FREQUENCY CENTRAL

Build documentation for:

BOOM!!

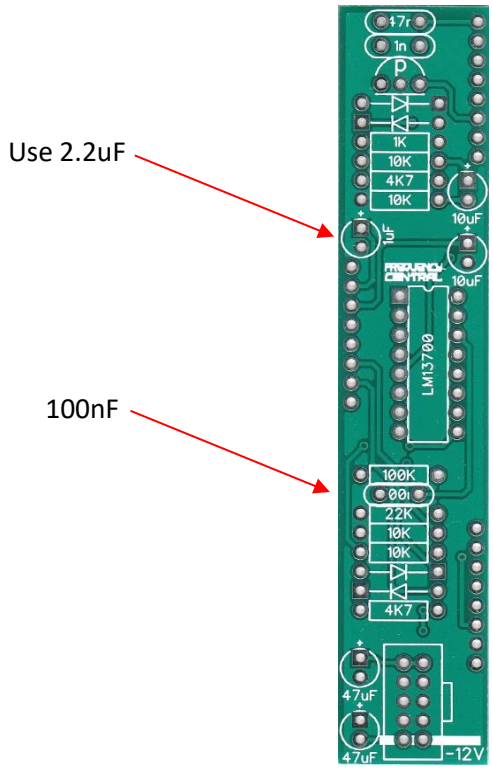
Boom!! is an analogue drum voice with a broad range of applications.

The heart of the module is a VCO which is built around one half of a LM13700. The VCO has a HIGH/LOW switch as well as a SWEEP control, which defines it's response to the onboard envelope.

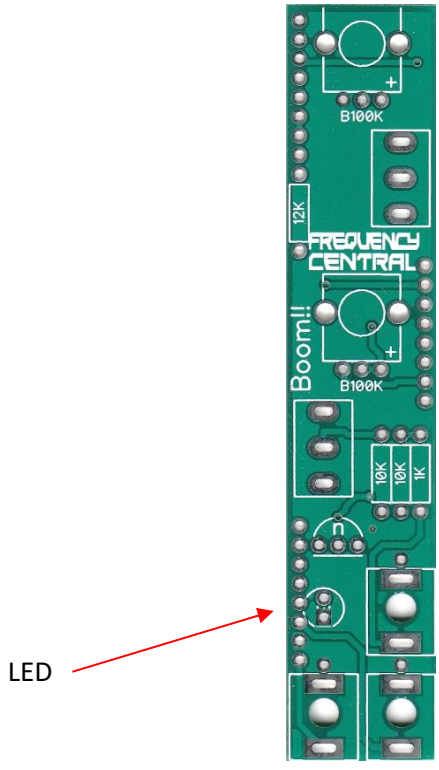
The other half of the LM13700 has been carefully crafted into a release envelope generator, which has a FAST/SLOW switch in addition to a RELEASE control. As a bonus, the input of the envelope is velocity sensitive, allowing for a dynamic response.

Additionally, there is an envelope output, which can be used to drive external modules.

Main PCB



Control PCB



<u>Key to PCB screen print:</u>

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

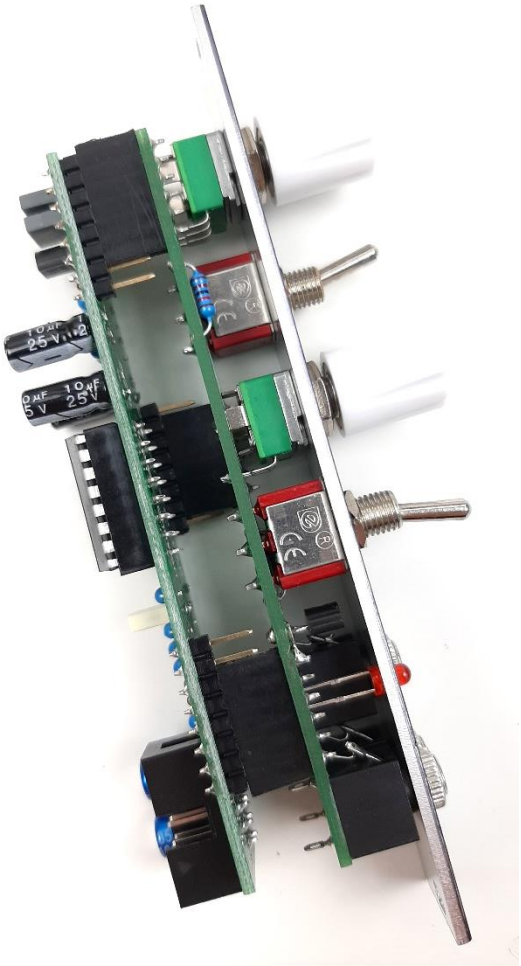
p: This signifies PNP BC557 transistors. Note the correct pinout as shown by the half circles.

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

Bill of Materials

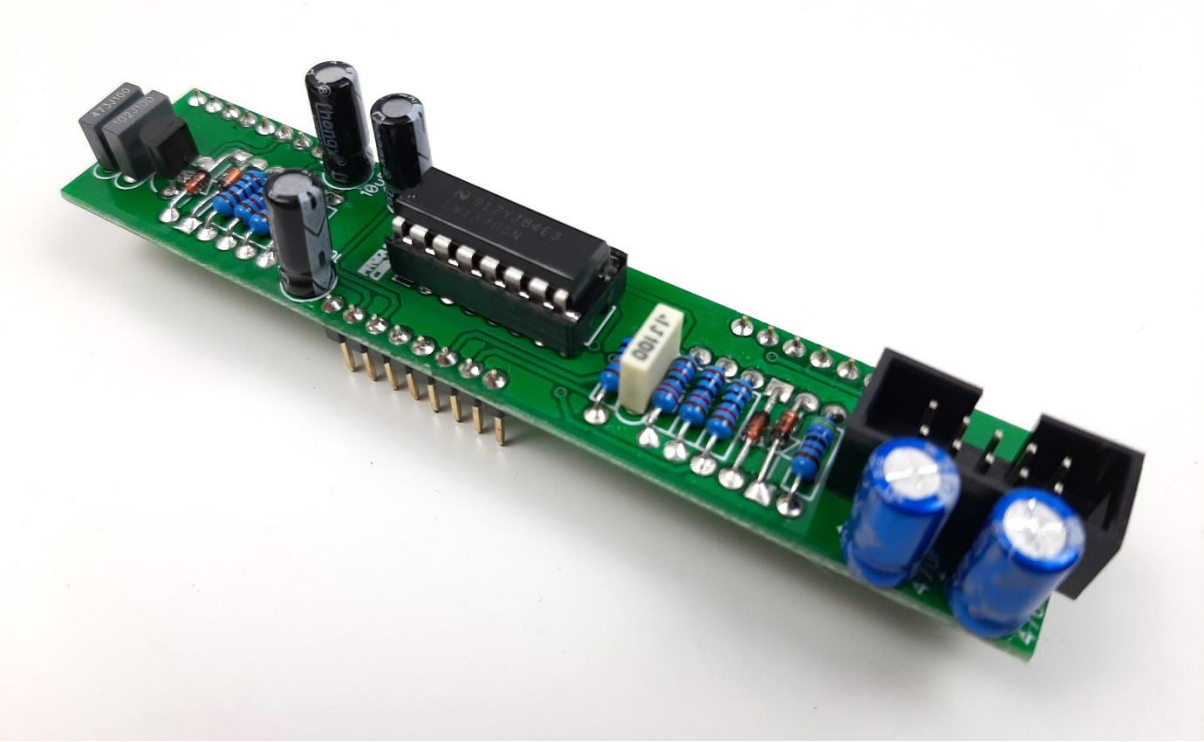
You will notice that all of the components listed below are also hyperlinks to where I buy each specific part from. You can also use the hyperlinks to find out more about what each component looks like. If you want to know even more, Google is your friend.

1K x 2 4K7 x 2 10K x 6 12K x 1 22K x 1 100K x 1 All resistors ¼ watt metal film.	1nF x 1 47nF x 1 100nF x 1 2.2uF (use instead of 1uF) x 1 10uF x 2 47uF x 2	LM13700 x 1 16 pin IC socket x 1 BC547 x 1 BC557 x 1 3mm red LED x 1 1N4148 x 4 Boom!! pcb set Boom!! panel	B100K x 2 metal shaft SPDT toggle x 2 3.5mm socket x 3 Male header x 1 (cut to size) Female header x 1 (cut to size) Power header x 1 Knobs x 2



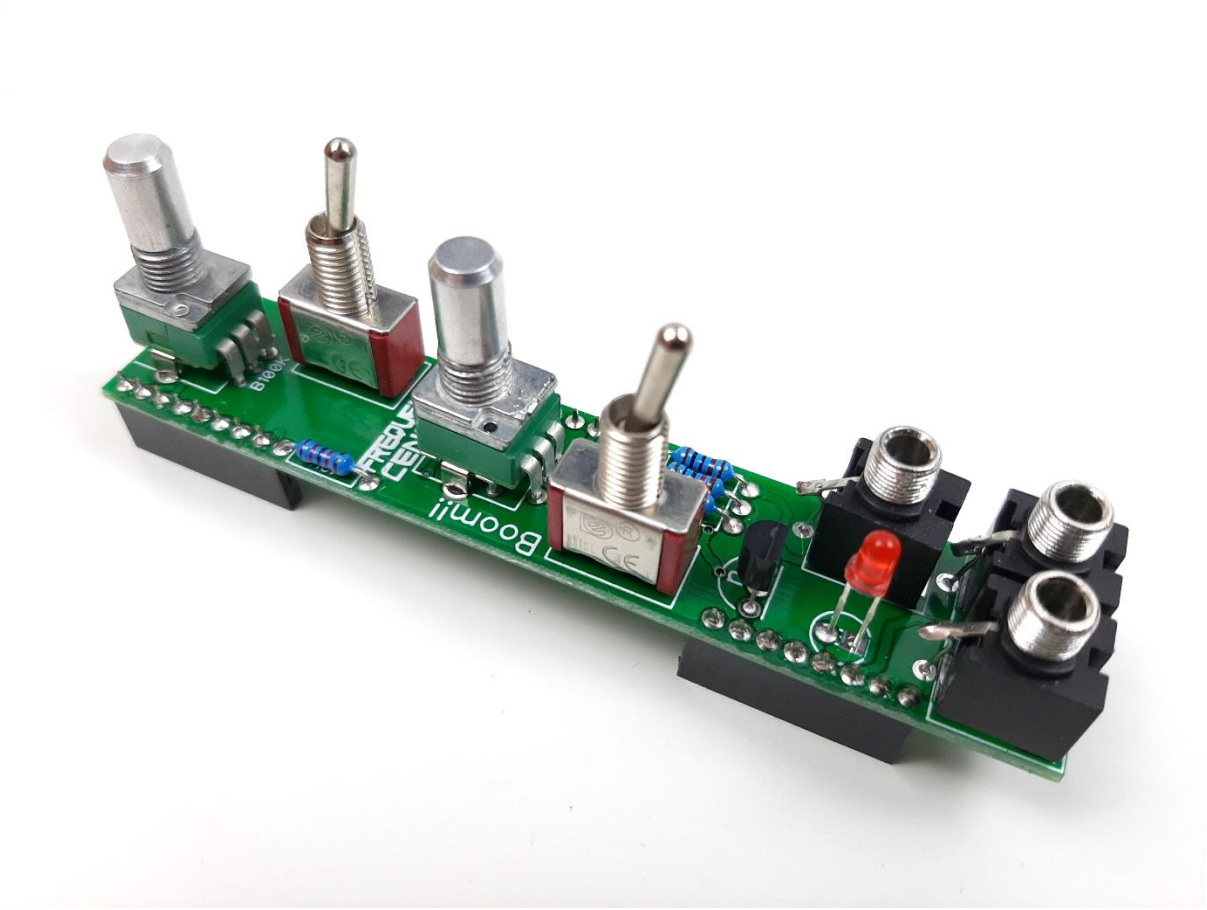
Main PCB assembly

1. Solder the diodes and all resistors. Watch the polarity of the diodes.
2. Solder the IC socket
3. Solder the non elecrolytic capacitor
4. Solder the BC557– 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.



Control PCB assembly

1. Solder the resistors
2. Solder the BC547 – watch the polarity.
3. Solder the 2 x pots.
4. Solder the 2 x switches and 3 x sockets. Use the panel to ensure these line up nicely. You can use cut off resistor legs to make the ground connections of the sockets.
5. Cut female headers to size and solder them into place. Make sure that they stick out of the bottom of the PCB.
6. Solder the LED. Use the panel to ensure that these line up nicely



Bolt the pots, switches and the sockets to the panel using their nuts and washers. Looks nice huh?

Calibration

1. There is nothing to calibrate on this module – cool!
2. Check it out now funk soul brother.

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 29/10/20

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