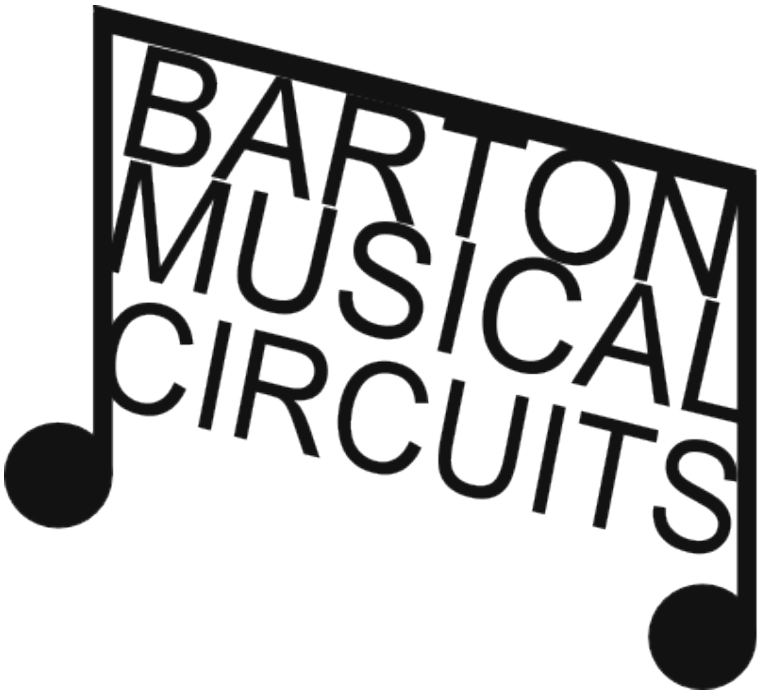
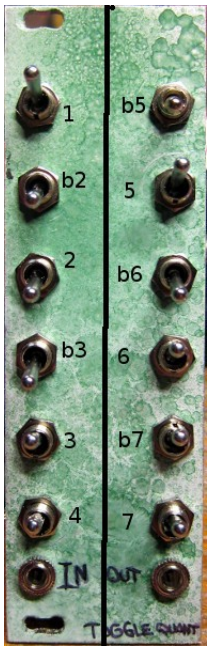


BMC060. Toggle Quantizer.

Last updated 5-20-20

If you have any questions, or need help trouble shooting, please e-mail Michael@Bartonmusicalcircuits.com

I. Overview/Features



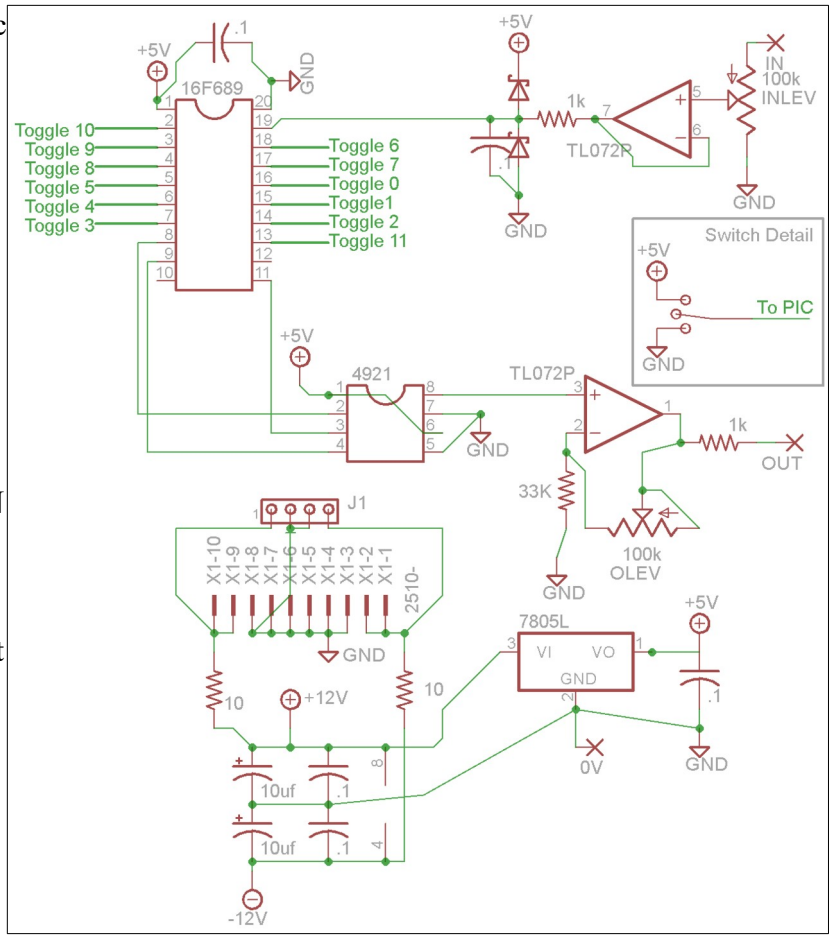
The Toggle Quantizer is a single channel quantizer with a range of 0 to 10Vs. Each volt is divided into 12 equally spaced voltages, to conform to the 12 half-steps of a musical octave. An input voltage will be quantized to the closest of these 12 steps that's possible.

The only controls on the module are 12 toggle switches, these set whether a particular half-step in the octave is a possible quantized output. This lets you control the musical scale by turning notes in the scale on or off. On the right is an image of the panel with the corresponding notes for each toggle labeled.

The module has only a single voltage input and voltage output.

This module uses an on-board voltage regulator and so it can be used with both +/-12V and +/-15V systems without any modification. The toggle switches are mounted directly to the circuit board, so only three wires need to be connected off the PCB, and the connections are short enough that cut resistor leads can be substituted for wires. The module does use trimmers to calibrate the input and output, so you'll need a voltmeter to calibrate this project.

II. Schematic.



To the right is the schematic for this module. To increase the legibility, only one toggle switch is shown. All twelve toggles work identically, switching between ground and +5V

In the upper right of the schematic is the 16F689 PIC microcontroller. It's powered by the +5V supply with a .1uf decoupling capacitor. Twelve of its pins connect to toggle switches.

Pin 19 of the PIC connects to the voltage input circuit. The IN wirepad connects the input jack to the 100K trimmer pot labeled "INLEV." This pot attenuates the input voltage. The wiper of the pot connects to an op-amp wired as a unity gain buffer. A 1K resistor connects the output of the buffer to PIN 19. Parallel to pin 19 are a pair of schottky diodes connected to +5V and ground, these prevent voltages outside of the PICs input voltage range from damaging the PIC. The 0.1uf capacitor parallel with pin 19 acts as a low pass filter to minimize jitter on the quantizer.

Pins 8, 9 and 11 connect to the MCP4921 digital to analog (DAC) converter chip. These pins send instructions in digital data packets which the 4921 converts to an analog voltage on pin 8. This voltage is then amplified by the other half of the TL072, with the 33K resistor to ground and the 100k trimmer marked "OLEV" setting the gain. A 1K resistor connects the output of the

amplifier to the OUT wirepad and on to the output jack.

At the bottom of the schematic we see the power supply connections. PCB footprints for Eurorack and MOTM power connectors are connected in parallel. The positive and negative rails are filtered by a 10ohm/10uf passive low pass filter and connected to the power pins of the TL072. 0.1uf capacitors are attached at the power pins of the TL072 to keep the power supply as clean as possible. The input of the 7805L voltage regulator connects to the filtered positive voltage and it's output creates the +5V supply.

III. Construction

A.Parts List

Semiconductors

Name	Quantity	Notes
16F689	1	Should be provided with your PCB
TL072	1	8 pin DIP package
MCP4921	1	8 pin DIP package
Schottky diode	2	Any small signal schottky
7805L Regulator	1	TO 92

Resistors

Name/Value	Quantity	Notes
10 ohm	2	1/4w metal Film for all resistors unless otherwise noted
1K ohm	2	
33K ohm	1	
100K Trimpot	2	3296W package

Capacitors

Name/Value	Quantity	Notes
.01uf	5	Ceramic disc. Value not critical
10uf	2	Electrolytic, at least 16V rating.

Other

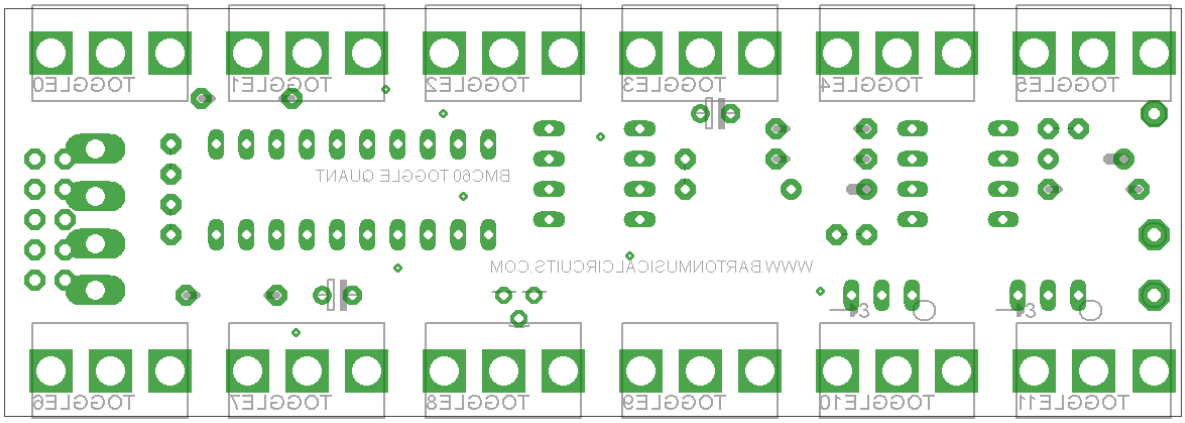
Name/Value	Quantity	Notes
Power connecter	1	Eurorack or MOTM
Toggle switch	12	SPDT. ON-ON Like this.
Mono or Switching Jack	2	
20 pin DIP socket	1	
8 pin DIP socket	2	

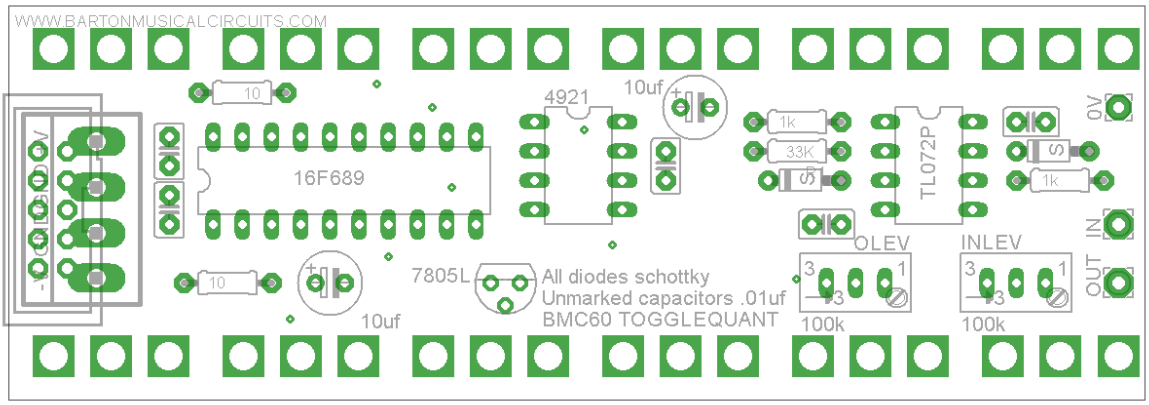
B. The PCB

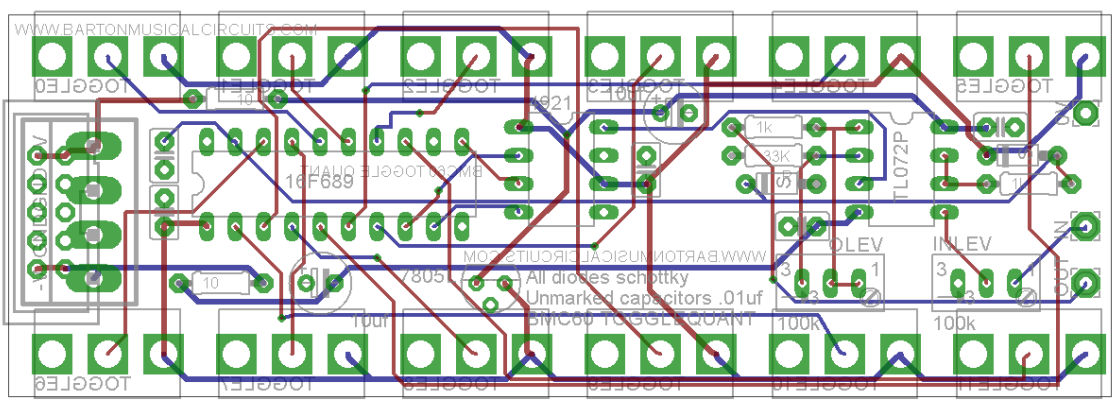
The PCB is 99mm x 35mm. It is designed to have toggle mounted on one side and all of the other components mounted to the other. The recommended order for stuffing the PCB is:
1.Resistors and diodes
2.0.1uf capacitors
3.IC sockets
4.7805 voltage regulator
5.Power connector
6.100K trimpots
7.10uf capacitors
8.Toggles
9.Offboard wiring.

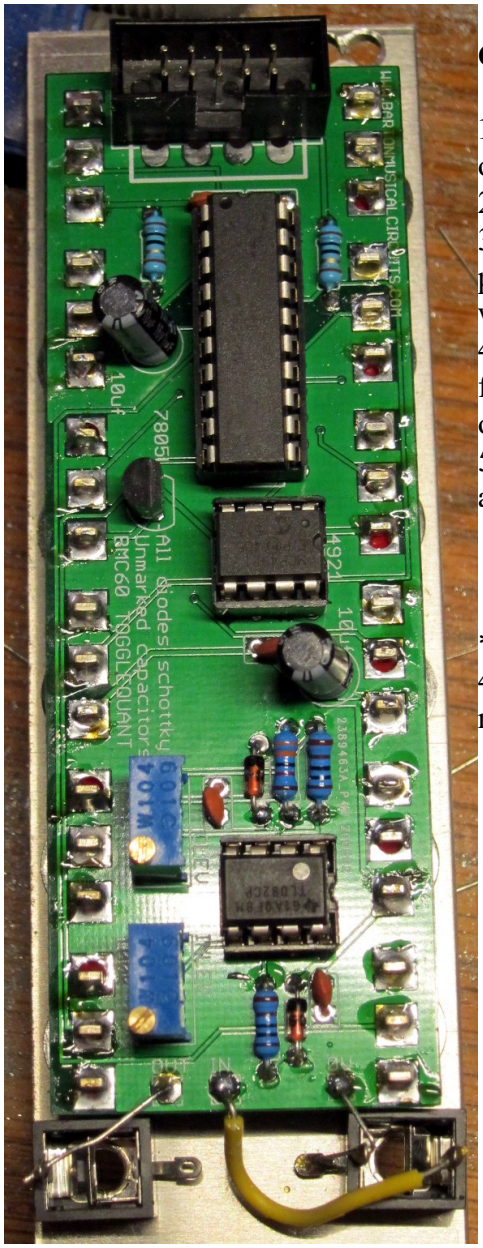
When soldering toggles onto the PCB, I recommend just soldering a single pad. This will put less strain on both the PCB and the switches when you're mounting the PCB to the panel. After the PCB is mounted to the panel, reflow the existing solder joints on the toggles, then solder the rest of the joints. In my experience this is the easiest way to get a solid module.

Below are images of both sides of the PCB without traces, and an image of the PCB traces, then photos of an actual module.









C. Callibration

1.Input a steady positive voltage source, like the voltage output of a keyboard or a PWM turned all the way up.
2.Measure the voltage on the IN wirepad
3.Measure the voltage on pin 19 of the PIC. Adjust the INLEV pot until the voltage on pin 19 is half of the voltage on the IN wirepad*
4.Input a voltage that can be varied by small degrees. Set the first (top left) toggle on and the rest off. This will make only one output per octave.
5.Increase your input voltage until you get a change in output, adjust OUTLEV so the change is exactly 1V.

*For maximum accuracy, instead make the voltage on pin 19 46% of the input voltage. The difference is hard to notice in most applications.