IN PROGRESS

MIDIbox BLM 16x16+X PCB and case build guide

Introduction

In 2010, MIDIboxer TK. made huge waves with his BLM (button LED matrix) 16×16+X. Never before had we seen such a mangle of duo LEDs, verowire and shift registers. The monome had been known for several years prior, as had smaller matrices for the SEQ, but the 16×16+X was a step further with its colourful LEDs and dedicated column, row and shift button. Arguably these pioneering devices inspired widespread development in commercial products such as the Novation Launchpad and Ableton Push.

TK.'s prototype was only ever meant to be that. In his words it took 40 hours and he was never prepared to do the same work again. Although the unit still works some five years later, the point-to-point wiring is somewhat fragile and the cheap tactile buttons are unergonomic. There were a few attempts at full PCB solutions, but the scale of the project is fairly daunting and sadly none eventuated.

My interest in a fresh start was piqued upon hearing of adafruit's UNTZtrument, which uses their own 4*4 silicone button pads. Years back we had considered pads from Sparkfun and Livid, but adafruit's are significantly smaller and cheaper. Rubberised buttons also have the best tactile feeling and don't click when they're pressed. One issue was the small internal space inside the button (only accepting a 3mm LED) so it was determined from the start to go for an SMT solution.

Concept of the project

Initial development of the BLM was done by TK., as noted elsewhere. A 17*17 button array is scanned using SCALAR modules consisting of 74HC165 (DIN) and 74HC595 (DOUT) shift register chips, the latter as current sinks via BJT transistors. Additional 595 chips drive LED anodes, while the cathodes share the same sinks as before. In total five SCALAR modules are present, one for the left-most and bottom “extra” column and row, and four for the remaining blocks of 16×4.

In the BLM 16×16+X PCB, the matrix is fully connected and all SCALAR modules are on-board, allowing for a cleaner build with less wiring, less physical space required and better signal integrity. A 4-layer PCB was used with one layer dedicated to ground/0V. There was also space to include four 75mm sliders which allow for additional control. The serial chains are brought out by a MIDIbox-standard J8/9 2*5 DIL header with the SC (clock), RC (latch), DIN and DOUT signals, along with +5V and 0v/ground. These may be interfaced with a MBHP Core8 or Core32 module, but most users will stick to the tailor-made miniCore (see later).

The following circuit changes were made:

- RC termination possible for SC and RC lines, although it seems unnecessary to do so.
- Pull-down resistors on current sink transistor bases (important when bases are floating and

should decrease power consumption).
- Schottky diodes from base to collector of current sink transistors. This helps to turn transistors “off” quicker and reduces “ghosting” in the matrix.
- Output enable (/OE) delay implemented on pin 6 of 74HC595 chips. At power up the capacitor will charge, holding all outputs in a high impedance state until the +5V rail is stable. At power down this capacitor discharges through a pull down resistor, aided by a diode. One delay circuit is provided for each SCALAR section.

For minimal external cabling, the BLM connects to a SEQ v4 via an 8-pin DIN connector. This combines MIDI in, MIDI out and power; the communications protocol sent and received by the BLM is standard MIDI. Most users will incorporate a Quad IIC board into their SEQ case, which is the simplest way to arrange the DIN8 connector, optocoupler and header pins.

Build guide

General considerations: read before building

- The 289 through hole diodes are the most tedious part of the build. They should be trimmed as short as possible before soldering from the rear (i.e. the same side as the component). If the legs protrude though too much then you'll have trouble getting the silicone button pads to sit flat. Note that it's not necessary to have the diode sitting flush to the PCB – in fact the diodes must be raised over most of the surface mount transistors.
- I suggest that the PCB is initially clamped (using a scrap piece of material to protect it from the clamp) to the work space, otherwise your parts may jump around if the board gets bumped. If it is truly flush with your bench it will also help to keep the diodes/solder from going through the plated holes. Try to protect the exposed button pads on the PCB top by using a piece of clean paper or other material; don’t damage the surface by scratching it.
- LEDs are a little bit sensitive; don’t heat them for more than about 2 seconds at a time and apparently they don’t like static discharges… As far as I know, the humidity warnings you get with them are only really applicable to oven reflow soldering.
- Rev 1.1 boards have quite small ground plane isolation, and the soldermask is somewhat prone to scratching. Take extra care when soldering capacitors and ICs that you don't accidentally short the +5V rail, as this will make troubleshooting difficult.