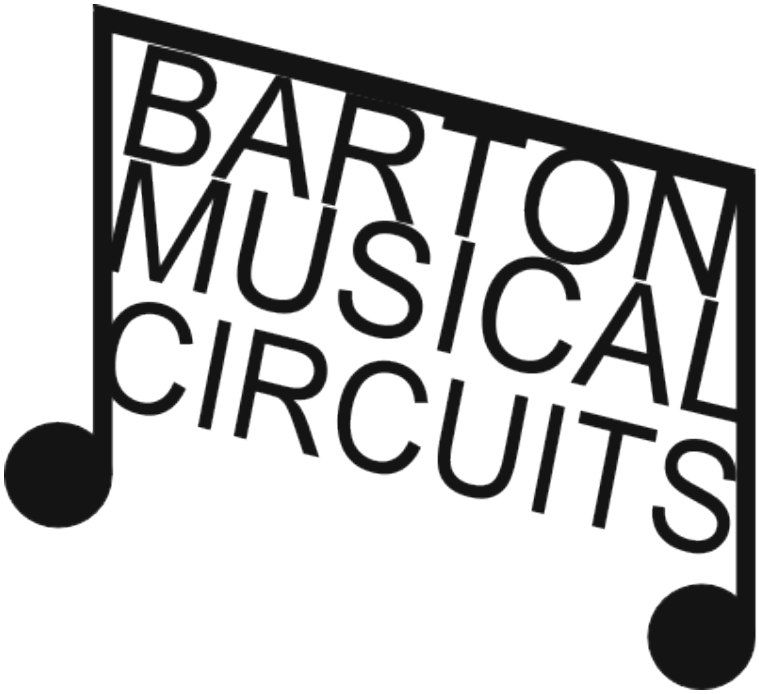



# BMC24. MIDI TO GATE CONVERTER DOCUMENTATION.

**This documentation is for use with the "Euro Style" bottom board.**



*Revision 1. 6-25-14*

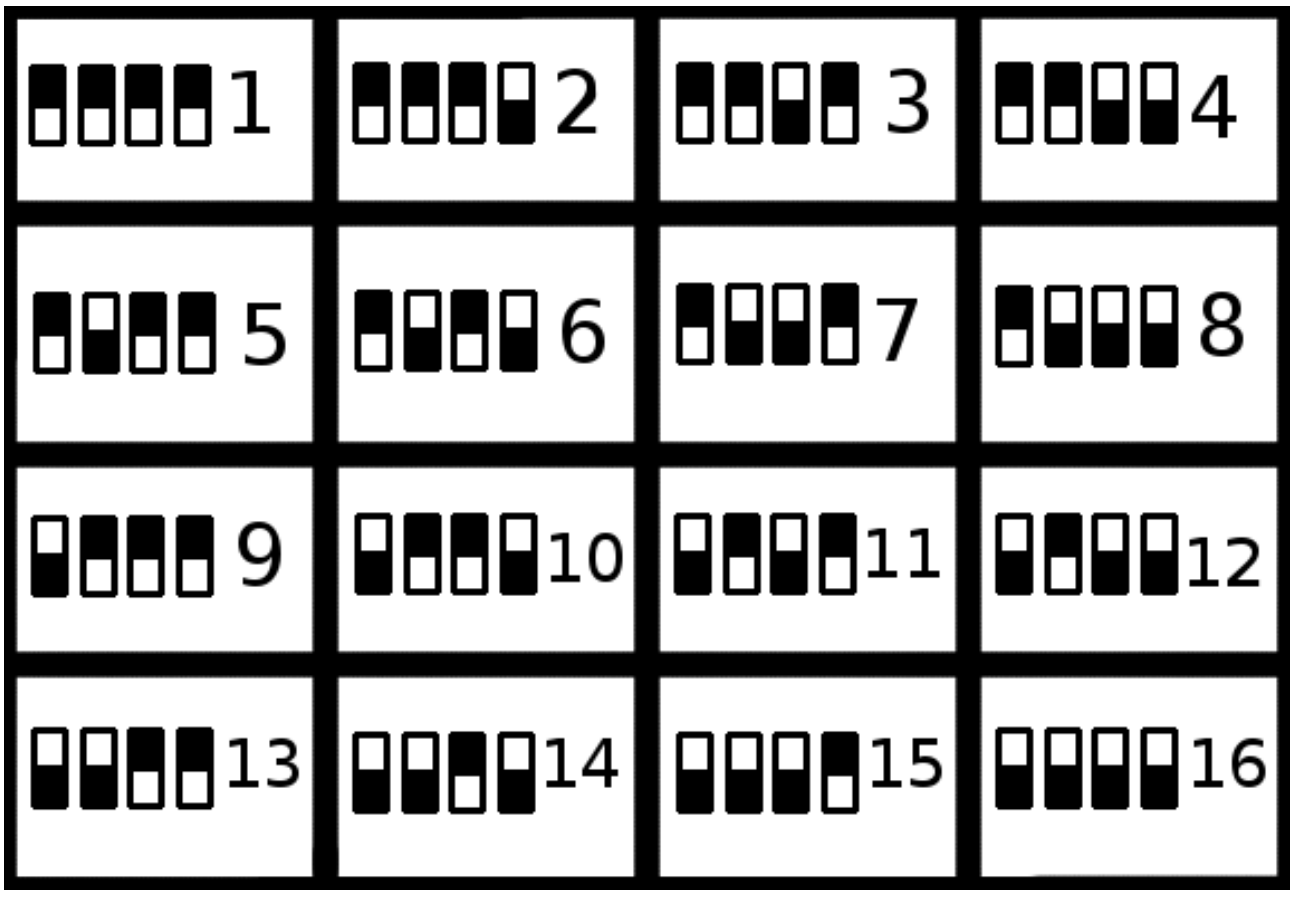
## A. USING THE MIDI TO GATE CONVERTER

*The Midi to Gate converter (which will be referred to as M2G) is a very simple to use module. It has 8 outputs which correspond to notes C1 (note #36) through G1 (note #45) when transmitted from a MIDI device like a drum machine, keyboard, sequencer or DAW. It has only three controls:*

*-A toggle switch which selects between gate or trigger outputs. This affects all outputs simultaneously.*

*-A master reset pushbutton. I've had MIDI equipment in the past that locks up, and this is an easier solution than powering down your entire rack.*

*-An on-board DIP switch for selecting which MIDI channel it should listen to. The chart on the bottom of the page explains which settings of the switches corresponds to which channels. In the chart, the switch is rendered as if you were looking at the top of the PCB with the power connecter on your right.*



## B. Parts List

### Semiconductors

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser Part* |
|---|---|---|---|---|
| *16F628A* | *1* | *Comes with PCBs* | *n/a* | *n/a* |
| *TL064* | *2* | *14pin DIP, TL074 works too* | *A-1134* | *595-TL064INE4* |
| *6N137* | *1* | *8pin DIP* | *A-871* | *859-6N137M* |
| *7805* | *1* | *TO-220 Package* | *A-179* | *863-NCP7805TG* |
| *1N4148* | *1* | | *A-157* | *583-1N4148-T* |
| *Red LED* | *8* | *3mm package* | *A-261* | *859-LTL-4221N* |

### Resistors

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser* |
|---|---|---|---|---|
| *10 ohm* | *2* | *1/4w metal film* | *A-2198* | *660-MF1/4DCT52R10R0F* |
| *200 ohm* | *1* | *" "* | *A-2620* | *660-MF1/4DCT52A2000F* |
| *100K ohm* | *3* | *" "* | *A-2248* | *660-MF1/4DCT52A2000F* |
| *1K ohm* | *17* | *" "* | *A-2200* | *660-MF1/4DC1001F* |

### Capacitors

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser* |
|---|---|---|---|---|
| *10uf* | *3* | *Electrolytic* | *A-4534* | *140-REA100M1HBK0511P* |
| *.01uf* | *5* | *Ceramic Disc* | *A-550* | *594-D103M29Z5UH6UJ5R* |

### Connecters

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser Part* |
|---|---|---|---|---|
| *Power Connecter* | *1* | *Tayda part requires modification* | *A-198* | *571-1-1634688-0* |
| *14 pin male header* | *1* | *Don't need if using Tayda power** | *A-198* | *649-54102-T30-07* |
| *14 pin female header* | *1* | *Tayda part requires modification* | *A-195* | *571-215309-7* |
| *3.5mm Jack* | *8* | | *A-865* | *16PJ138* |
| *Midi Jack* | *1* | | *n/a* | *568-NYS325* |

### Switches

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser Part* |
|---|---|---|---|---|
| *Dip Switch* | *1* | *8 pin* | *A-5018* | *774-2084* |

| | | | | |
|---|---|---|---|---|
| *SPDT Toggle* | *1* | *Panel Mount mini-toggle* | *A-4567* | *108-1MS1T2B3M2QE-EVX* |
| *Pushbutton Switch* | *1* | *Panel mount* | *A-5051* | *706-30-601RED* |

## Hardware

| *Value* | *Qty* | *Notes* | *Tayda Part* | *Mouser Part* |
|---|---|---|---|---|
| *#6-32 Screw 1/4" length* | *2* | *For Midi jack.*** | *n/a* | *608-R6-32X1/4* |
| *#6-32 Hex Nut* | *2* | *For Midi Jack.*** | *n/a* | *534-9602* |
| *#4-40 Threaded standoff 7/16" Length* | *2* | *Optional**** | *n/a* | *761-2056-440-AL-7* |
| *#4-40 3/16" Screw* | *4* | *Optional**** | *n/a* | *608-SS-0440-AC-03* |

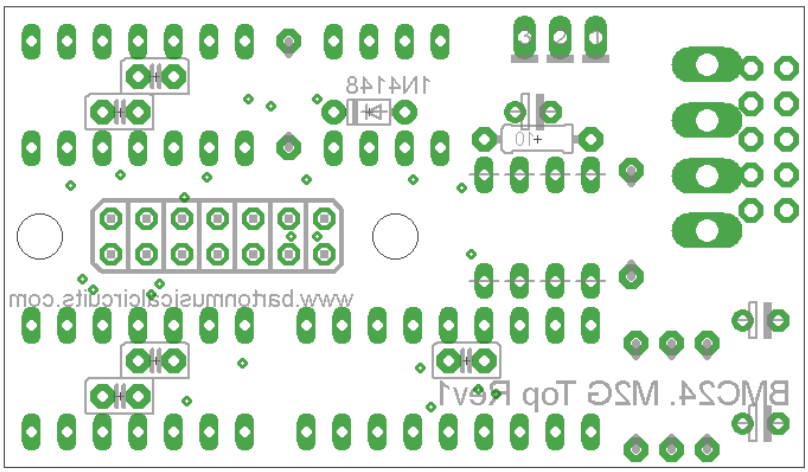
*Tayda is http://www.taydaelectronics.com/*
*Mouser is http://www.mouser.com/*
*I recommend sourcing almost everything from Tayda, they are much less expensive, and I used their parts when prototyping this project. I would suggest only getting the following parts from Mouser:*
*The Power Connecter*
*The Midi Jack*
*The Hardware(if needed)*

**The Tayda part for the power connecter is just a 40 pin dual row male header. This can easily be broken down into a power connecter and the header for the inter-pcb connection.*
***This can be replaced with any Nut and Bolt that fits the hole which you have on hand. It will almost definitely be cheaper to source locally than buying from Mouser.*
****These parts are used to secure the two PCBs together. If you only use your modular system in a home studio, or it's very rare that you move your system around, you don't need to install these. If you travel with your modular regularly, you should install them.*

## C.BUILDING INSTRUCTIONS

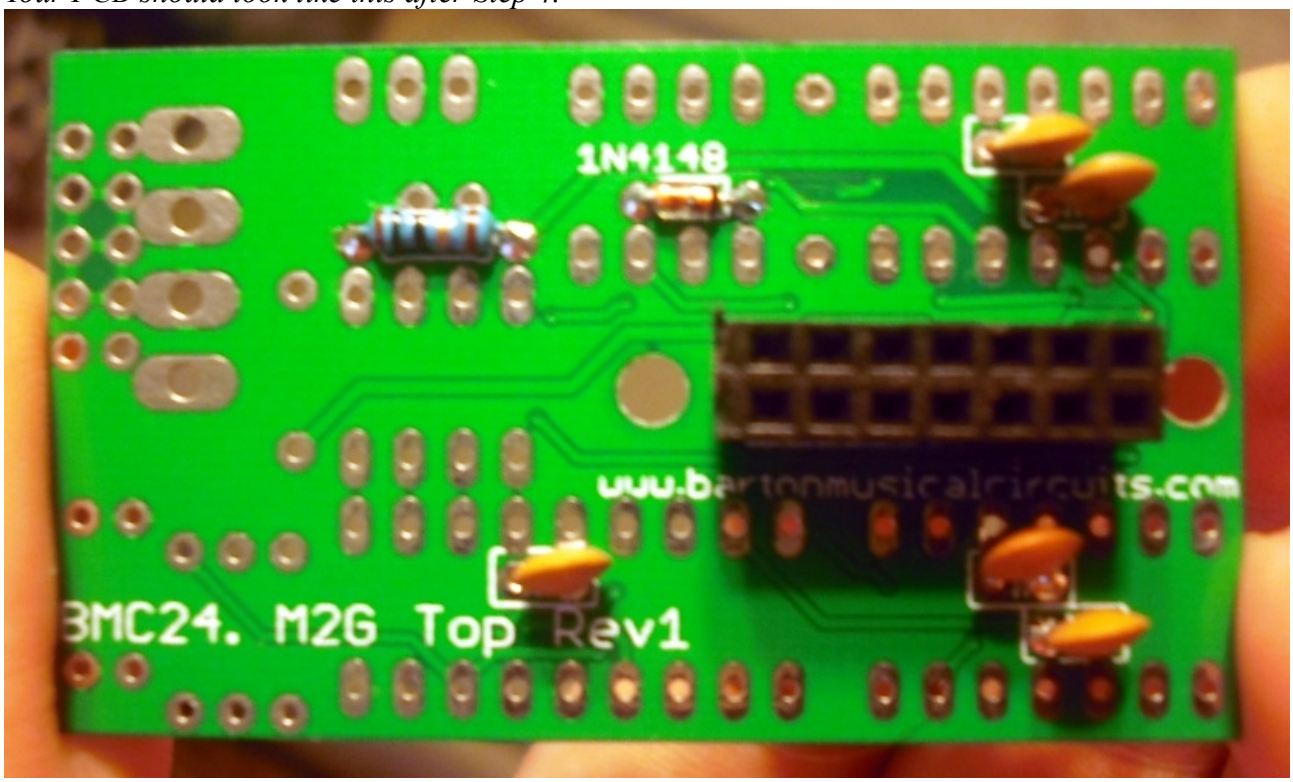
*This project is built across two PCBs, each of which has components mounted on both side. This document will help you build this project in the easiest way. These instructions assume that you already have basic experience with soldering and placing parts on a PCB.*

**SIDE 1 - TOP PCB, BOTTOM**

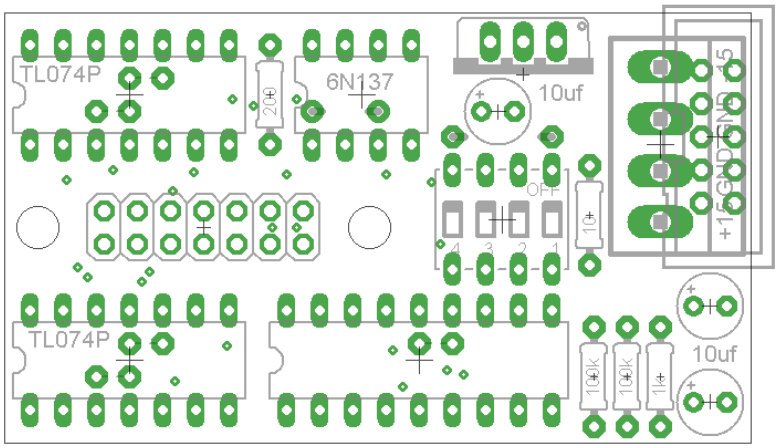


**STEP 1 -** *Stuff and solder into place the 10 ohm resistor and the 1n4148 diode. Make sure the diode is oriented correctly, with the stripe pointing away from the resistor.*
**STEP 2** - *Stuff and solder into place the 5 .01uf ceramic disc capacitors.*
**STEP 3** - *If you've purchased a 14 pin female header, skip to Step 4. Take your 20 pin header and a pair of pliers and break off the end to create a 14 pin header. You may want to score it with a blade to try and get a cleaner break.*
**STEP 4** - *Stuff and solder into place the 14 pin female header.*

*Your PCB should look like this after Step 4.*

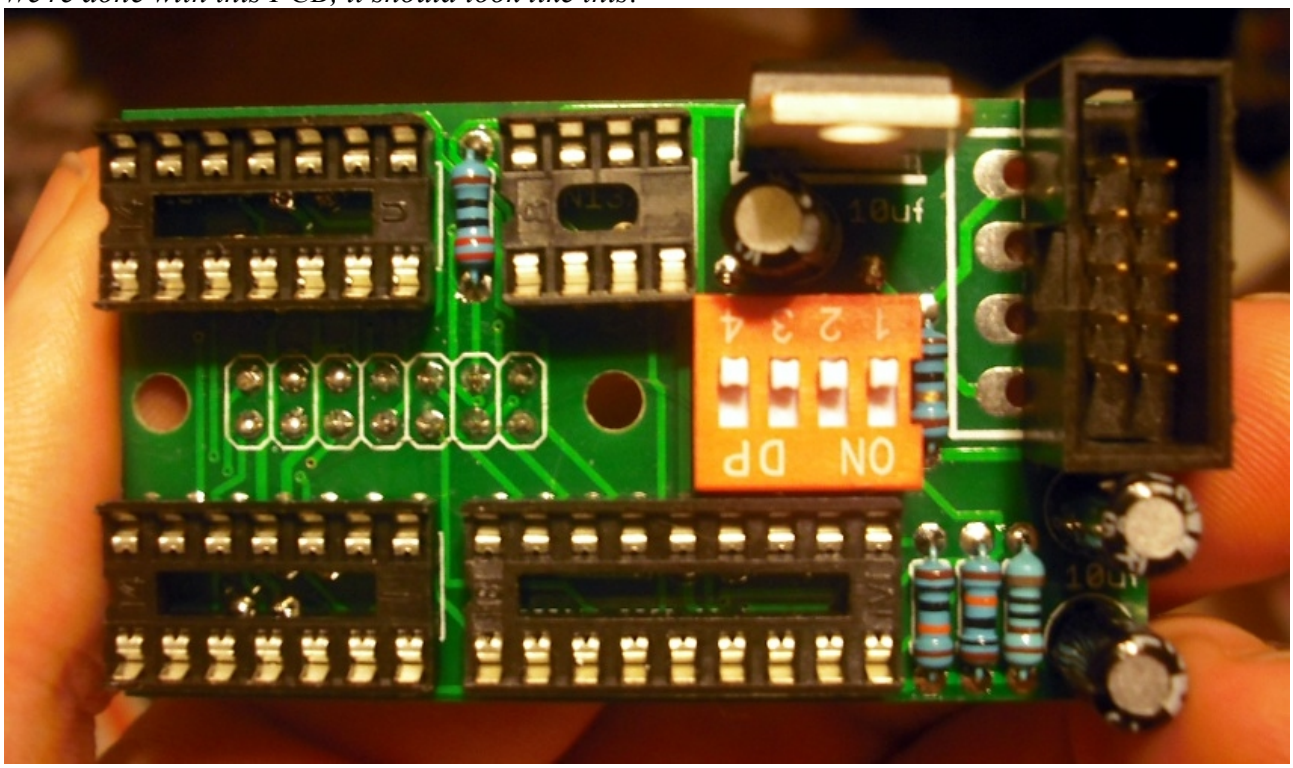


**SIDE 2 - TOP PCB, TOP**

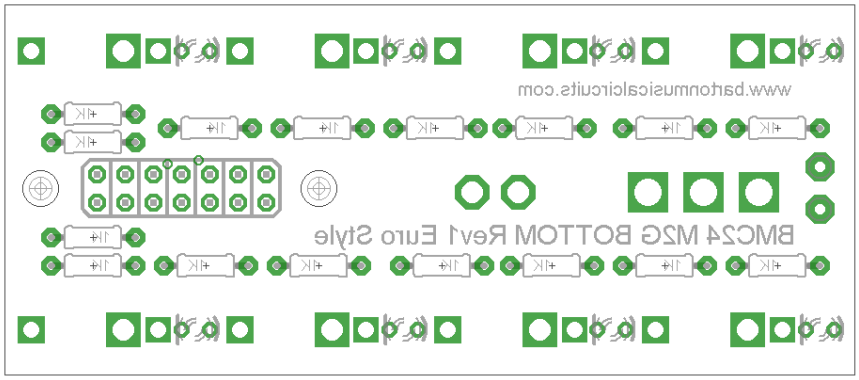



**STEP 5 -** *Stuff and solder into place all the resistors. This includes a 10 ohm, 200 ohm, 1k ohm and a 100K ohm.*
**STEP 6 -** *Stuff and solder into place all the DIP sockets for the microchips. This should be two 14 pins, an 18 pin and an 8 pin socket. All sockets should oriented with the indentation pointing away from the power connecters. We will put the chips in later, when it's time to test.*
**STEP 7 -** *Stuff and solder into place the DIP switch. Orient it so that the switches are OFF when pushed away from the 18 pin socket.*
**STEP 8 -** *Stuff and solder your Eurorack power connecter. If using a shrouded header, orient it so the indentation is pointing inwards.*
**STEP 9 -** *Stuff and solder your electrolytic caps. Make sure they are oriented so that the stripe is pointing towards the side of the board with the power connecter.*
**STEP 10 -** *Stuff and solder your 7805 power regulator. Make sure it's oriented so the tall, flat side is facing inwards.*

*We're done with this PCB, it should look like this:*

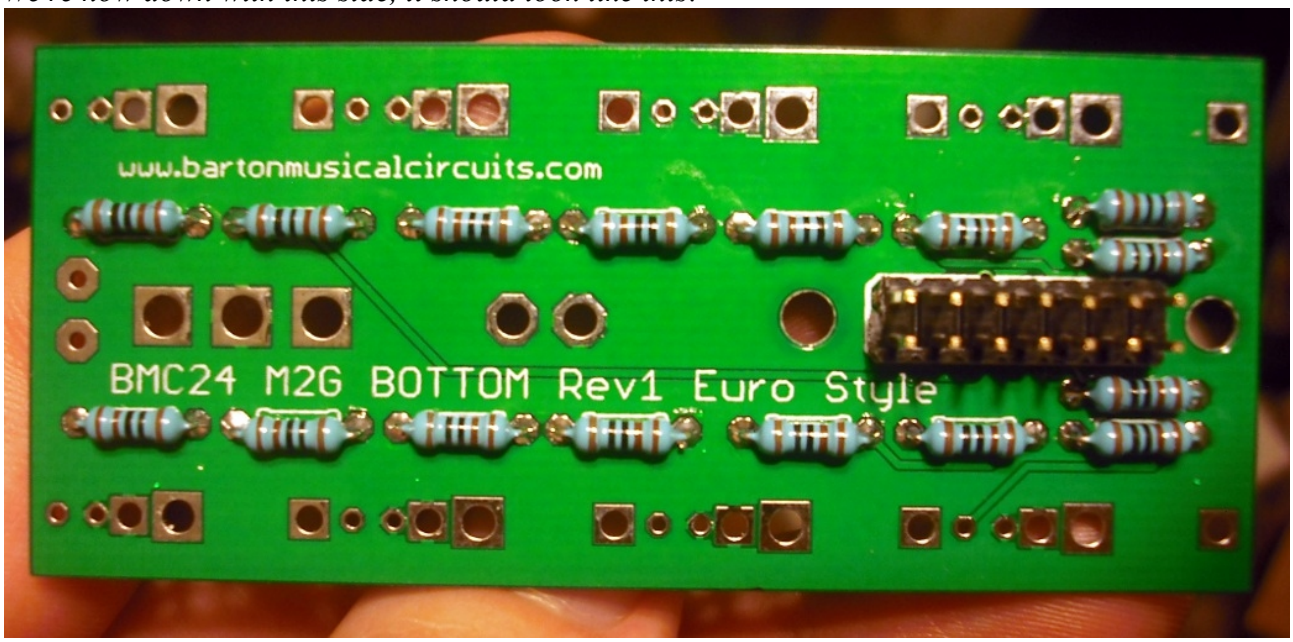


**SIDE 3 - BOTTOM PCB, TOP**

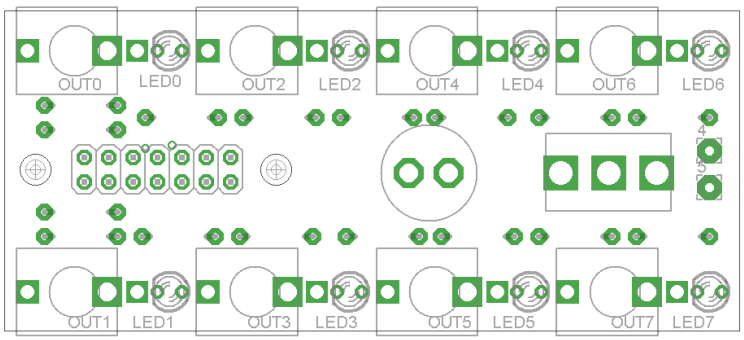


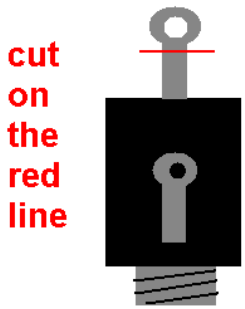
**STEP 11 -** *Stuff and solder the 1K resistors.*
**STEP 12 -** *Stuff and solder the 14pin male header.*

*We're now down with this side, it should look like this:*




**SIDE 4 - BOTTOM PCB, BOTTOM**

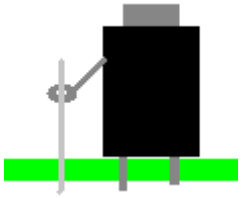



**STEP 13 -** *Prepare the 3.5mm jacks for stuffing. This means cutting the lugs off so they will fit in the PCB, like in the diagram to the right.*

**STEP 14 -** *Stuff all the sockets in place, but only solder one connecter now. After the panel is*

*mounted we will then solder the other connecters.*



**STEP 15 -** *Take a resistor lead or short piece of solid wire and connect the ground pad (the small pad between the socket and the LED) to the sleeve soldering lug on one of the jacks (see photo below). If you are using an aluminum panel, this should connect your panel to ground and thus ground all of your jacks. If using a non-conductive panel (like acrylic) repeat this for each jack.*

**STEP 16 -** *Stuff and solder one lead on each of the LEDs. They should be oriented so that the shorter lead is next to the ground pad. An easy way to make sure they are the correct height is to place the PCB on a flat with the jacks down, to make sure the LEDs are jutting out past the jacks. It's easiest to place and solder in one at a time. Like the jacks we're only soldering one of the leads so we have more flexibility when mounting the panel.*

**STEP 17 -** *Stuff and solder one lug of the pushbutton switch.*

**STEP 18 -** *Stuff and solder one lug of the toggle switch.*

**STEP 19 -** *Cut two pieces of hookup wire about 2-3 inches long and solder one end of each to the "4" and "5" wiring pads. These will connect to the MIDI connecter later.*

*Your PCBs are now completely stuffed, It should look like this after finishing Step 18.*


*Before mounting the PCB, if using a standoff for the PCBs, you need to mount the standoff now. Not everyone needs standoffs. If you don't regularly move your system around, you should be fine without them. You can skip ahead if you're just going to use this in a studio environment and want to save a little time/money.*

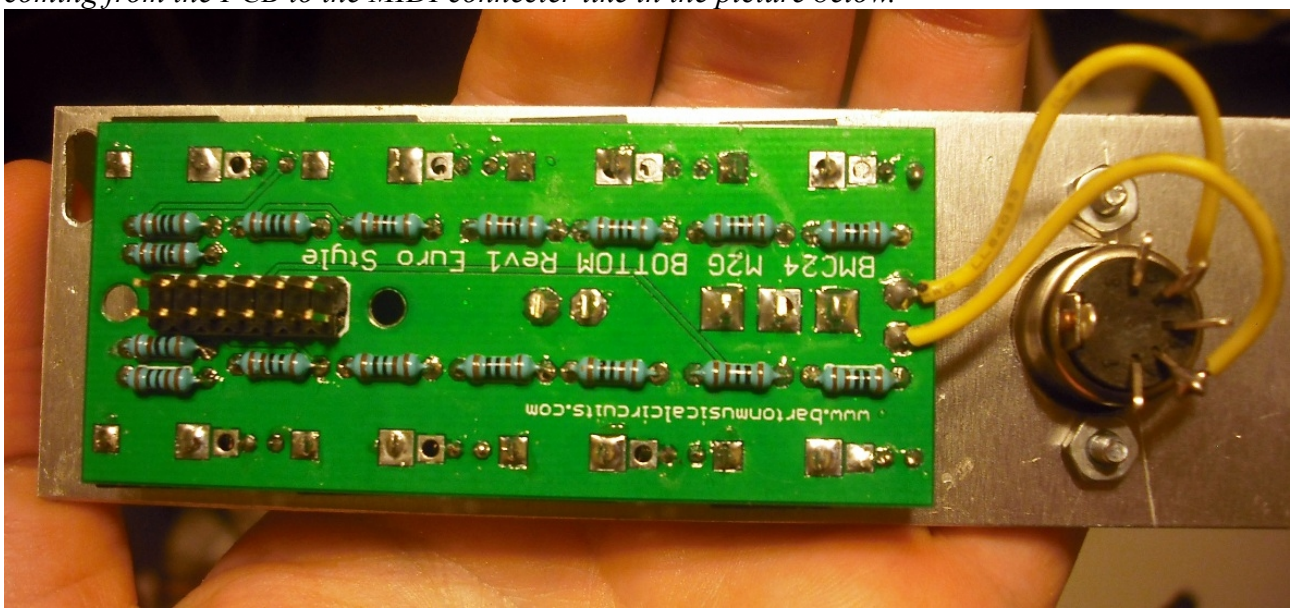
*The standoffs should be mounted on the top of the bottom board. The screws should go through the board in the holes on either side of the headers.*

**MOUNTING THE PCB TO THE PANEL**

**STEP 20 -** *Remove all nuts, washers etc from your jacks and switches. Straighten all the LEDs so that they stand straight up off the PCB.*

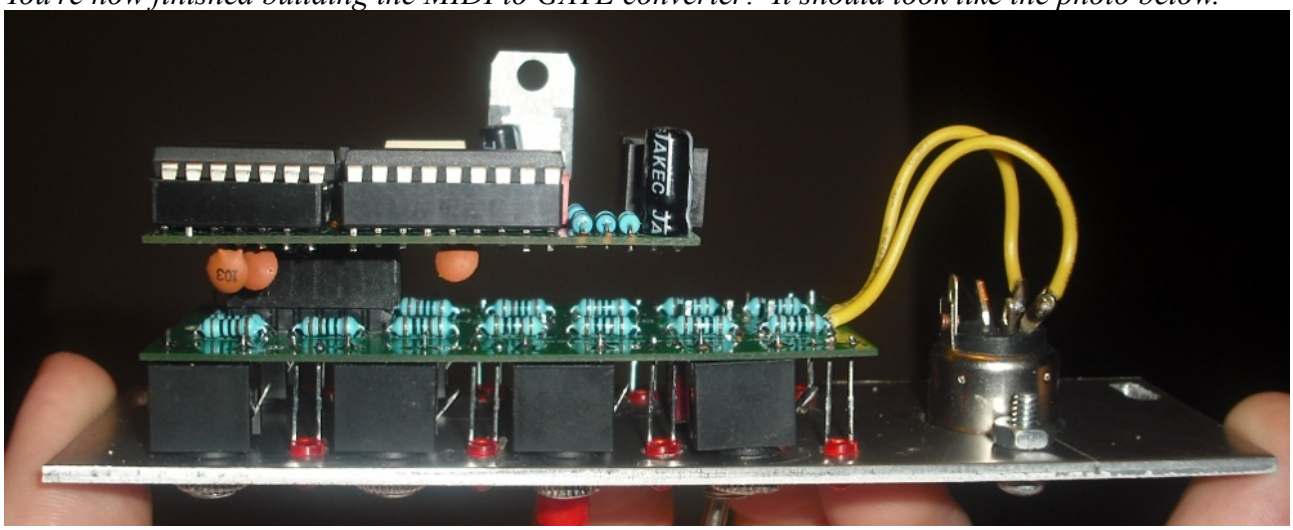
**STEP 21 -** *Gently place the PCB onto the panel. If a certain part is too far up, you may need to reheat it's solder joint while pushing the PCB towards the panel, this is most likely to happen with LEDs.*

**STEP 22 -** *Once everything is fitted into place, reattach all the nuts to the components.*
**STEP 23 -** *When the PCB is solidly attached to the panel, solder all the empty solder joints on the jacks, switches and LEDs.*
**STEP 24 -** *Using 2 screws and 2 nuts, attach the MIDI connecter to the panel, then solder the wires coming from the PCB to the MIDI connecter like in the picture below.*




**STEP 25 -** *Connect the top PCB to the bottom PCB. If using standoffs, screw the top PCB to the standoffs.*
**STEP 26 -** *Gently bend all the ICs legs inwards slightly using needle nose pliers, then insert them into the DIP sockets.*

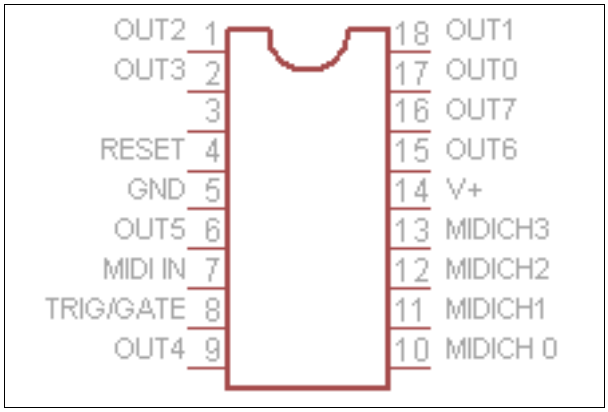
*You're now finished building the MIDI to GATE converter! It should look like the photo below.*

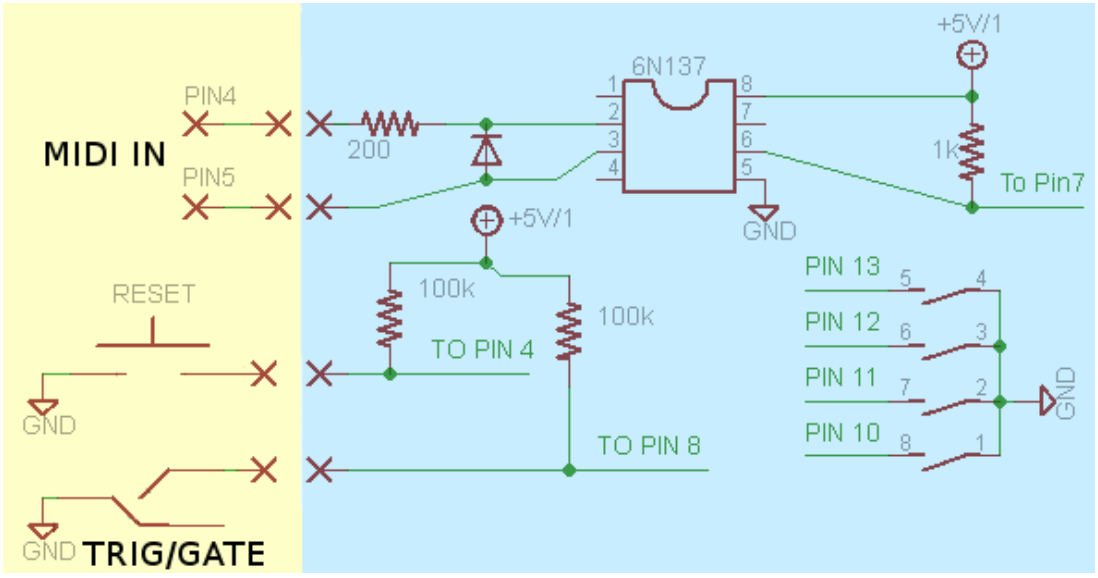


## D. Schematics

### 1. Chip Pinout

*On the right you can see the pinout for the 16F628A micro-controller at the heart of the design. V+ should be tied to +5V.*




### 2. Inputs

*In the above diagram we see the inputs for the Midi to Gate Converter. The Yellow section represents the bottom board, and the blue section represents the top board. The Xs on the boarder between the two represent the headers connecting the board.*

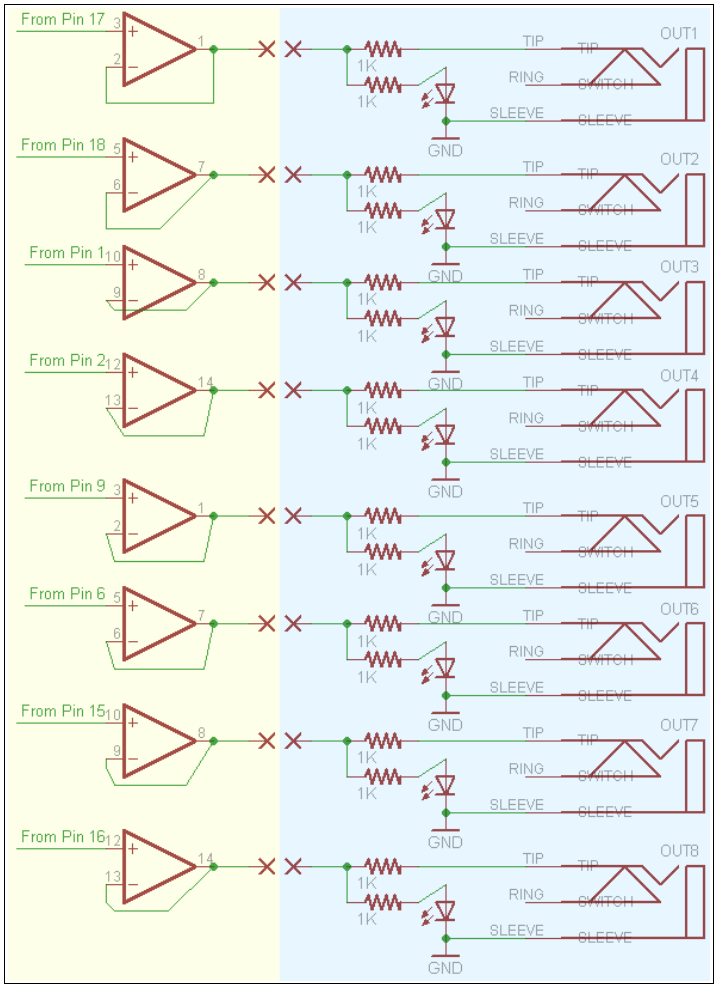
*The Midi jack is wired onto the bottom board on the two wiring pads marked "Pin 5" and "Pin 4." The signal is then sent to the 6N137 optocoupler. The 200 ohm resistor in series limits the current that can be drawn and the diode provides polarity protection. The output of the optocoupler goes through a pull-up resistor and then onto Pin 7 of the micro-controller.*

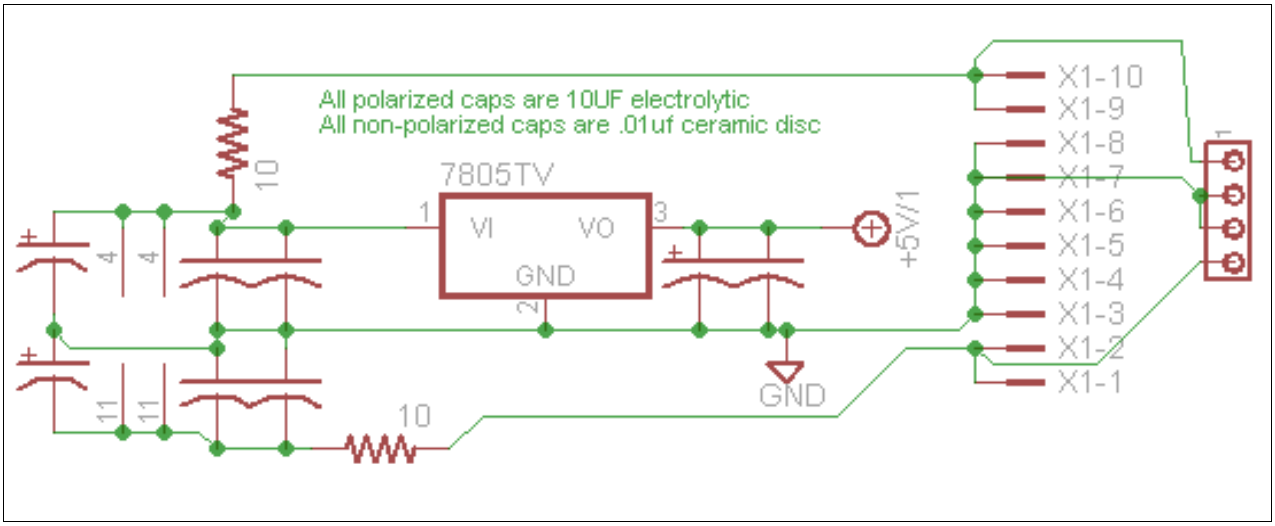
*The Reset and Trig/Gate switches are both connected to pins on the micro-controller which are normally held at +5V. Activating the switches grounds these pins out.*

*On the lower right portion of the diagram is the DIP switch used for selecting MIDI channel. All the pins are held at +5V using the micro-controller's internal weak-pull ups and the switch is grounding these pins out*

## 3. OUTPUTS

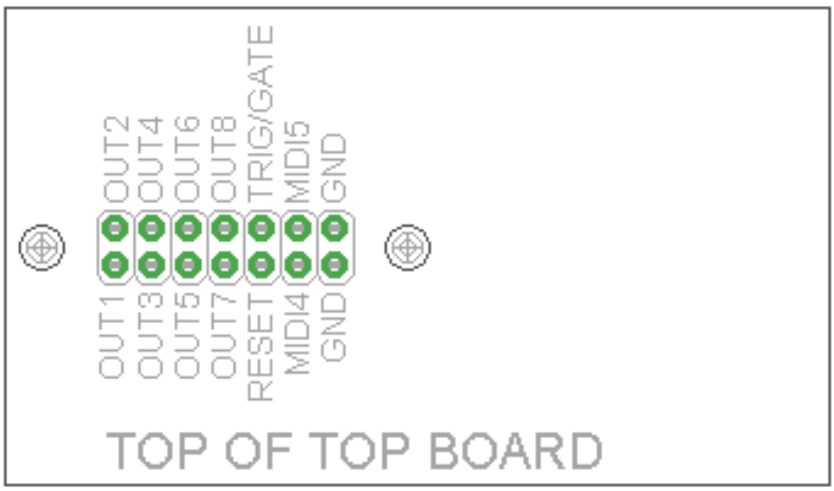
*Each of the eight outputs is identical to each other. On the top board, an op-amp acts as a buffer, the output is then sent through the board connecting headers. On the bottom board, the output goes to an LED through a 1K current limiting resistor and to the output jack through a 1K impedance setting resistor.*




## 4.Power

*Above is the schematic for the power section. The power supply connecter goes through a 10 ohm resistor and then onto the power rails. Each rail is filtered by a 10 uf capacitor and then a .01uf capacitor at each ICs power supply. The positive rail then goes to the 7805 voltage regulator making the +5V supply.*




## 5.Board Headers Pin out

*Above is the pin out for the inter board connecters. This diagram is the view from looking at the top of the top board. This pin out can be useful for troubleshooting or for creating your own bottom board.*