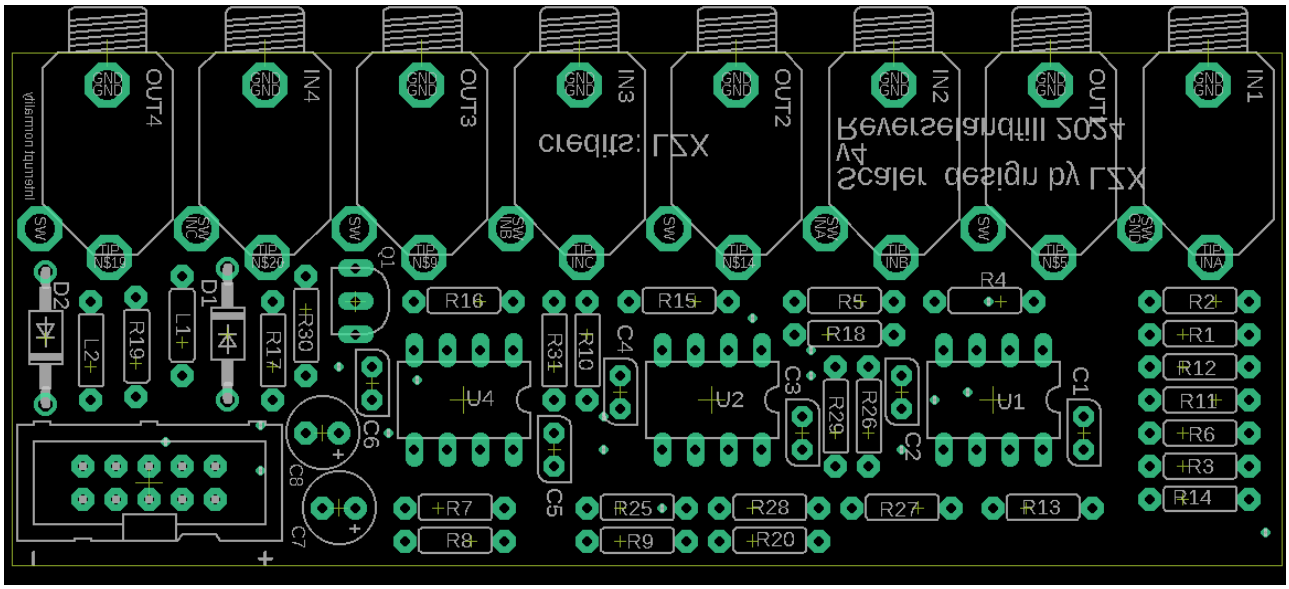
Scaler Buildguide v2 2024



Use this guide together with the Scaler BOMs ,
so make sure you have the latest versions:
https://github.com/MartijnVerhallen

This Scaler module is a recalculated version of the LZX Cadet V Scaler,
meaning that all 4 optional builds are available both in schematic, BOM and simulated format.
The input jacks are normalled together for easy patching
The PCB is also slightly smaller than the original.

The Scaler kit contains all parts to be able to build 1 of the four possible options:

a) +1v to +5v
b) +1v to +-5v
c) +-5v to +1v
d) +-5v to +-1v

Option C might be the most used option. You can use this to control your video modules with (audio Eurorack) LFOs , VCOs and other modules the have a +-5v output.

If you want to use audio modules as video effects, you can use 2 Scaler modules:
option B to boost the video voltage up to audio level, and option C to scale it back down again.

Note that the Scaler opamps are TL072. This will cause some blurring.
You can replace them with LM6172 but note that most audio modules won't have high frequency opamps, so they might also blur the edges of the video signal.

Some video modules can handle +-1v , use option D
If you want to use modules that have an positive only input 0 to +5v , use option A
(for example envelopes or digital modules)

Note 2: This Scaler options are build around “optimal” voltage inputs.
When a module doesn't reach the full scale that is expected, the output voltage of the Scaler will also be slightly different.

Choose an option and use the corresponding BOM!

Resistors
Solder all resistors!
DNP means “Do Not Place”
0r or link means that you have to solder a wire link. Use a snipped off resistor lead.

Diodes
These parts have an orientation. the black line on the diode should match the white line on the PCB.
Place the black 1N4001 diodes at D1 and D2 and solder them.

Ferrites
Place the 68r ferrite beads at F1 and F2 and solder them.

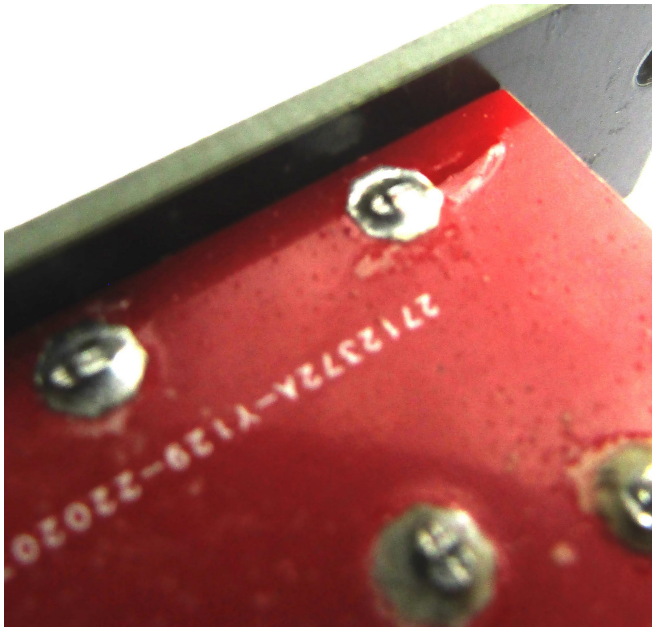
Ceramic Capacitors
Place the 100nF capacitors and solder them.

IC sockets
Place the sockets (take care of the orientation
Solder one pin and check if the socket is flat to the surface of the PCB, reheat if needed.
Then solder the rest of the pins.
IC1 is the TL431 voltage regulator.

Power header
Place the 10pin power header and flip the PCB around. Solder one pin and check if the header is flat to the PCB surface. Reheat the soldered pin if needed, then solder the other pins.

Electrolythic Capacitors
Place the three 10uF and the one 1uF Electrolythic Capacitors.
The Longer lead is the PLUS side, marked on the PCB with a small + sign and a triangle.

Jacks
Place the jack sockets on the PCB. DO NOT SOLDER YET!
To align the jacks and switch correctly, we now mount the panel first.
Attach the nuts to the jacks and loosely fasten them.
First solder only one of tab of each jack.
Check if all jacks are aligned to the panel, then solder the rest of the tabs.
Snip off the tabs after soldering, as they stick out too far and can cause shorts when mounted in your rack.



Checking the PCB for faults:
Check the orientation of the IC's , diodes, electrolythic capacitors and the power header
Look at your solder work for shorts and missed pads.

Reverselandfill.org
Design by M.Verhallen
Credits: LZX