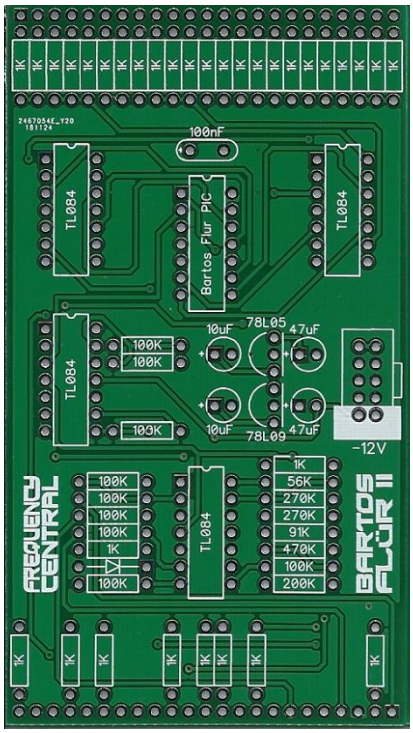
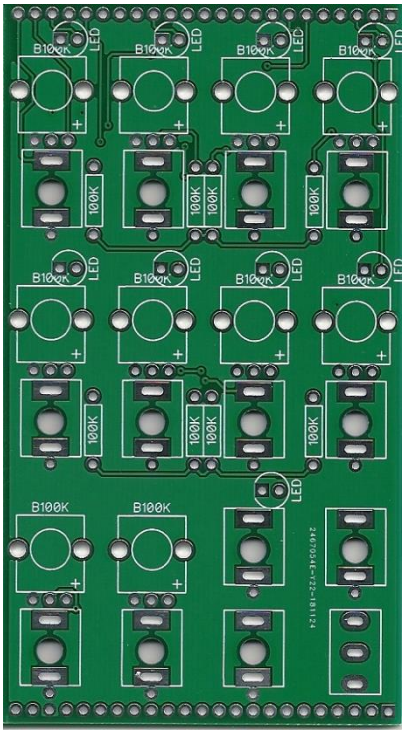
FREQUENCY CENTRAL

Build documentation for:

BARTOS FLUR II

Featuring code by Jetroid





Bartos Flur is essentially an 8 channel window comparator. There are two inputs, one suitable for processing unipolar signals, the other suitable for processing bipolar signals. Additionally, both inputs can be used for offsets if no signal is connected. There are 8 window outputs, each of which goes high at its relevant window. Each output also has an associated attenuator, the 8 attenuators are summed and output at the CV Bus output. A trigger bus output sends out a pulse as input signals pass through each of the 8 windows. The option exists to select either trigger or gate outputs. Finally, there is a clock input, which tells Bartos Flur when to sample the inputs, if this is left unconnected, Bartos Flur samples continuously.

Please observe the correct polarity for all ICs, voltage regulators, diodes and electrolytic capacitors.

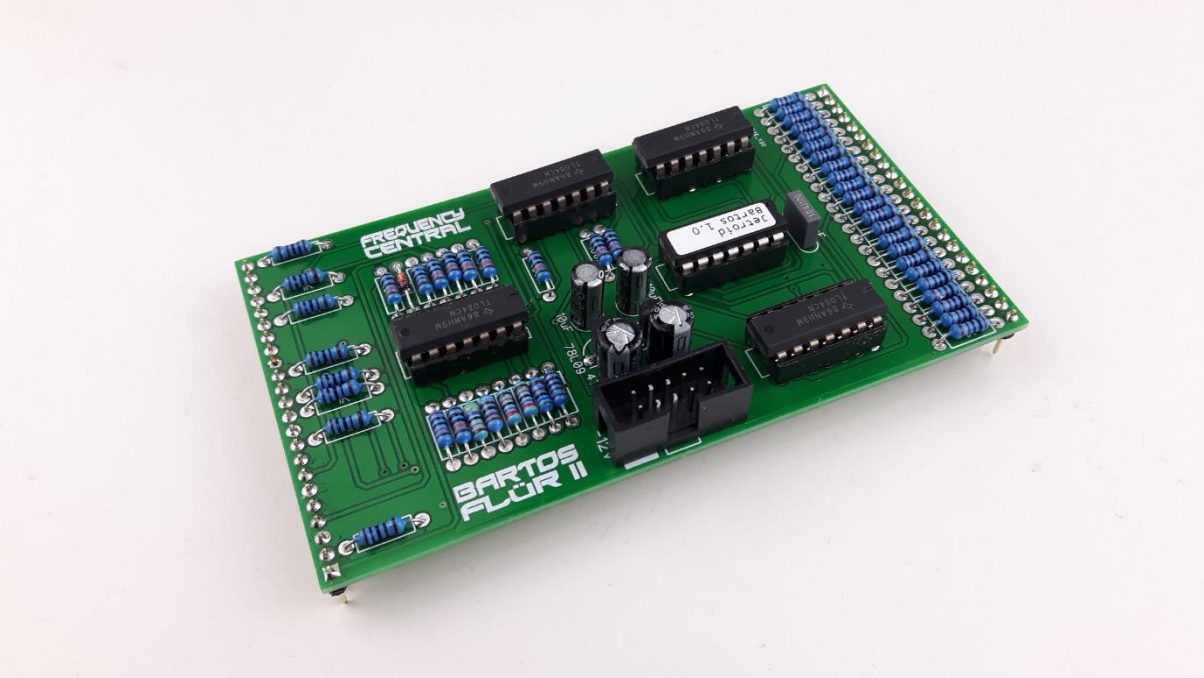
Bill of Materials

1K x 33 56K x 1 91K x 1 100K x 17 200K x 1 270K x 2 470K x 1 All resistors ¼ watt metal film.	100nF x 2 10uF x 2 47uF x 2	Bartos Flur PIC TL084 x 4 1N4148 x 1 78L05 x 1 78L09 x 1 3mm red LED x 9 14 pin IC socket x 5	B100K x 10 (or these)* 3.5mm socket x 13 Male header x 1 (cut to size) Female headerx 1 (cut to size) Power header x 1 SPDT switch x 1

* I prefer the Song Heui tall trimmers because they have a longer shaft and a white notch.

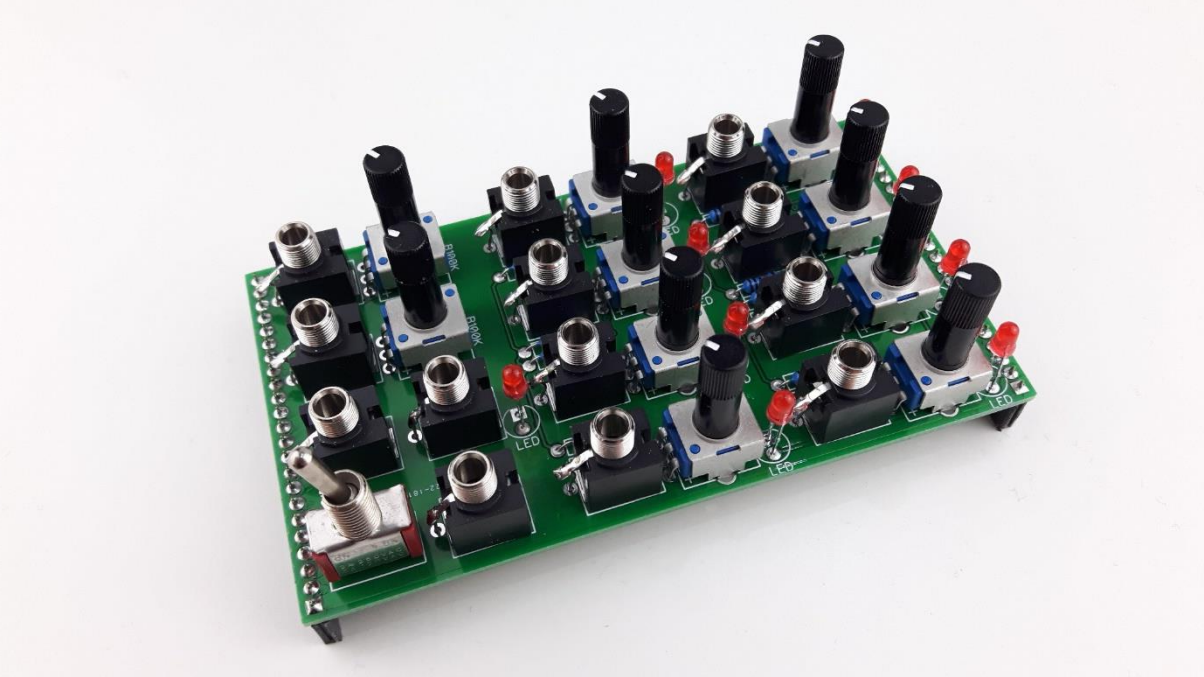
Main PCB assembly

1. Solder the diode and all resistors
2. Solder all four IC sockets
3. Solder the 100nF capacitor
4. Solder the 78L05 and 78L09 – watch the polarity!
5. Solder the box power header. Make sure the notch lines up with the screenprint legend. If in doubt, have a look at a power cable, and make sure when inserted into the header the red stripe lines up with the -12V screenprint.
6. Solder all electrolytic capacitors
7. Cut male headers to size and solder them into place. Make sure that they stick out of the bottom of the PCB.



Pots ‘n’ sockets PCB

1. Solder all resistors
2. Place all sockets on the PCB, making sure the ground tabs line up with the PCB’s ground pads – be careful here as there are four different orientations - then place the panel over them. This will assure that the sockets are correctly positioned. Flip the whole lot over and solder the sockets into place.
3. Use cut off resistor legs to connect the sockets’ ground tabs line up with the PCB’s ground pads.
4. Solder the potentiometers and switch in place
5. Cut female headers to size and solder them into place. Make sure that they stick out of the bottom of the PCB.
6. Put all nine LEDs through their pads – be careful here as there are two different orientations. Present the panel to the PCB, flip the whole lot over, make sure the LEDs stick though the holes in the panel, solder in place.



There’s no calibration to do! **But – make sure that you plug the main PCB into the pots ‘n’ sockets PCB the right way around!**

RDH 09/03/19

http://www.frequencycentral.co.uk/