BARTON MUSICAL CIRCUITS

BMC104 Parallel Melody Generator

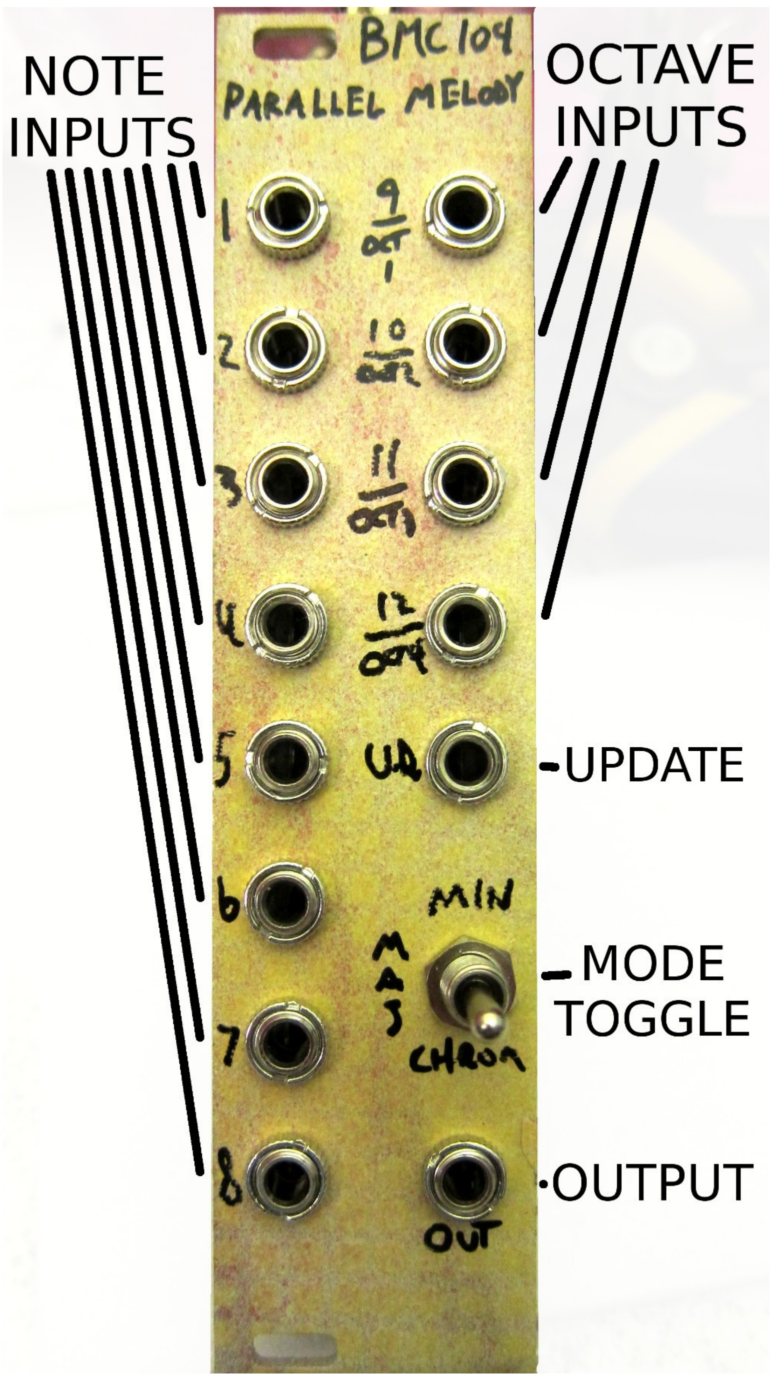
Last updated 7-9-2023

I. Features

The Parallel Melody Generator is a module that creates CV sequences from gate/trigger sequences. Eight note inputs correspond to the eight notes in a scale. As gates are applied to the note inputs, the output CV will change to the note corresponding to the last gate input. Separately, four inputs control the octave of the CV output.

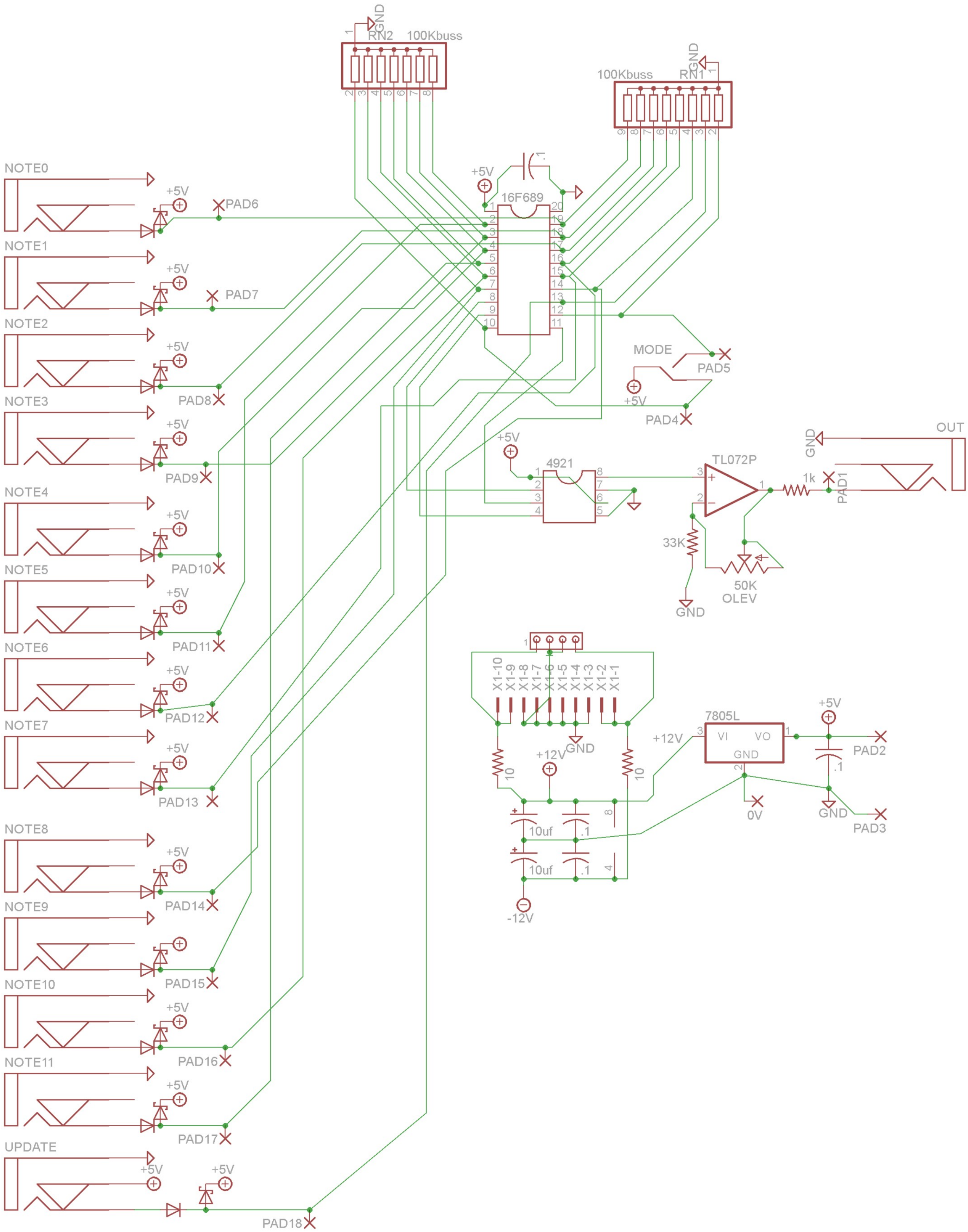
A toggle switch selects between Major, Minor and Chromatic scales. In chromatic mode, the range is limited to a single octave, and the octave inputs correspond to the 4 additional notes in a chromatic scale.

An “update” input controls when the output CV is updated. When no input is patched into this jack, the output will update continuously.

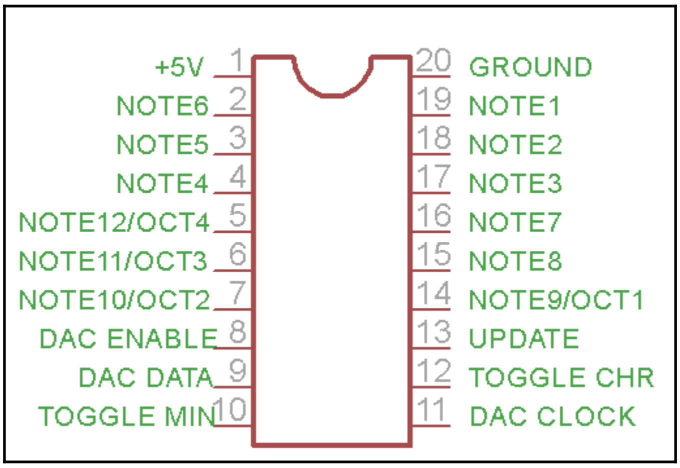


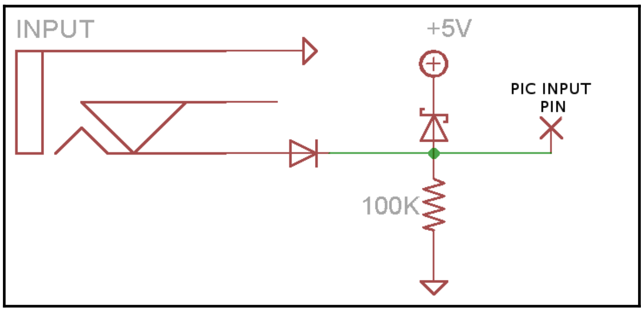
II Schematics

Below is the master schematic for the module, and on the next page is an explanation of the different parts of the circuit. The circuit is split into two circuit boards, but this schematic shows both together. The “PAD1” through “PAD18” symbols mark the connectors that join the panel PCB with the jacks and toggle to the top PCB with the ICs and power connector.



To the right is a pinout diagram for the 16F689 PIC microcontroller that is at the center of this design. Pins 2-7, 10 and 12-19 are all digital input pins. The schematic for a digital input is shown below. Pins 8-9 and 11 are the output pins that go to the Digital to Analog conversion circuit, also shown below. Power connections are made at pins 1 and 20.





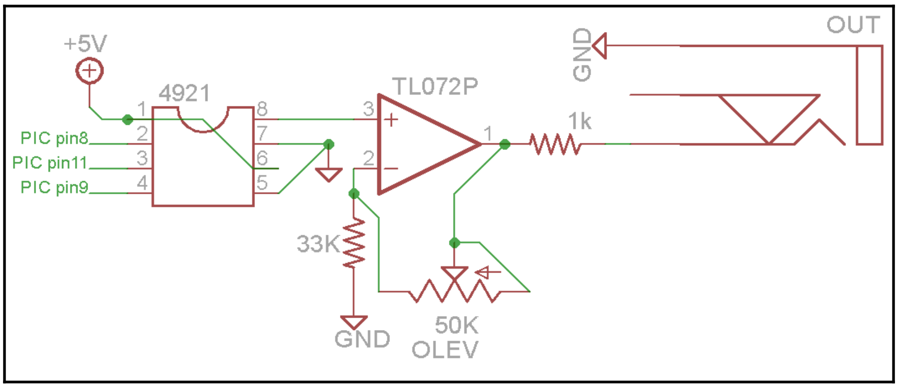
To the left is a generic input schematic. The tip of the input jack connects to the anode of a diode which provides polarity protection for the PIC only allowing positive voltages to pass.

The cathode connects to the pin of the PIC, a 100K pull down resistor that grounds the pin when no input is present and a schottky diode connected to +5V. This schottky provides extra protection from voltages over +5V from damaging the PIC. If your system only uses gates/triggers of +5V, the schottky can be omitted or replaced with a regular switching diode. The 100K resistor is shown as a single resistor in this schematic, but is in actuality a part of a bussed array, meaning many resistors that all have one end tied together to a common source, which is ground in our case.

The UPDATE input jack is the only jack that is different, as it's switch is connected to +5V to provide a continious +5V signal when nothing is plugged in.

The input pins for the toggle switch connect to the two outside lugs of the toggle and the center lug is connected to +5V. The toggle is on-off-on type, so it can provide voltage to either input pin or to neither.

To the right is the output circuit. The MCP4921 is our digital to analog conversion chip. It receives data on input pins 2, 3 and 4 from the PIC. The +5V and ground are connected to the power supply and voltage reference pins. Pin 8 goes to the positive input of an op-amp. The gain of this amplifier is controlled by a variable voltage divider in the feedback path from it's output to negative input. The output goes through a 1K resistor to the output jack.



III. Construction

Parts List

Semiconductors

Value	Qty	Notes
16F689 PIC	1	Came with your PCB
MCP4921	1	8 pin DIP package
TL072	1	8 pin DIP package, other dual op-amps should work fine
78L05	1	TO-92 package +5V voltage regulator
1N4148	13	Or other small signal switching diode.
1N60P	13	Or other small schottky. If using a system with gate/triggers never exceeding +5V can be omitted or replaced with 1N4148

Resistors

Value	Qty	Notes
10 ohm	2	1/4W Metal Film
1K	1	1/4W Metal Film
33K	1	1/4W Metal Film
100K 8 pin bussed array	1	Or if making your own arrays, you'll need 15 100K resistors.
100K 9 pin bussed array	1	
50K Cermet pot	1	3296W package or similar

Capacitors

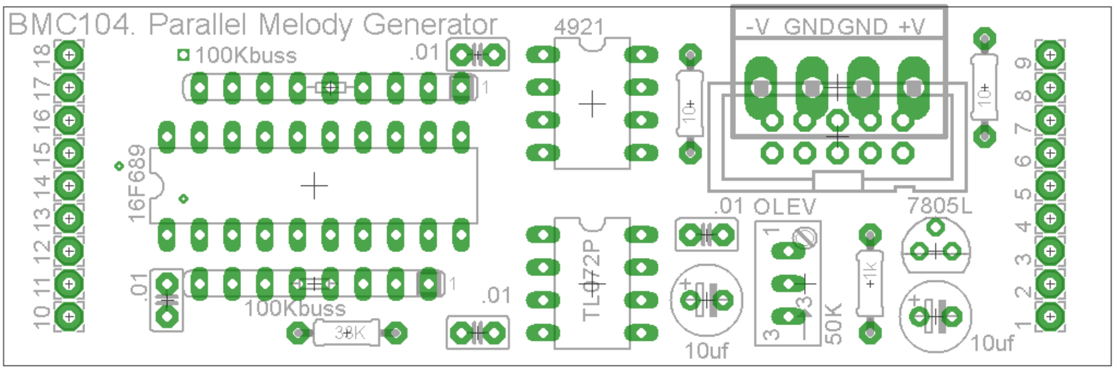
Value	Qty	Notes
10uf	2	Electorlytic
.01uf	4	Ceramic disc.

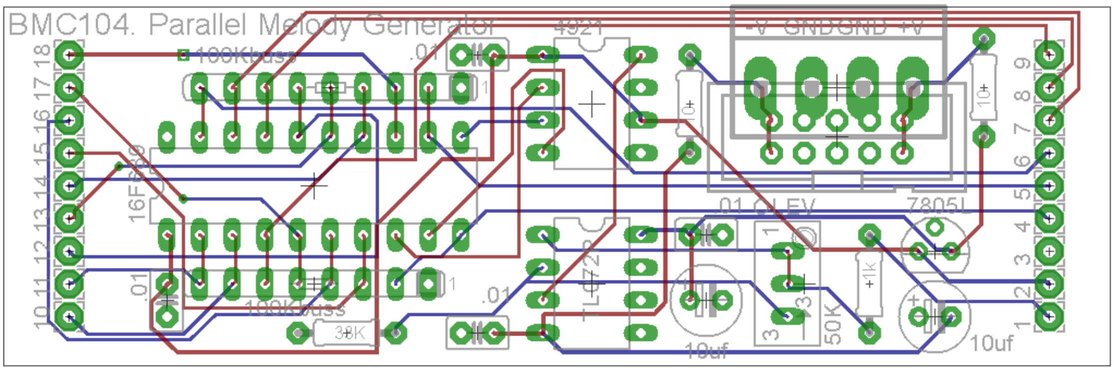
Other

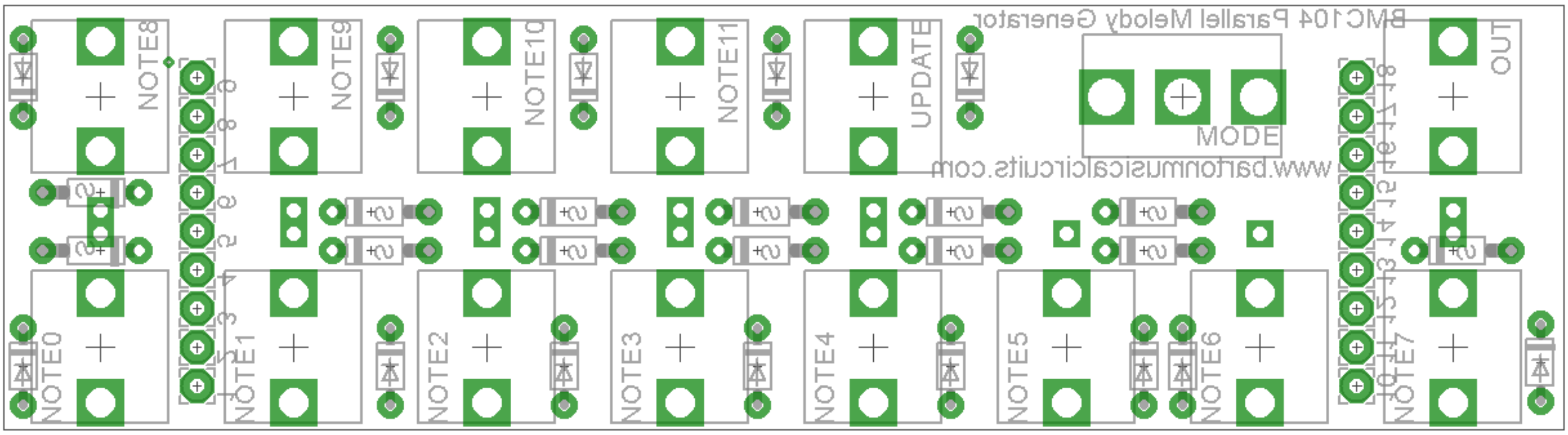
Value	Qty	Notes
Power Connecter	1	Eurorack or MOTM style
8 Pin DIP Socket	2	
20 Pin DIP socket	1	
Single Row 2.54mm connector male	18	Buy a 40 pin header like this and break it down
Singe row 2.54mm connector female	18	Buy a 40 pin header like this and break it down.
Switching Input Jack	14	Designed for these.

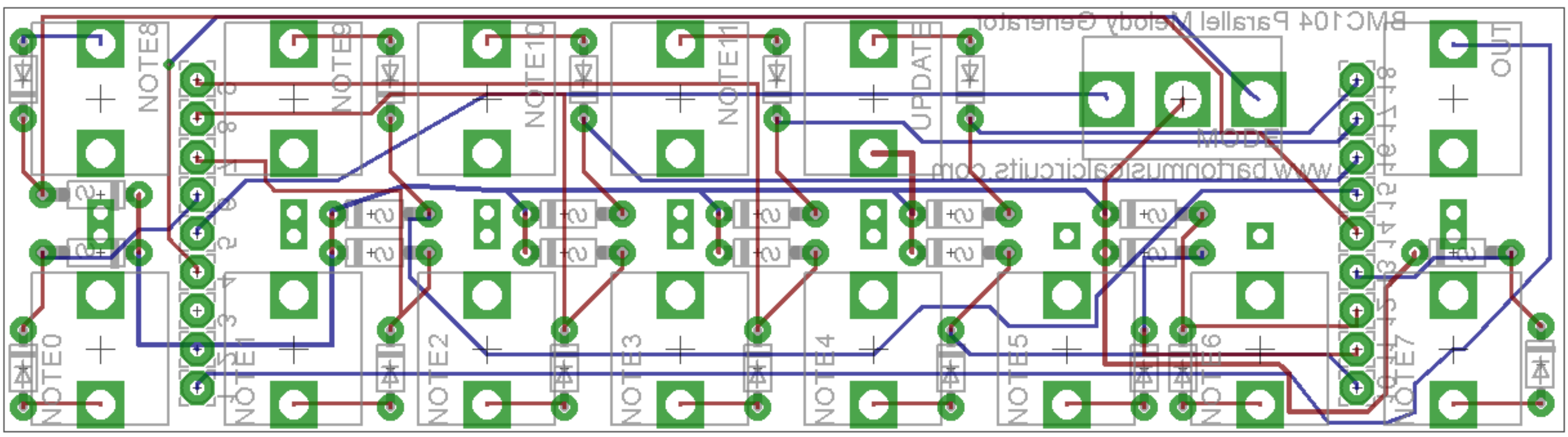
The Board

The panel board's dimensions are 28mm x 100mm and the top board's dimensions are 28mm x 89mm. Below are renderings of the PCBs with and without traces.



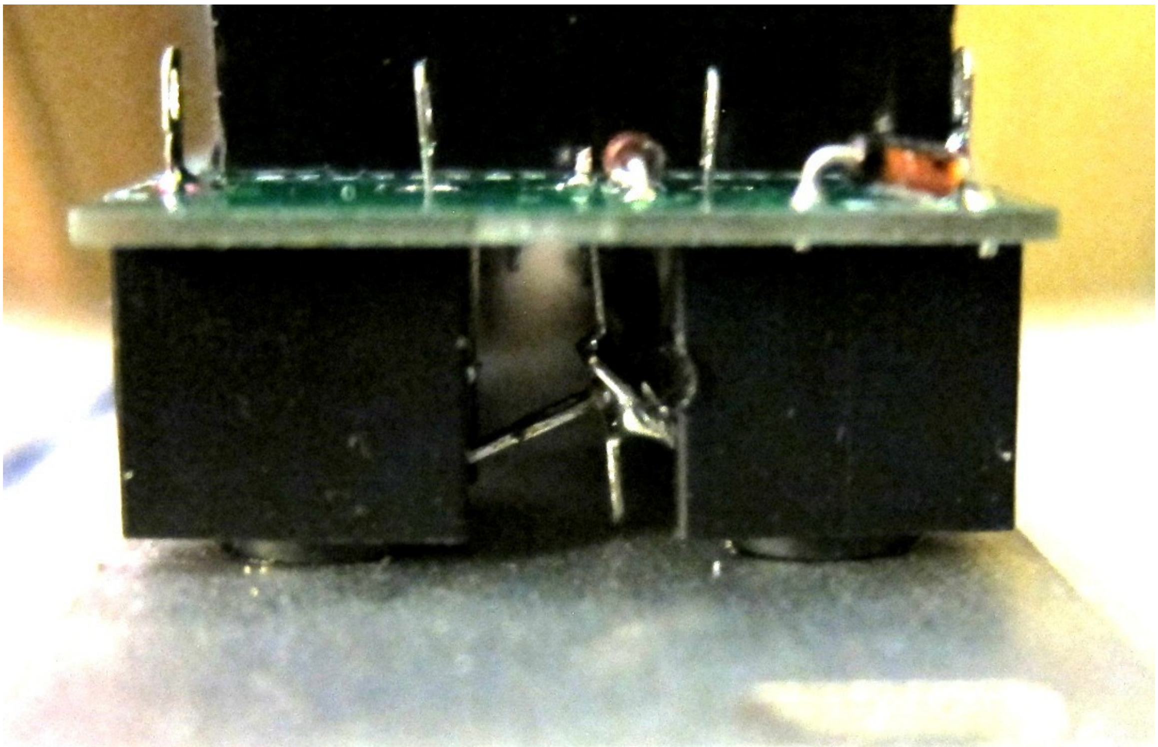






Build Order
I suggest stuffing the PCBs in the following order:
TOP PCB
1.Individual Resistors
2.DIP sockets
3.Ceramic capacitors
4.7805 regulator
5.Resistor Arrays
6. Power Header
7.Cermet Pot
8.Electrolytic capacitors
9.Single row headers to connect to other PCB (install on the bottom of the PCB)
BOTTOM PCB
1.Diodes
2.Single row headers to connect to other PCB.
3.Jacks (just solder tip connectors at first, solder other connections after PCB is secured to panel)
4.Toggle switch (just solder one lug, solder other connections after PCB is secured to panel)

If you wish to ground your jacks/panel, use a resistor lead as a jumper and solder a connection from a jack's sleeve lug to one of the small ground pads. If you are using a non-conductive panel, you will need to do this for every jack.



CALIBRATION
1. Connect the red lead of your voltmeter to the tip of the output jack, or to the tip of a patch chord connected to the output jack. Connect your black lead to ground. Power up the unit
2. Apply a gate to the Note 1 input and then to the Octave 1/Note9 input. The output voltage should now be at 0V.
3. Apply a gate to the Octave 2/Note 10 input, adjust the trimpot until the output voltage is at 1V.
4. Apply a gate to the Octave 4/Note 11 input, adjust the trimpot until the output voltage is at 3V.
5. Go back and forth between octave inputs and adjust until you get as close to 1V jumps between octaves as you can.