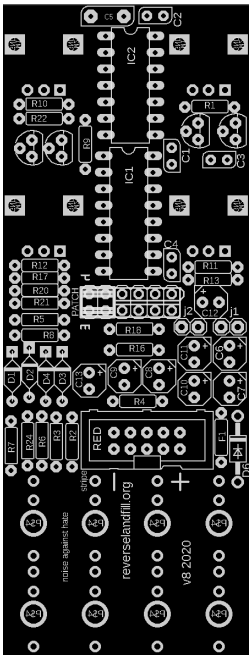
Noise Oscillator Build Document. V8



The Noise! Synthesizer is a CMOS IC based instrument.
The synth has 4 squarewave oscillators that modulate each other and go throught a clock divider.
After that, the separate divider outputs are mixed and shaped together.

Start with the ***Resistors*****:**

Note that some of the resistors are placed close together. Be careful not to make solderbridges!

The one **10R** is used at the power input as a fuse.
Bend the legs of the resistor 90 degrees and place the **10R** resistor at the **F1** location on the PCB.
Solder the resistor and cut off the legs with a sidecutter.

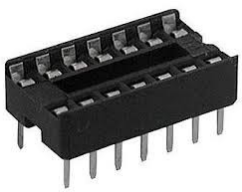
The twelve **1K** resistors act as output protectors.
Place the resistors at **R2, R3, R4, R9, R13, R16, R17, R18, R20, R21, R22, R24**
Solder and cut the legs.

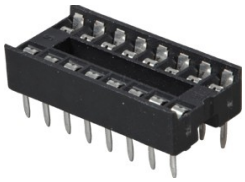
R5 to R8 are the four **1M** resistors. Again; solder them in place!

R1, R10, R11, R12 are **100R**, they limit the range of the Frequency Potmeters.

Diodes:
D1 to D4 are 9.1v zener diodes. The white mark on the pcb must correspond with the black line on the diodes.
They limit the incoming CV signal to 9.1 volt.
D6 is a reverse power protection, the **1N4001**

IC sockets and the IC's:
Take the foam with the *two* IC sockets and the IC's. There is *one* with 16 pins and *one* with 14 pins!





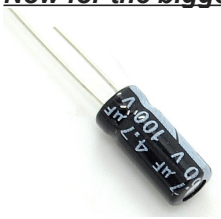
These IC-sockets are to make the placement (and possible removal) of the IC's easier.
Take out the *two* IC-sockets. Attention! These sockets have a direction. See the semi-circle gap?
On the PCB you can see this marking as well. Place the 16-pin IC socket in **IC1**.
Flip over the PCB and solder two legs, on in the upper row, one in the lower row, diagonally from each other.
Flip the PCB back to the other side to check if the IC-socket is flat to the PCB.
If not, push the socket lightly to the PCB and reheat the two solderd legs. It should click to the PCB.
Place the 14 pin IC socket in **IC2** and use the same method as before to solder it in. Solder all remaining legs.

Insert both IC's in the sockets. The **CD4040 (clock divider)** has 16 pins, the **CD4093 (quad NAND)** has 14 pins.
Bend the pins so that they are in a 90 degree angle. Push them in carefully but firmly.

Transistors:
These are the 2n3904 transistors. (they are unmarked on the pcb) Mind the orientation!
They act as CV input buffers. Carefully insert the legs and solder them in.

Capacitors!
There are a lot of capacitors in the kit.
They determine the frequency range of the oscillators and stabilise the power and outputs.
We start with the *two* yellow capacitors that look like resistors; **C1 and C2**, they are **100nF** (104)
Mount these “standing up”, else they won't fit in the footprint!
The **470pF** goes in **C3**. This is a slightly smaller orange disk with the code "471".
C4 is **330nF**, a bigger yellow blob.
The small rectangular shaped **470nF** film capacitor is placed at **C5**. It is a tight fit next to the IC.

Now for the bigger capacitors:



These parts have a **polarity**. The long leg is the PLUS, The short leg is MINUS.
The value of these components is printed on the side as well.
Start with the *three* **10uF** value**.** They go in **C6, C7** and **C12.** Long leg goes into the PLUS!!
(take care to get the right one, there are also *four* **4.7uF** capacitors!)

Now we do the four **4.7uF**. These go in **C8**, **C9, C10** and **C13**.
Last comes the **100uF (or 220uF)**, which goes into **C11**.

Powerheader
Insert the shrouded 10 pin header. Solder one or two pins, then check if the header is flat to the pcb.
Reheat if necessary. Then solder all pins.

Expansion headers
Patch = two 1x6 pins headers.
Z1 and **Z2** = two 1x2 pins headers
Solder the headers. Make sure they are mounted straight.

Flip the PCB around. It is time for the pots and jacks!

The Potentiometers:
These parts are variable resistors. With these you can adjust the frequency of the oscillators.
You have four of these.
Insert then into the PCB, but don't solder them yet.

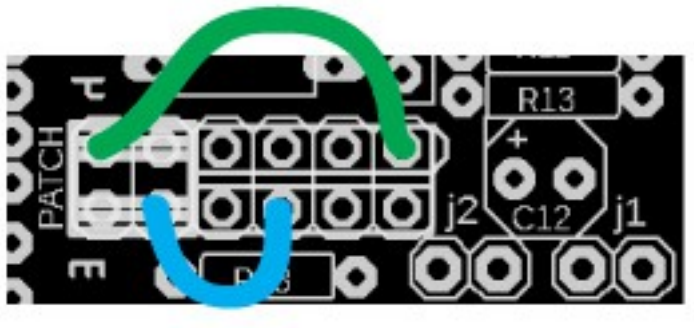
Jacks:
Insert all 8 jack sockets, don't solder them yet.

Front panel:
Attach the PCB to the frontpanel. Fasten a nut to one of the potmeters.
Check if all pots and jacks are correctly seated.
Then solder one pin on all these parts. Now check again if the parts are correctly aligned.
Now solder all remaining pins.

Fasten all the nuts to the pots and jacks.

Knobs:
Turn the four potmeters all the way to the left (CCW) and fit the knobs on.
Push them firmly in, while supporting the back of the potmeter.

Patchbay:
Output 3 and output 4 must be patch to route signal to the outputs.
Use the green Dupont wires to patch.
There are 8 outputs (these are clock divisions of the original noise signal).
Each destination has 2 inputs, marked with the white lines and the number “3” and “4”
It is possible to mix several outputs together, when using more patchcables.
This will result is a more complex noise structure.



Jumpers:
J1 and J2 determine the range of OSC1 and OSC2.
Place a jumper to lower the frequencies of the Oscillators into LFO range.

Mods:
Make a panel for all possible patchbay connections.
You can use a 16pin IDC connector to attach the patchbay to this new panel.
Use banana's, toch sensors, CD4066 switches or other creative solutions!
Add switches to the J1 and J2 jumper locations.
To lower the frequency range even more, change the capacitors C11 (OSC1) and C12 (OSC2).

Have fun with your Noise! Synthesizer!!! :)
martijn@reverselandfill.org
www.reverselandfill.org