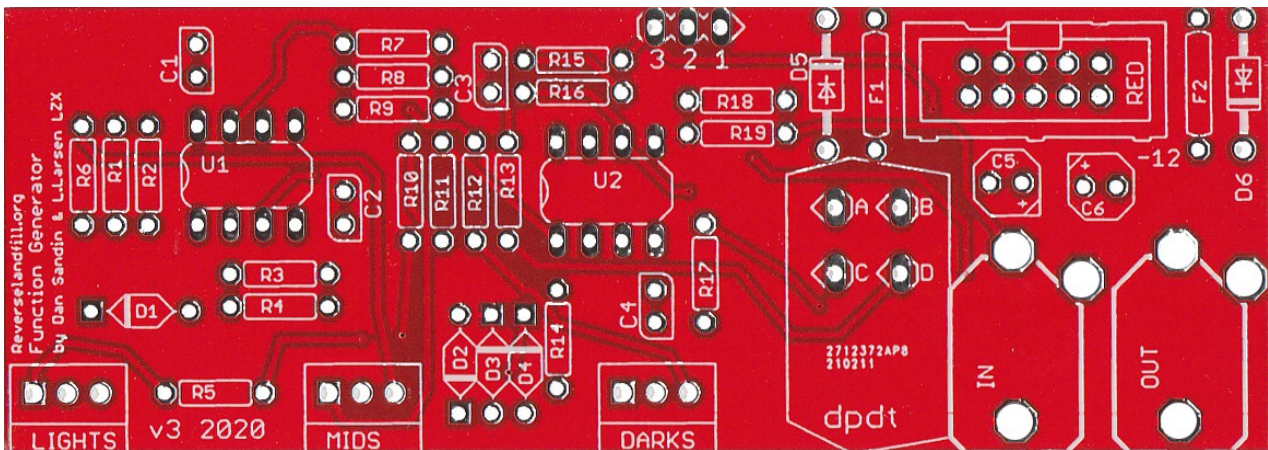


Triple Function Generator BOM version 2 – 2020

Boards:		amount
PCB		3
Panel		1
Resistors	value	total of 3 pcbs
r1, r5, r6, r10, r12, r14	4.99k	18
r2, r16	1k	6
r3	330r	3
r4	100r	3
r7, r8, r11	10k	9
r9	4.2k alternative: 4.32k	3
r13	5.66k alternative: 5.6k or 5.62k	3
r15, r18	499r	6
r17	24k	3
r19	100k	3
Diodes		
d1, d2, d3, d4	1n5711	12
d5, d6	1n4001	6
Ferrite		
f1, f2	ferrite bead. alternative: 10r	6
Capacitors		
c1, c2, c3, c4	100nF 2.5mm ceramic	12
c5, c6	10uF electrolytic	6
IC		
u1, u2	8pin socket	6
u1, u2	LM6172	6
Potmeters		
Lights, Mids, Darks knobs	10k lin 9mm right angled GTP6M-13X16-S from TME	9 9
Switch		
a, b, c, d	DPDT on-on	3
Jacks	PJ302M	6
Power		
10 pin shrouded header	or unshrouded	3
10 pin to 16 pin power cable		3
or use a multicable		1
Normalised inputs		
3 pin header		3
connect pin 1 on pcb1 to pin 2 on pcb2		dupont wire
connect pin 1 on pcb2 to pin 2 on pcb3		dupont wire

Triple Function Generator - Buildguide - v3 pcb 2020



There are 3 identical pcb's. You can populate the pcb's at the same time.
Make sure to take a break at times. Drink some nice ginger tea !

Resistors:

Start with the 6x 4.99k value (R1, R5, R6, R10, R12, R14)

2x 1k (R2, R16):

1x 330R (R3):

1x 100R (R4):

3x 10k (R7, R8, R11):

1x 4.2k (R9): - You can use 4.32K as alternative

1x 5.66k (R13): - You can use 5.6K or 5.62K as alternative

2x 499R (R15, R18):

1x 24K (R17):

1x 100K (R19):

Ferrite:

Solder the 2x Ferrite beads in F1 and F2. You can use 10R resistors as alternatives.

Diodes:

Solder the 4x 1n5711 Schottky diodes into place: D1, D2, D3, D4

The black band should match the white marking.

Then solder the 2x 1n4001 (reverse power protection) at D5 and D6

IC Sockets:

If you use sockets, they should go in U1 and U2. First solder 2 pins and see if the sockets are aligned correctly. If not, re-heat the solderpads while pushing down on the socket.

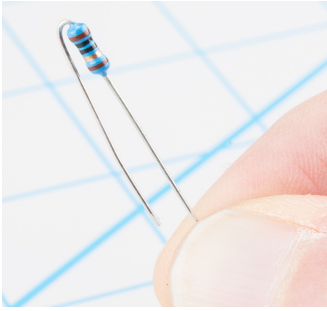
They will click flat to the pcb. Now solder the remaining pins.

Place the LM6172 IC's. Make sure the notch on the IC matches the socket and the marking on the pcb. Bend the pins so that the IC fits in the socket and push it firmly into place.

Capacitors:

Solder the 4x ceramic 100nF capacitors to C1, C2, C3, C4.

The kit has yellow capacitors that look like resistors, Those should be soldered in 'standing up':



Next are the Electrolytics, C5 and C6. The longer leg goes in the hole with the + (PLUS) sign.

Power header:

Fit the 10pin shrouded header into place. Take care of the orientation. The open side should face UP. (to the top of the PCB)

First solder one pin and make sure the header is aligned to the PCB. Re-heat if needed.

Then solder the remaining pins.

3 Pin Header:

Solder the 3 pin header into place.

First solder one pin and make sure the header is aligned to the PCB. Re-heat if needed.

Then solder the remaining pins.

This header is to normalise the input / output jacks. (parallel or serial connections are possible)

More on this later!

Potmeters:

Use 3x 10k linear 9mm potmeters.

First solder one pin and make sure the potmeter is aligned to the PCB. (90 degrees angle)

Re-heat if needed and push down on the body of the pot.

Wait with soldering the rest of the pins!

Jack sockets:

Use PJ302M jacks. Solder one pin and check if the jack sockets are aligned to the PCB.

Re-heat if needed and push down on the body of the jack.

Wait with soldering the rest of the pins!

Panel: (do this with one PCB at the time)

Mount the PCB to the panel to see if it fits. If so, remove the PCB from the panel.

Now solder the remaining pins of the Potmetes and Jacks.

Mount one of the PCB's to the panel again.

Mount the DPDT switch to the panel. Use resistor leads to solder PADS A, B, C, D.

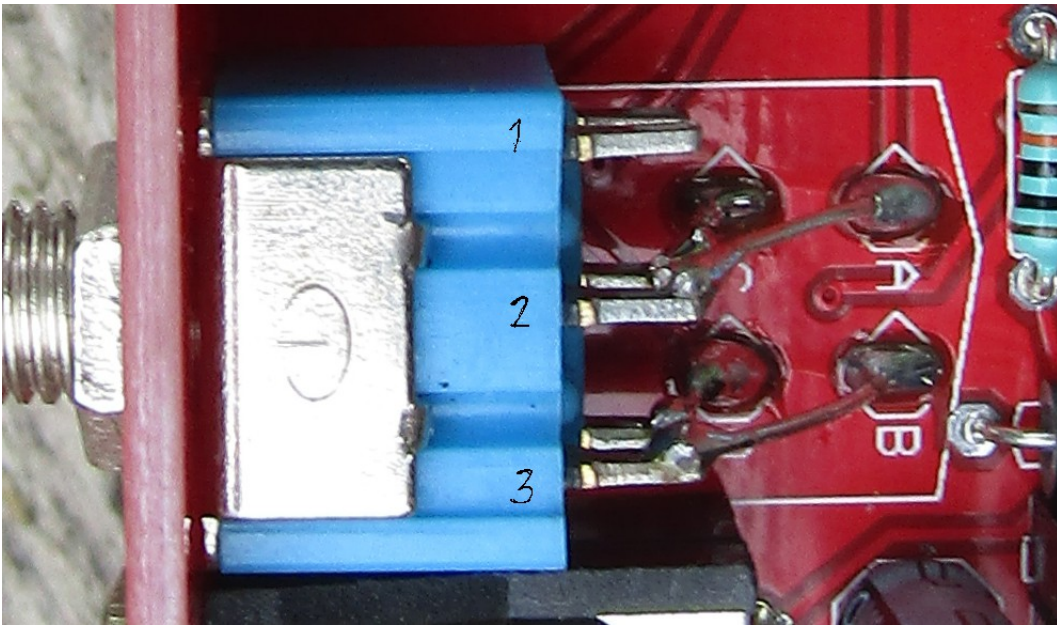
The switch looks like this: six pins, in rows of three.

1 2 3

4 5 6

Solder pad A to '2' , pad B to '3' , pad C to '5' , pad D to '6'.

Make sure the leads don't touch each other!



Now do the same procedure to the next PCB and then the last PCB.

Normalised inputs / outputs:

You can interconnect the 3 PCB's to achieve normalisation of the jacks.

With the headers soldered , you can use female Dupont cables to make the connection modular.

In my opinion the *Parallel routing* is the most useful.

This allows for easy video re-coloring and RGB functions.

The normalisation breaks when you insert a cable into input 2 and/or 3.

The 3pin header is setup like this:

1: IN----- (tip of the input jack)

2: OUT----- (switch of the input jack)

3: CHAIN—(tip of the output jack)

Use these configurations:

Parallel: (one input source gets routed to all three inputs.)

connect pin 1 on pcb1 to pin 2 on pcb2

connect pin 1 on pcb2 to pin 2 on pcb3

Series: (the output gets routed to the next Function Generator input, three in a row)

connect pin 3 on pcb1 to pin 2 on pcb2

connect pin 3 on pcb2 to pin 2 on pcb3

Power Cable:

Use the multicable to power the 3 PCB's.

If you want to make this cable yourself, just get 3x 10pin female IDC sockets and 1x 16pin female IDC socket. Snip off a long piece of flatcable (10pin) and slide the 3x 10pin connectors on the cable. Make sure they face the same way (pin 1 should be on the RED side) Now crimp these 3 connectors with enough space between them.

Measure this using the 3x panel-mounted PCB's.

At the end of the flatcable, slide on the 16pin connector. Check the orientation and crimp it.

Alternatively you can use 3 power cables. (but this can be somewhat unwieldy)

Knobs:

Turn all pots to CCW and mount the knobs. Now you are done!

This project is Open Source.

You can use the schematic to make your own boards, but please:

-Mention my name "Reverselandfill" or "M.Verhallen"

-Only do non-commercial runs. Private use only.

Credits: L.Larsen of LZX and D.Sandin

This project has been made possible by the LZX community forum.

Thanks to all contributing members!

<https://community.lzxindustries.net/>

www.reverselandfill.org - Questions? Mail me at martijn@reverselandfill.org