Electric Druid 4 second Digital Delay Project

Overview

Serving Suggestion

The Digidelay project aims to make building a good quality, feature-rich digital delay pedal with plenty of delay time an achievable aim for a competent DIYer. All parts are through-hole, and no rare or difficult-to-obtain parts are used.

The delay offers 32KHz/16-bit processing, and includes Delay Time, Repeats and Level controls, as well as Highpass and Lowpass shelving filters to control the tone of the echos.

It also allows echo splashes, tap tempo and delay trails, making it a fully-featured digital delay.

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 7 is arranged in roughly this order, depending on your component choices, 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 "DigiDelay" wording the right way up at the top left, you'll see that the components are arranged in four rows. The top row is passive components. The second row is for the ICs. The third row is more passive components. The fourth row is short because of the cutout for the In/Out sockets. It has all the power supply circuitry.

Resistors

Start with the resistors.

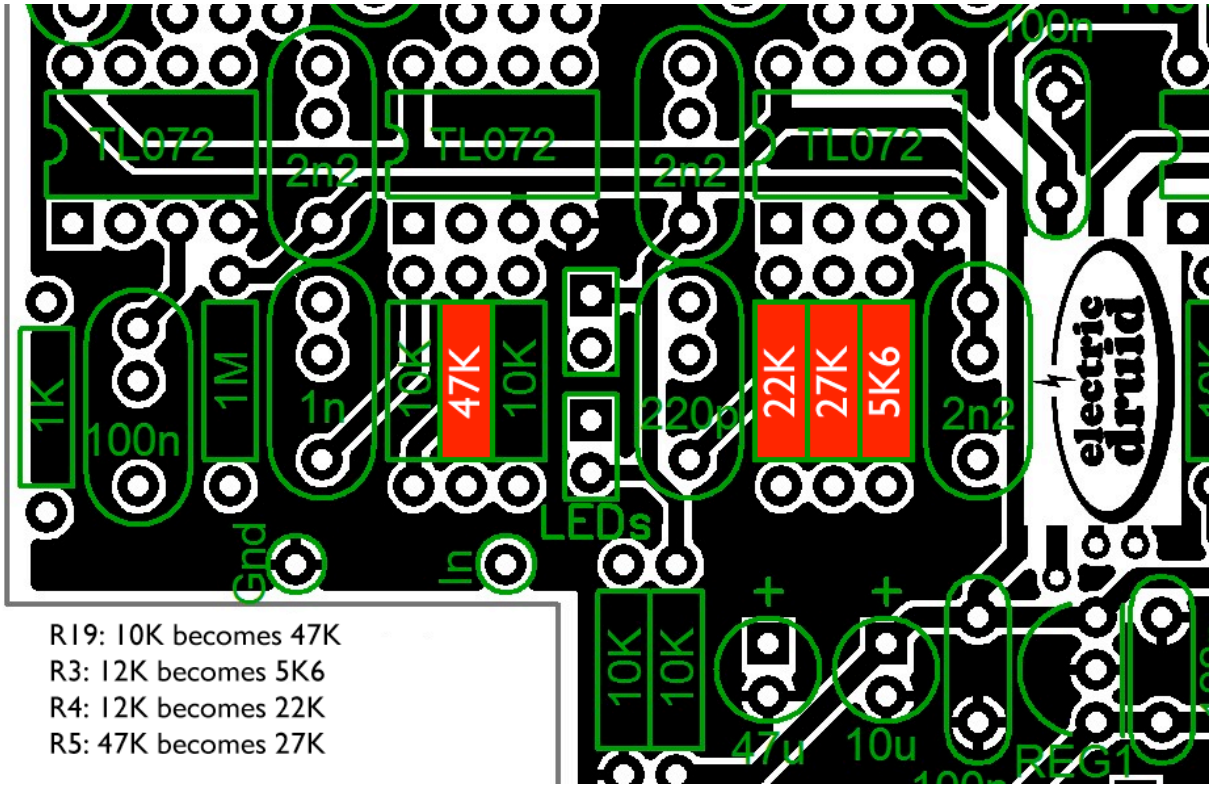
There are two options for four of the resistors, depending on whether you want the pedal configured for Synth/Line levels or Guitar levels, so we'll do those ones first:

Synth/Line level:

- 10K resistor x 1 - third row, below second TL072 chip
- 12K resistor x 2 - third row, below third TL072 chip
- 47K resistor x 1 - third row, below third TL072 chip

Guitar level

- 47K resistor x 1 (marked as 10K on the silkscreen) - third row, below second TL072 chip
- 22K resistor x 1 (marked as 12K on the silkscreen) - third row, below third TL072 chip
- 27K resistor x 1 (marked as 47K on the silkscreen) - third row, below third TL072 chip
- 5K6 resistor x 1 (marked as 12K on the silkscreen) - third row, below third TL072 chip



The rest of the resistors are the same for both options.

- 1K resistor x 1 - third row, far left
- 1M resistor x 1 - third row, far left
- 12K resistor x 2 - two on the top row
- 47K resistor x 1 - between the 12Ks you just did
- 100K resistor x 3 - third row, below DIGIDLY chip
- 10K resistor x 17 - eight on the top row, three on the third row, two on the fourth row, and four on the far right by the SRAMs
- 15K resistor x 2 - top row, centre left
- 43K resistor x 1 - top row, far left
- 560R resistor x 1 - third row, below DIGIDLY chip
- 200R resistor x 2 - third row, below DIGIDLY chip

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 below the fourth row of the PCB. This protects the PCB against reverse voltage, so be sure to check the orientation carefully.

Ground link wire

Install a wire link to connect the two ground planes (marked "Link"). This is located just above the Bypass and Dgnd pads on the lower edge of the PCB.

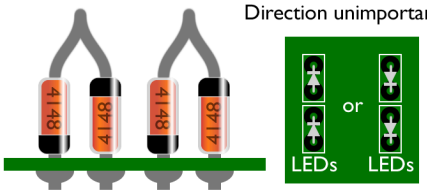
IC sockets

All six 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. This is particularly true with the large 28-pin socket. 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.

Regulators

The 78L33 +3.3V regulators REG1 and REG2 are both in the fourth row next to the power protection diode. Be sure to line up the flat side and the curved side with the markings on the PCB.

Overload protection diodes



Direction unimportant

These are marked as "LED" on the PCB, at the left side of the third row.

You can either install LEDs with a forward voltage <1.6V or two pairs of 1N4148 diodes in series, as shown left.

Note that it doesn't matter which way around the diodes/LEDs are orientated. The important thing is that **both point in the same direction**.

Ceramic bypass capacitors

There are five 100n ceramic power supply bypass capacitors, one at each end of the DIGIDLY chip, and three beside the 3.3V regulators on the fourth row. These are **not** the 100n film cap on the third 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 1 - third row, far left. The only "fat" 100n on the board.
- 2n2 (222, 0.0022u) capacitor x 3 - two between the TL072s, second row, and one next to the "electric druid" logo
- 220p (221) capacitor x 3 - two in the top far left, one on the third row on the left
- 1n (102, 0.001u) capacitor x 3 - two in the centre of the top row, one on the third row on the left,
- 470n (474, 0.47u) capacitor x 2 - both on the third row, right, below the DIGIDLY chip.

Tantalum capacitor

There's are only one of these, but you need to watch the polarity. It's on the top row, above the DIGIDLY chip. They're expensive and are easily damaged by reverse voltage. Note that usually the positive leg is marked, rather than the negative.

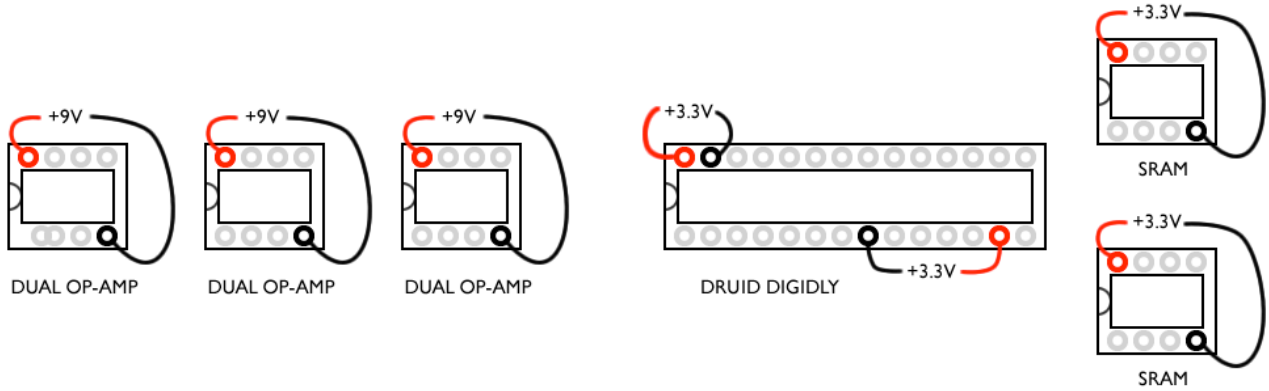
Electrolytic capacitors

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

- 47u capacitor - fourth row, far left
- 10u capacitor x 3 - fourth row

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 are two separate 3.3V power supplies, one for the digital circuits (uP and SRAMs) and one for the analog side (pots and ADC). There should be 3.3V power across pins 8 and pin 13 of the DIGIDLY, and also the analog supply on pins 27 and 28. The SRAMs have 3.3V power on pins 4 and 8.



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.

LEDs

I like to leave these until I've drilled the enclosure, since then it is possible to fit the board into the enclosure with the LEDs loosely fitted, turn the whole lot upside down, and solder the LEDs with the legs exactly the right length to protrude through the front of the enclosure. Do them along with the off-board wiring.

Install ICs

If the voltage check was ok, you can install the six chips; three dual op-amps, the Electric Druid DIGIDELAY processor, and the two SRAM 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 pots.

When drilling the hole for the Power Socket, be careful to give yourself enough clearance above the PCB.

There is a printable drilling guide available on the website at:

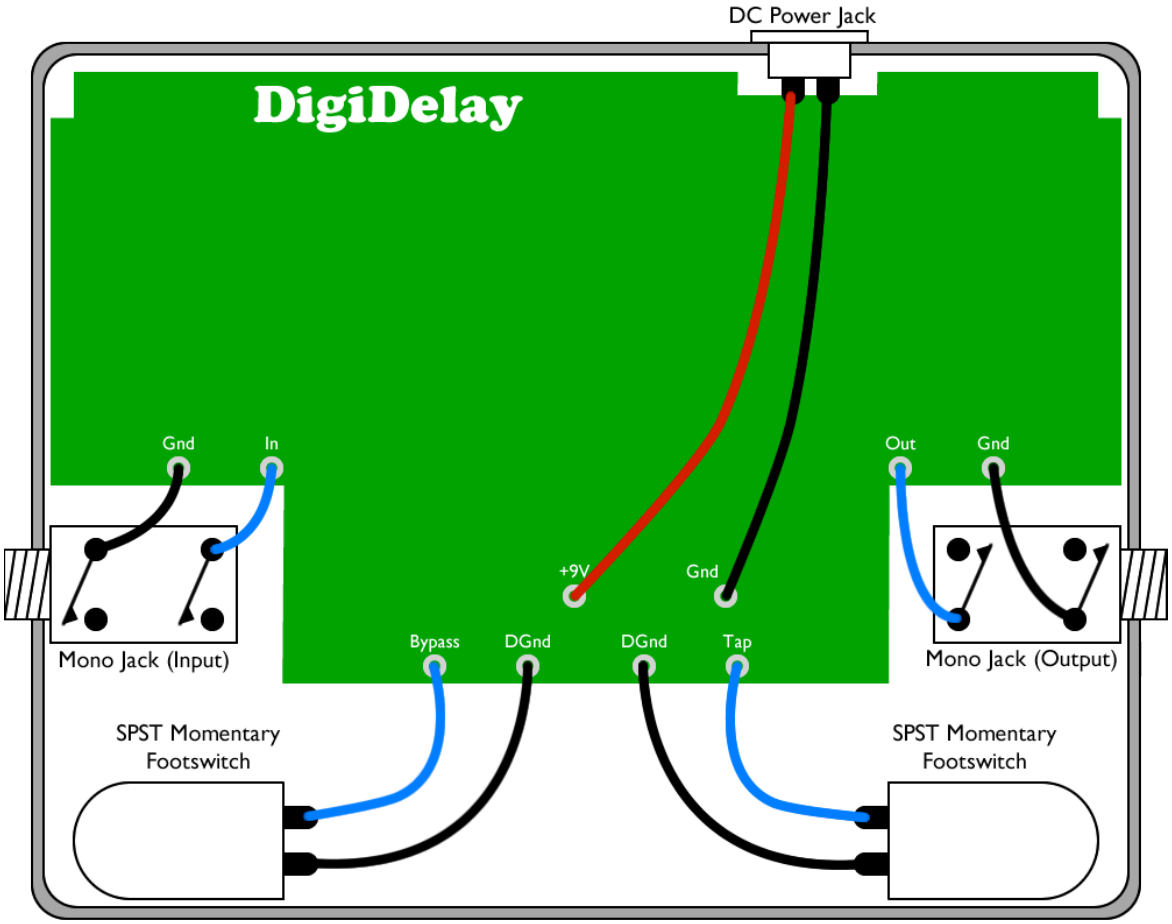
http://electricdruid.net/wp-content/uploads/2017/01/DigiDelayPanel.pdf

Off-board wiring

The off-board wiring for the Digidelay is very simple because the pedal uses a buffered bypass. This means the switching is done on the PCB, which reduces wiring. The LEDs are also board-mounted, which saves some more wires too.

The simplest way to wire the pedal is shown below.

It is also possible to wire the pedal with a 3PDT for true bypass switching in the typical way if required.



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 DigiDelay pedal!

There are no adjustments to make to the pedal, but make sure you have the Level control turned up when you're testing so you can hear some delays, and turn the Repeats up to make the effect nice and obvious. The effect on/off LED near the bypass footswitch should be on.

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

Guitar Level

Order	Ref	Description	Value	Quantity	Done?
1	R3	1% Metal film resistor	5K6	1	
2	R4	1% Metal film resistor	22K	1	
3	R5	1% Metal film resistor	27K	1	
4	R19	1% Metal film resistor	47K	1	

Synth/Line Level

Order	Ref	Description	Value	Quantity	Done?
1	R3, R4	1% Metal film resistor	12K	2	
2	R5	1% Metal film resistor	47K	1	
3	R19	1% Metal film resistor	10K	1	

Both options

Order	Ref	Description	Value	Quantity	Done?
1	R1	1% Metal film resistor	1K	1	
2	R2	1% Metal film resistor	1M	1	
3	R12, R13	1% Metal film resistor	12K	2	
4	R14	1% Metal film resistor	47K	2	
5	R6, R7, R21	1% Metal film resistor	100K	3	
6	R8, R9, R15, R16, R18, R20, R26, R27, R28, R29, R30, R31, R32, R33, R34, R35, R36	1% Metal film resistor	10K	17	
7	R10, R11	1% Metal film resistor	15K	2	
8	R17	1% Metal film resistor	43K	1	
9	R22	1% Metal film resistor	560R	1	
10	R23, R24	1% Metal film resistor	200R	2	
11	D1	Rectifier Diode	1N4002	1	
12	3 x TL072, 2 x SRAM	IC sockets	8-pin DIP	5	
13	DIGIDLY	IC socket	28-pin DIP	1	
14	REG1, REG2	+3.3V Regulator	78L33	2	
15	LEDs (Overload protection diodes)	LEDs with Vf<1.6V or series pairs of 1N4148	3mm LED / 1N4148	2 or 4	
16	C13, C15, C17, C19, C20	Ceramic capacitor	100n	5	
17	C1	Film capacitor	100n	1	
18	C2, C6, C8	Film capacitor	2n2	3	
19	C3, C7, C9	Film capacitor	220p	3	
20	C4, C5, C10	Film capacitor	1n	3	
21	C11, C12	Film capacitor	470n	2	
22	C14	Tantalum capacitor	10u Tant	1	
23	C16, C18, C21	Electrolytic capacitor	10u	3	

Order	Ref	Description	Value	Quantity	Done?
24	C22	Electrolytic capacitor	47u	1	
25	LED1, LED2	Low current (max 4mA) light-emitting diodes	5mm LED	2	
26	VR1, VR2, VR3, VR4, VR5	16mm Pots	10K Linear	5	
27	Unmarked	Pot dust covers or plastic			
28	IC1, IC2, IC3	Dual audio op-amp	TL072	3	
29	IC4, IC5	1Mbit/128KB SRAM	23LC1024	2	
30	uP1	dsPIC 33FJ64GP802	DIGIDELAY	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
- 2 x Mono 1/4"/6.35mm Input and Output jacks
- 2 x Stomp switch, SPST momentary (clickless/soft touch is nice for this project)
- Power Input socket, 2.1mm
- 5 x Knobs

There may be sufficient space in the 1590BB enclosure to include batteries, but the current drain of the pedal is enough to make it an expensive proposition.

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.

Op-amp choice is not critical. Any 8-pin dual op-amp with the standard pinout will work. A good quality, low noise audio op-amp like the TL072 is ideal, but LF353 or MC1458 would also work.

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 overload protection diodes can be either LEDs or small signal diodes. The original unit used 3mm LEDs, but finding LEDs with a low enough forward voltage (Vf<1.6V) is not easy. A pair of 1N4148 or 1N914 signal diodes in series gives a Vf of 1.3V, which is fine. Their role is to limit the signal to a level the DIGIDELAY chip's ADC can handle.

I can't find/afford a tantalum capacitor!

Tantalum capacitors are really just a specific type of electrolytic capacitor. In this case, we ideally need a 10uF capacitor with an equivalent series resistance of less than 2 ohms (ESR < 2ohms). There are "low ESR" series of aluminium electrolytics which meet this criteria. In practice, a typical aluminium electrolytic cap will work, although it may not be ideal. The lower the ESR, the better.

Ideas for potential upgrades or customizations

Adding a switch for Delay Tails

The delay tails jumper can be wired to a switch on the panel, if you can find space.

Tweaking the firmware

The DIGIDLY firmware is available for download on www.electricdruid.net/diy-digital-delay. The code is released under a non-commercial, share-alike license, so you can tweak it however you like for your own use. Full license terms are at www.electricdruid.net/legalstuff. Since most of the functionality of a digital unit like this is done in software, the range of modifications that are possible by altering the firmware is huge. The PCB can even be used as the basis for your own digital effect, something completely unlike a delay. The board offers In/Out buffers and filtering, and five control pots and two switches to alter parameters in the code, so the hardware is extremely open-ended.