C18	Electrolytic capacitor	47u	2	
25	PR1	Preset (Resonance Trim)	100K	1	
26	PR2	Preset (Balance Trim)	10K	1	
27	VR1, VR2, VR3, VR4	16mm Pots	10K Linear	4	
28	Unmarked	Pot dust covers or plastic			
29	IC2, IC3	Dual audio op-amp	TL072	2	
30	IC1	Bucket Brigade Delay	MN3209/07	1	
31	uP1	PIC 12F1501	FLANGE	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
- Stereo 1/4"/6.35mm Input jack
- Mono 1/4"/6.35mm Output jack
- Stomp switch, DPDT or 3PDT if you want an LED indicator
- Power Input socket, 2.1mm
- LED + LED holder + appropriate resistor
- 4 x Knobs
- Another switch for the 'Tap' option, if required (See "Adding more switches and options" on Page 10).

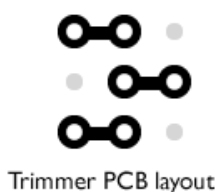
Because of the current drain of this pedal, we don't recommend using batteries instead of a power adaptor. Ok, \*maybe\*, if you've got 9V rechargeables.

## 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. However, in the interests of lowest noise, we recommend you use 1% metal film resistors and 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.

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.

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!



Trimmer PCB layout



Possible Trimmer footprints

The PCB is laid out with holes for various types of trimmer resistor. Any of the following layouts work fine. Multi-turn trimmers are recommended for fine adjustment.

The power protection diode suggested is 1N4002. Pretty much any diode from this series is ok. 1N4001 will work, 1N4003 or 1N4004 are fine too, although total overkill.

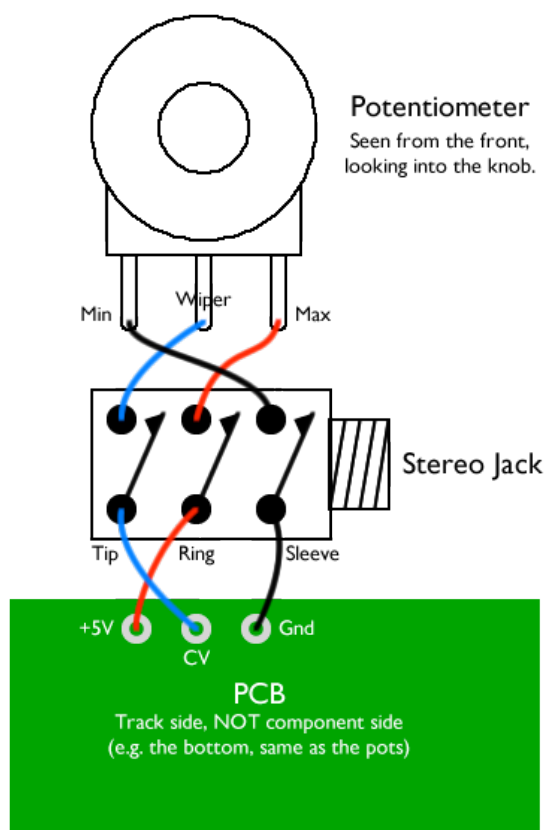
The 1N4148 signal diodes can be replaced with other small signal silicon diodes. 1N914 is a direct replacement and can be considered identical. Their role is to limit the signal to a level the BBD can handle (and to provide a soft clipping if driven too hard), so the forward voltage is the most significant parameter.

## Ideas for potential upgrades or customizations

### Adding CV inputs

Since the Electric Druid FLANGE chips operate using 0-5V control voltages like many other Druid chips, it is possible to add CV control of the Frequency, Depth, or Rate controls.

### Adding Expression pedal inputs



It is also possible to use an expression pedal in place of the pots. If a normally-closed stereo/TRS jack socket is wired in, 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 expression pedal may not be this way!**

Don't worry too much about this wiring. So long as you only use passive expression pedals, you can't hurt anything, even if it's completely wrong. If you get it back-to-front before you get it the right-way-around, no matter.

Both versions of the chip allow you to manually sweep the flanger with the farthest-left control (Manual or Rate). Connecting an expression pedal instead of this control would give you a foot-pedal wah-style flanger.

### Adding more switches and options

Both the MULTI FLANGE and the 4 KNOB FLANGE make use of the switch input marked "Tap" on the PCB. This input should have a switch which connects it to ground.

The MULTI FLANGE chip offers the option of top-down or bottom-up flanging. The switch could be a slide or toggle, or could be added as a footswitch. The switch should be latching.

The 4 KNOB FLANGE chip uses the "Tap" input to reset the waveform phase to zero. This allows you to keep the flanger in time, or can be used as a special effect, since you get a downsweep after you tap. This input needs a momentary switch, and might be most useful as a footswitch.