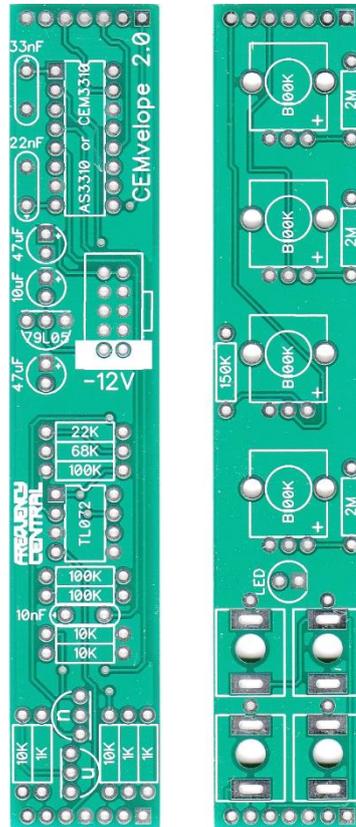


FREQUENCY CENTRAL

Build documentation for:

CEMVELOPE

Based on the Pro-One ADSR, featuring CEM3310 or AS3310 chip



CEMvelope is based on the legendary ADSR section of the Sequential Circuits Pro-One monosynth, with minor adaptations to integrate it into Euro format. Smooth and snappy! It has a gate input as well as a trigger input, the trigger input works in conjunction with the gate input for multiple triggering from a suitable controller device. CEMvelope has at it's heart a CEM3310 or AS3310. You can source your own chip, or buy direct from me when you order your PCB set and panel.

Key to PCB screen print:

n: This signifies NPN BC547 transistors. Note the correct pinout as shown by the half circles.

The PCB shows the correct orientation for BC547. Other similar transistor types can be used, but please observe the correct pinout.

Please observe the correct polarity of the electrolytic capacitors.

Bill of Materials

1K x 3	10nF x 1	AS3310 or CEM3310	Alpha 9mm B100K x 4
10K x 4	22nF x 1		
22K x 1	33nF x 1	TL072 x 1	
68K	10uF x 1	BC547 x 2	3.5mm socket x 4
100K x 3	47uF x 2	79L05 x 1	
150K x 1		3mm red LED x 1	Male header (cut to size)
2M x 3		8 pin IC socket x 1	
		16 pin IC socket x 1	Female header (cut to size)
All resistors ¼ watt metal film.		CEMvelope PCBs	Box header x 1
		CEMvelope panel	Davies 1900H knob x 4

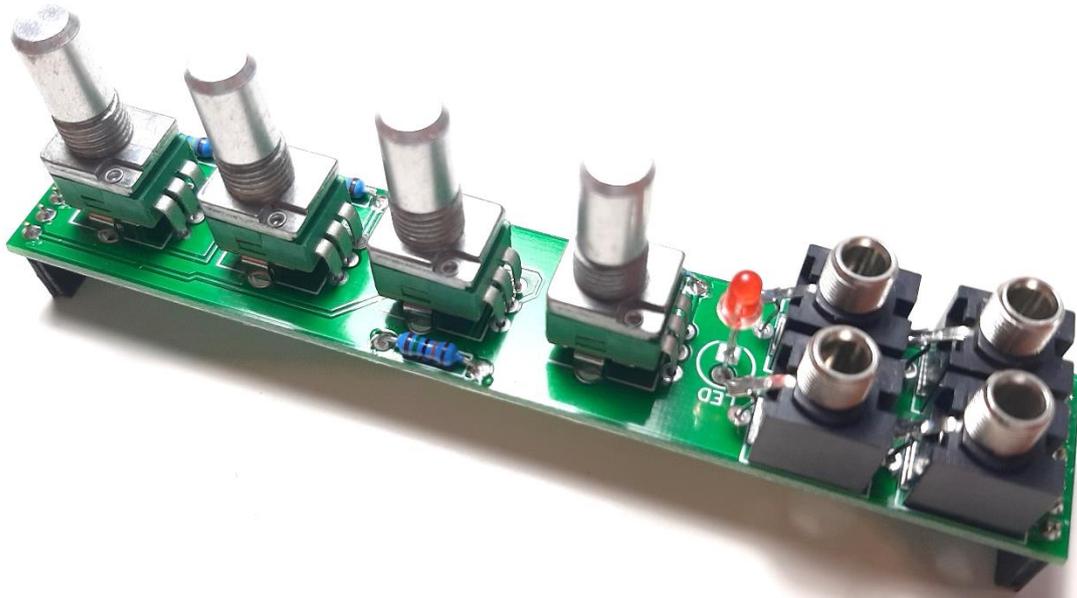
*Damn! Tayda don't stock this part. I often ask them to. If you ask too then maybe they will!



Main PCB assembly

1. Solder all resistors
2. Solder both IC sockets
3. Solder all three non-electrolytic capacitors
4. Solder both transistors and 79L09 – watch the polarity!
5. Solder the power header.

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.

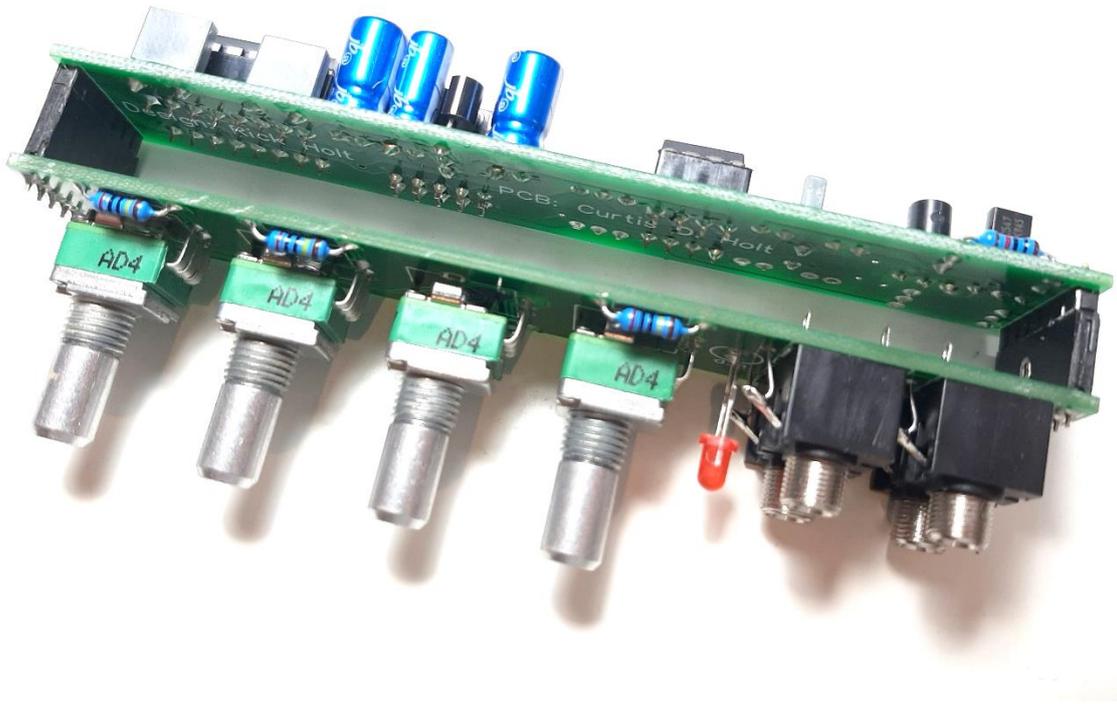


Pots 'n' sockets PCB

1. Place all pots on the PCB, and fold over their mounting tabs at the rear of the PCB, then place the panel over them. This will assure that they are correctly positioned. Flip the whole lot over and solder the pots into place.
2. Place all sockets on the PCB, making sure the ground tabs line up with the PCB's ground pads, 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. Cut female headers to size and solder them into place. Make sure that they stick out of the bottom of the PCB.

Note: Not all pots and sockets are equal in height. Providing you use the ones in the links provided, everything will line up perfectly.

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!**



Revised RDH 04/08/20

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