# FREQUENCY CENTRAL

Build documentation for:

TEMPOGRAF

Tap Tempo clock module with Start/Stop and division functions

Tempograf is a small 4Hp clocking module derived from our popular Chronograf module. It takes some of the key features of Chronograf and presents them in a small form, but strips out some of the more advanced features. The result is a handy little master clock module which can be used as a controller for our range of sequencing modules such as Polygraf, Little Melody and High Towers (as well as some forthcoming modules).

Main PCB

Control PCB

Top of PCB

Top of PCB

100nF

Stripe indicates -12V

Bottom of PCB
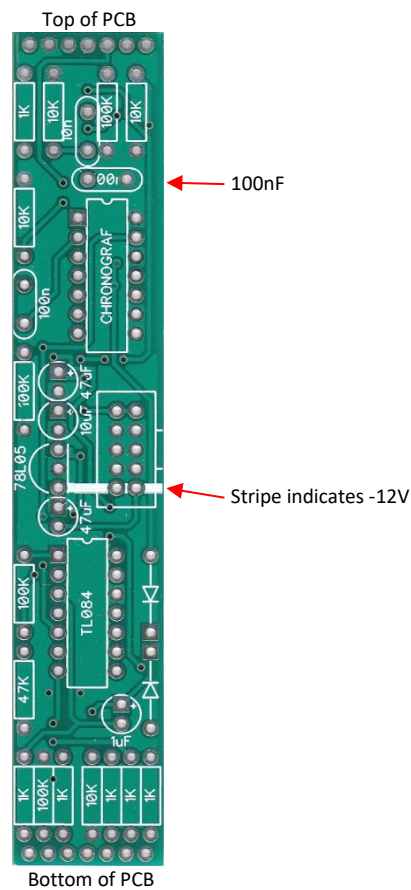
Bottom of PCB
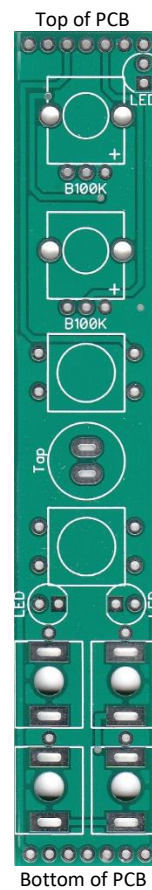
Bill of Materials

You will notice that all of the components listed below are also hyperlinks to where I buy each specific part from. You can also use the hyperlinks to find out more about what each component looks like. If you want to know even more, Google is your friend.

| 1K x 6	10nF x 1	Chronograf PIC	B100K x 2
10K x 4	100nF x 2	TL084 x 1	
47K x 1			Tactile switch x 2
100K x 3	1uF electrolytic x 1	1N4148 x 2	
	10uF electrolytic x 1	Red LED x 3	Momentary switch x 1
All resistors ¼ watt metal film.	47uF electrolytic x 2		
		78L05 x 1	3.5mm socket x 4
		14 pin IC socket x 2	10 pin box header
			Male header x 1
		Tempograf PCB set	(cut to size)
		Tempograf panel	Female header x 1
			(cut to size)
			Knobs x 2

Main PCB assembly
1. Solder all resistors and both diodes
2. Solder both IC sockets
3. Solder the 78L05
4. Solder all three non-electrolytic capacitors
5. Solder all four electrolytic capacitors
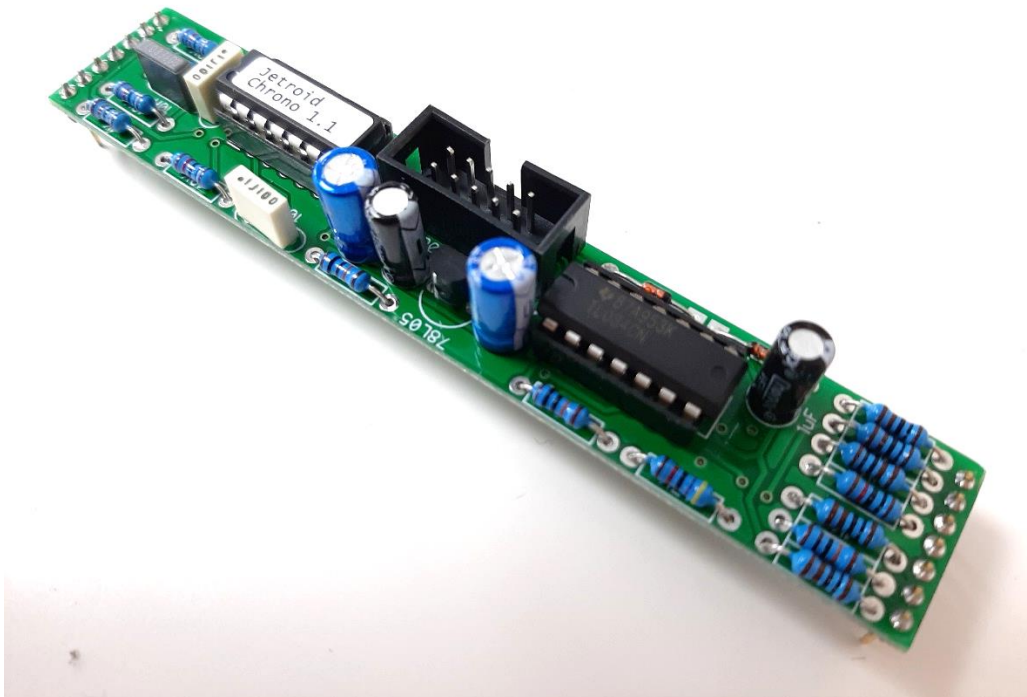
Control PCB assembly

1. Place the PCB on a flat surface. Place the 2 tactile switches into their solder pad. Do not push them all the way through, they should sit proud of the PCB (see photo), their legs should not protrude through the other side. Solder them into place from the top of the PCB.

2. Solder the 2 potentiometers and the Tap LFO momentary switch. Use the panel to make everything line up nicely.

3. Place all four 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. 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.

5. Put all three LEDs through their pads. 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.

6. Mount the Control PCB to the panel using the pots' and sockets' nuts.
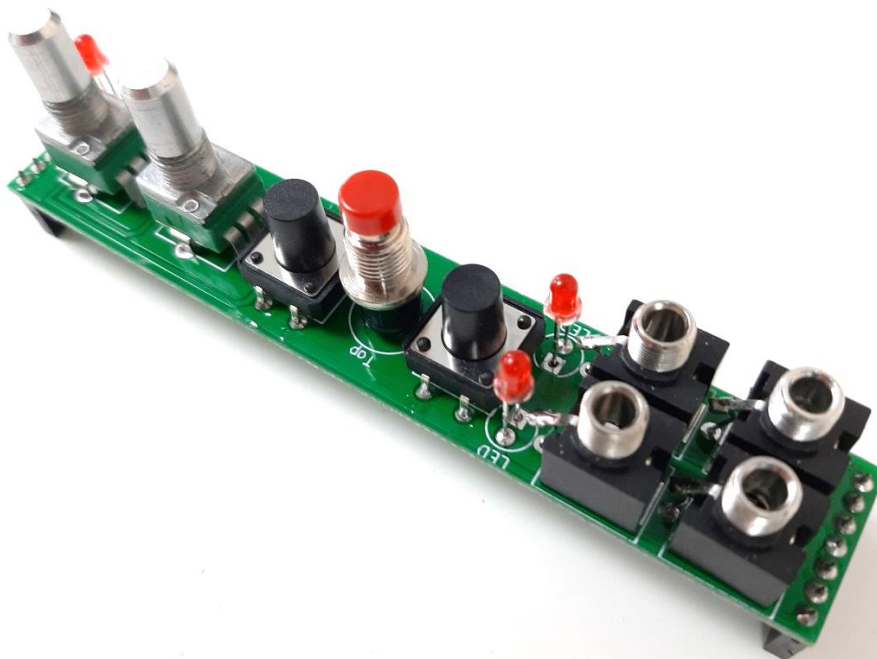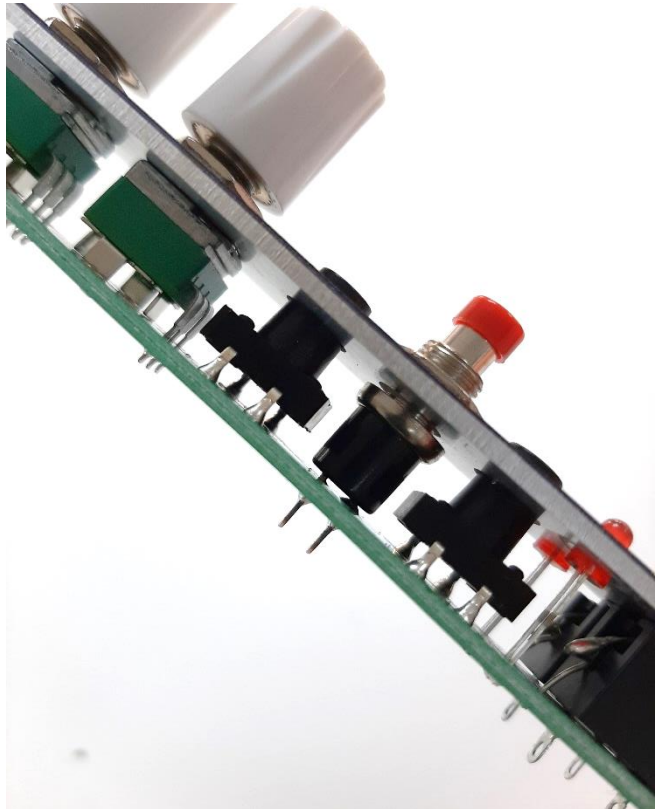
Troubleshooting

Not all DIY builds work first time. The vast majority of build issues are down to soldering inconsistencies. This is far more likely than a bad IC, for example. The first step of successful troubleshooting should always be to reflow all soldering to eliminate any dry joints (bad connections) or solder bridges (short circuits). This is also an opportunity to closely inspect your work – you might find some unsoldered pads, or an IC not inserted into its socket, for example. Next steps are to double check all resistor values are correct, and to check polarities of all diodes, transistors, ICs and electrolytic capacitors. This is not an exhaustive troubleshooting guide, but should address 95% of build issues.

RDH 11/02/21

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