Wobble Triple LFO/EG - build instructions

fuzzySynths

This module provides a programmable three channels of control voltage from 0-10v, with 3 trigger inputs. There are dual-colour illuminated buttons and a pot for each channel.

Two 12 bit Digital-Analog Converters (DACs) provide the output for 4 channels, 3 coming out to the front panel and a 4th on the back of the module. Standard TL072 opamps amplify the unipolar outgoing signal up to 0-10v. Transistors protect the microcontroller from excessive voltages on the 3 gate inputs.

Preparation

You should have 3 separate wobble PCBs, labelled A, B & C.

Board A mounts the jacks, pots and switches. Board B amplifies the output signal and limits incoming signals to no more than 5v, to protect the microcontroller. Board C hosts the Nano microcontroller as well as the 2 DACs. It is a deep build, going back 50mm, with a lot of connections between boards.

There are a couple of issues with version 1.1 PCBs - if you have one of these, please see the addendum at the end of this document

These instructions assume a degree of experience building and soldering, plus tools - you will need a soldering iron and solder, a multimeter, and snips / cutters. A small hacksaw makes it easier to cut the header sockets to the correct length.

We'll populate the boards starting with the smallest components. The components go on the back of the PCBs, with the pots & jacks go on the other side of board A. The front side is marked with the version number and board eg "v1.1 A".

Resistors

Stuff the board with resistors, then cover with some card to flip the board over without them falling out. Solder in place, then trim the leads short.

Board A

R1, R2, R3: 3 x 270R

R1-3 limit power through the button LEDs

Board B

R7, R8, R10, R11: 1k x 4

R20, R21, R22, R23: 2k7 x 4

R16, R17, R18, R19: 3k9 x 4

R9, R13, R15: 10k x 3

R6, R12, R14: 100k x 3

The 2k7 and 3k9 together set the amplification of the TL072 to a gain of 2.44 (gain = 1 + R18 / R22). The maximum output from the DAC is 4.096v, so a gain of 2.44 brings this to +10v.

Board C

R4, R5: 10k x 2

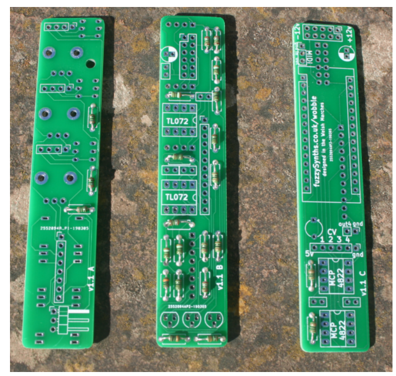


Figure 1: Resistors placed

Sockets and smaller components

Working up in height, add the 8 pin DIP sockets which hold the ICs, then the bypass capacitors and tantalum for the DACs. Ensure the notch on the socket matches the notch on the board silkscreen markings, so the IC is orientated correctly.

DIP sockets

U1, U4 (board B): 8 pin DIP sockets x 2

U2, U3 (board C): 8 pin DIP sockets x 2
Use sockets, rather than solder the ICs directly.

Capacitors

C1, C8, C9, C2 (board B): 100nF ceramic capacitors x 4

C5, C6 (board C): 100nF ceramic capacitors x 2

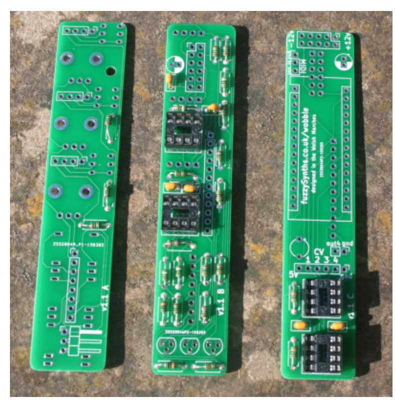


Figure 2: DIP and bypass caps placed

Transistors

Q1, Q2, Q3 (board B): 2N3904 x 3
The orientation matters, the flat edge should fit the markings on the board.

The transistors invert incoming voltage on the gate input sockets. This protects the arduino from negative voltages and anything over 5v, which would otherwise fry it.

Connectors

Fitting 3 channels into a small space is a challenge, meaning lots of connections between boards. Starting from board A, the male pins fit on the front (component side) of the boards and the female sockets on the back of the next board, so they should all fit together. Unless you have a kit with the lengths already cut, it is easiest & cheapest to snip off the size you need from a long connector.

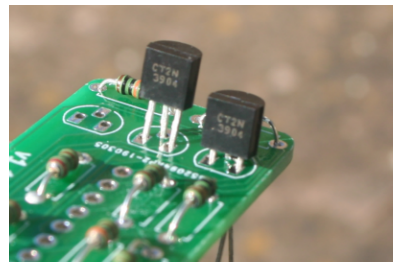


Figure 3: correct transistor orientation

Pin headers

J1, J5, J6 (board A): 4 pin long, single row x 3

J20 (board A): 8 pin single row x 1

J12 (board B): 12 pins single row x 1

J10 (board B): 2 x 6 pins (double row) x 1

J11 (board C): 2 x 5 pins (double row) x 1 (power header)

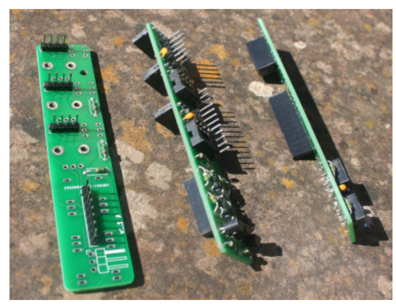


Figure 4: stacking boards

Header sockets

To ensure the boards stack together, it's easiest now to put the sockets onto the corresponding pins, then fit the board onto the sockets and solder so they fit correctly. Board A connects to board B, and B to C. It can be done either way, so let's agree the sockets face forward, except the 2 rows used to connect the Arduino on the front of board C.

J13, J21, J22 (board B): 4 pin single row header sockets x 3

J23 (board B): 8 pin single row header sockets x 1

Connect board B to board C as above.

J15 (board C, reverse): 12 pin single row header sockets x 1

J14 (board C, reverse): 2 x 6 pin (double row header) sockets x 1

Now we'll add the Arduino Nano to board C. You can solder it directly, or add female sockets (15 pin x 2) if you might want to remove it at some point. Ensure the Nano is in the sockets before soldering them in to ensure a good fit. If you solder directly to the PCB, connect a USB lead first to make sure it will connect without it fouling the DIP sockets (with the ICs in) - this might meaning pushing the other end in further.

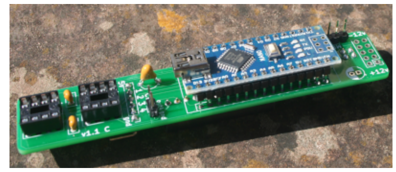


Figure 5: soldering Nano directly

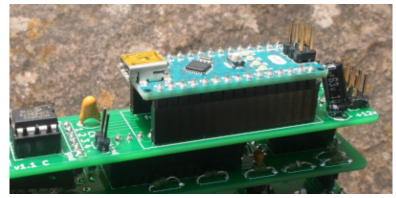


Figure 6: socketing the Nano

A1 (board C, front): 2 x single row header sockets, 15 pins long

Optional

J24 (board A: right angled 3 pins single row x 1
(gives access to normalised inputs) [1]

[1] see below for v1.1 PCBs

J17 (board C): 2 pins single row x 1 (buffered 4th output)

J18 (board C): 3 pins single row x 1 (MIDI header)

J19 (board C): 6 pins right angle single row x 1
These pins give direct access to the DAC outputs, plus a 5v supply and ground, which lets the module be run purely from USB, making it a useful workbench tool. The right angled pins are much better placed underneath the board, on the opposite side to the arduino, so they don't interfere with the USB lead.

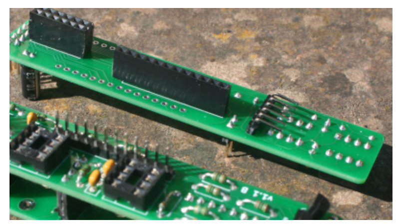


Figure 7: DAC outs, J19 orientation

Now stop and check closely for any solder bridges between pins, with a magnifying glass and / or the continuity tester of your multimeter, looking for shorts between pins. None of these pins should be electrically connected to their neighbour. Some of the components we will add next will cover the pins, so this is your last chance to check.

Polarised capacitors

BE CAREFUL, as the next three components are polarised & need to go in the correct orientation. The longer lead goes in the hole marked with a +, while the stripe on the electrolytics marks the negative lead and should match the white part of the silkscreen.

C4 (board B) 10uF electrolytic capacitor x 1
C4 needs to fits between the boards - if it is tall (> 10mm), it may need to be bent over slightly so it fits.

C7 (board C): 10uF electrolytic capacitor x 1

C3 (board C): 10uF tantalum capacitor x 1
The longer lead, marked with a +, goes in the hole nearest the Arduino.

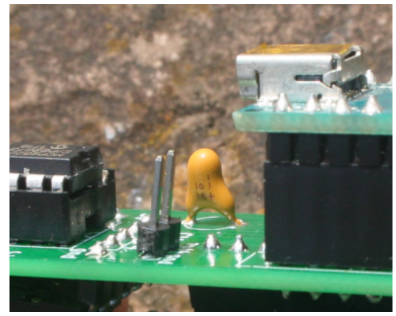


Figure 8: tantalum cap orientation

Final components

Board A

Board A should have the resistors in place, and headers on the front - double check these as it's about to get tight. We need to add the 3 pots, 6 jack sockets and 3 illuminated buttons to the other side of the board. It's worth fitting these loosely, then placing the panel on top to ensure they are aligned before soldering into place.

J2, J3, J4, J7, J8, J9: Thonkiconn 3.5mm jack sockets x 6

SW1, SW2, SW3: Illuminated buttons (PB6149L-xx-102) x 3
The buttons need pushing really firmly through the board - the pins should come all the way through. Red on the outside, green to the middle.

The buttons can be placed either way round, which affects the colour choices. The colour you want in edit & trigger mode should be visible on the outer side of the PCB, with the usual colour facing the middle

RV1, RV2, RV3: 10k linear (B) pots x 3
Some makes of potentiometers have large mounting legs to the side which may need filing to fit

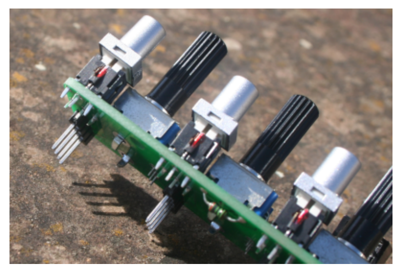


Figure 9: LED orientation

Board B

You can turn your soldering iron off now.

U1, U4 Insert the 2 x TL072 op-amps into the DIP sockets, ensuring the orientation is correct - the dot on the chip should be towards the middle of the board, matching the notch on the socket.

Board C

U2, U3 Insert the 2 x MCP4822 DACs into the sockets, ensuring the orientation is correct - the dot should be on the same side as the notch, facing up towards the Nano.

If your pots don't attach to the front panel, it may bend under button pressure, so screw an M3 through the mounting hole in board A to a spacer glued to the front panel .

Figure 10: spacer glued to front panel

Testing

The build is finished, but it's worth a final check for any shorts - it's much easier to check now than troubleshoot when something doesn't work.

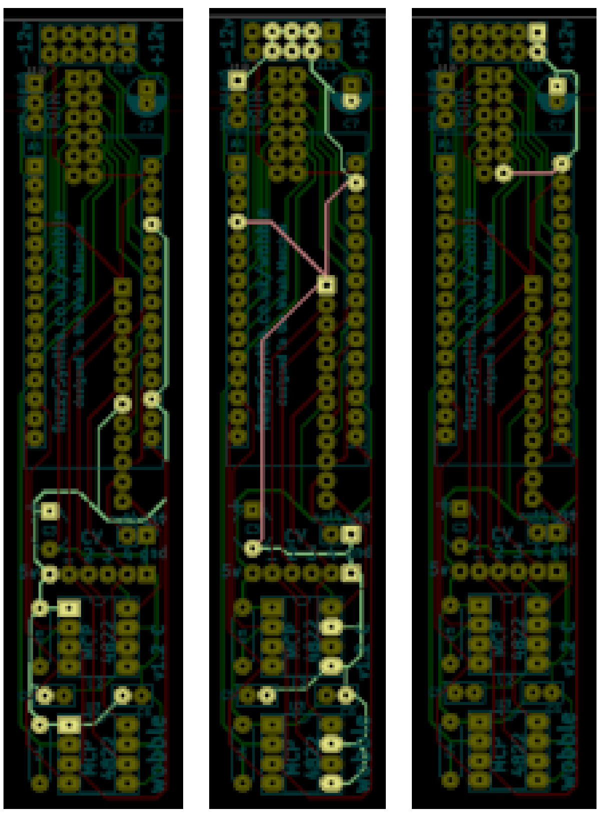
Set your multimeter to continuity mode, and let's check the power rails.

5v pin 1 of both DACs should be connected, as well as pin 18 and 27 of the Nano.

ground pins 4 and 29 of the Nano should connect to the ground pins, and pins 5 & 7 of both DACs.

12v pin 30 of the Nano should connect to the 12v power input

No adjacent IC pins should be connected.



Software

If your Arduino Nano is not pre-programmed, now's the time to do it. Connect to the Nano via USB or a programmer. It doesn't have to be attached to the board.

Latest version of software is at `https://github.com/fuzzySi/wobble`

The arduino IDE can be used to edit and upload the .ino file.

The .hex file can be uploaded using avrdude:

```
sudo avrdude -B 1 -V -p m328p -c usbtiny -P usb -U flash:w:wobble_v0_8.ino.eightanaloginputs.hex:i
```

Now stick the Nano in, connect to power, and turn it on. It will work connected to USB, but will be restricted to 0-4v via the pins at the back, not the front panel.

Notes for PCB v1.1

There are a couple of issues with v1.1 PCBs.

J24 (optional) The holes are offset slightly, meaning you may need to file the inner part of the overlying jack so it sits correctly

Silkscreen v1.1 PCBs have a silkscreen error meaning the components are not labelled, so please follow the diagram carefully