Bliptronome!

Wil Lindsay
www.straytechnologies.com
wil@straytechnologies.com

Bliptronic 5000 to Monome Clone Conversion
Build Instructions (version 3)

Building and Modding:

This kit was created as a synthesis of several ideas. The basic design uses the open-source "Arduino" platform to facilitate control of the Bliptronic 5000's existing hardware system, and translate it to a communication protocol understood by Monome compatible software. The circuit has been designed with quick and easy construction in mind, over flexible modification. Changes could be made to the off-board portions of the circuit, or facilitating the one unused sensor pin on the microcontroller. Additions to the design would make changes necessary in the source-code and to the PCB. The project is open-source and the firmware source-code package will be your best current resource for massive hacking and changes.

In this kit, your FTDI cable and Atmel chip have been pre-programmed and tested. Once the kit is built, either element can be changed or updated with no need to open the case or change hardware.

I've separated the process of building into several sections, to allow for breaks. A clear mind always makes for an easier build in a project like this. Over-all this project could be completed by a moderately skilled builder in a couple hours. That said, making an afternoon or a weekend of it may prevent tired mistakes.

Locate/recognize and verify all needed parts

Included Kit Hardware:

C1 .1uf capacitor - 2 legs sort of a gumdrop shape
J1 6 pin FTDI Header - 6 pins, with plastic base and a 90degree bend
J2 18 Pin Bliptronic ButtonPCB Header - rectangular female socket
J3 Jumper wire
R1-R9 10k ohm resistors (x9) brown-black-orange-gold stripes
U1 28 pin IC socket - holds the Atmel chip!
(U1) Atmel Atmega 328 microcontroller Integrated Circuit (IC) - the brain!
X1 16Mhz xtal - 2 legs, small metal canister
circuit board – note silkscreen markings on the "top" surface
ribbon cable – wire for the potentiometers
panel mount potentiometers w/ nuts (x4) – save your original knobs!
FTDI usb-to-serial cable

You provide:

Bliptronic 5000
Tools
solder, soldering iron, wire strippers, pliers, hot glue gun, phillips-head screwdriver

If you're missing something, drop me an email. I've got enough O.C.D. to count everything 3 times, but mistakes happen.

Remove old components from Bliptronic

I work on a folded towel to prevent scratching the case while working.

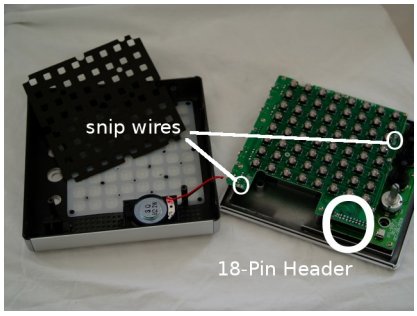
Step 1: Remove knobs. The knobs are simply “pressed” onto knurled stems. There are no pins or screws to worry about. I've found that sliding a knife, thin ruler or blade and gently prying them up removes them safely. Give some upward pressure. Spin the knob. Give some upward pressure... so forth. There is no particular need to keep the clear plastic washers found on some units.

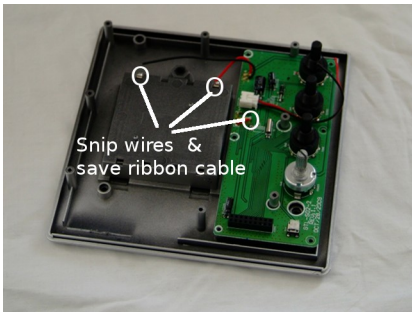
Step 2: Unscrew case. The case is held by 4 screws; one under each of the 4 rubber feet. I gently peel the feet and stick them in the battery compartment until done. Keep the screws in a safe place.

Step 3: Open Case. The easiest way seems to be to separate the top from bottom on the knob side of the case first. The only resistance will be near the audio jacks which are “pinched” into the case. Take note of your method, as the case will need to be re-assembled in a reverse fashion.

Step 4: Snip and remove the speaker wires. If you choose, this is an opportune time to remove the speaker from the Bliptronic case, as it is not used in the new configuration. The wires marked “OUT IN” can also be snipped close to the board, as they will not be used.

Step 5: Remove the Button-matrix PCB. This board is held down with 8 screws that will be re-used on reassembly. After removing the screws, lift the board straight up, away from the case, to disconnect the 18-pin button PCB header. Disconnect the 4 wire ribbon cable on the board's bottom. And set the board aside with the top of the case.





Step 6: Remove the Bliptronic “synth and pot” PCB. There are 6 screws holding this board down, of which we will only need 3 (so you have some extras to lose!). Note the location of the 18-pin female header, as the new board will need this located in the same spot. You will need to cut the 2 battery terminal lead wires to complete removal. This board is trash. If you find a use for it, drop me an email. For now, remove the 4-wire ribbon cable and snip it's plastic connector. This ribbon cable will be used for our potentiometers in a later step.

Populate the circuit board

The silkscreen markings on the board top relate to the components listed above. The order of events listed below will help keep the build process simple. Specific caveats and the recommended order are listed below. If you are a beginner solderer, there is an excellent tutorial at
http://www.curiousinventor.com/guides/How_To_Solder
and this is one of the better documents I've seen:
http://www.dxzone.com/cgi-bin/dir/jump2.cgi?ID=17512

In general:

Fit the component and verify its position.
– a note will be below if part orientation is important (ex: which end is which)
Flip the board and solder it in place
Trim the leads and move to the next component as you go
Pin spacing is tight on this small board. Verify that trimmed leads do not contact each other, creating an electrical short.
Verify that solder pads are not "bridged" by solder blobs.

NEVER solder anything on the board with the Atmel controller IC installed in the socket! It is susceptible to heat damage unlike the other components.

Build order:

(J3) Jumper Wire (white)

Hole-to-hole where the white line is silk-screened. Flip and solder the 2 ends. Trim ends.

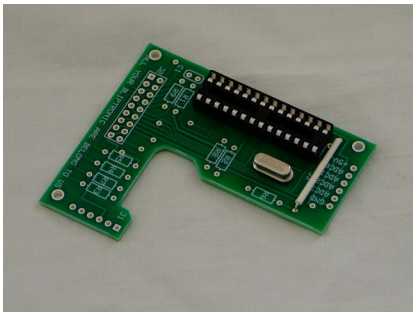
(U1) 28 Pin IC Socket

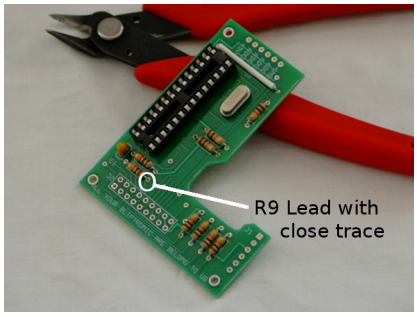
There is a semicircle cut in one end. Verify that this lines up with the notch on the silkscreen drawing for U1. This will help us verify correct orientation of the Atmel chip later. It's very important that this part be as flush to the board as possible. A large air gap between part and board will make for a difficult fit in the Bliptronic case later.

Fit the component. Flip and solder 2 corner pins. Verify flush seating (heat pins and re-seat if needed). Solder remaining pins. A missed pin may mean missed electrical flow later.

(X1) 16Mhz Crystal

Pin orientation is not important. Keep part nearly flush with board when mounting.





(R1-R9) Resistors

Pin orientation is not important. Trim the wires short to prevent them from touching any other components. *** Please note that a top-side trace comes very close to one of R9's solder pads. Be sure to not "over-glob" solder on this resistor, as it could potentially short the trace. This rare accident would result in a row of buttons not being read by software, and is repairable if you have that symptom.

(C1) capacitor

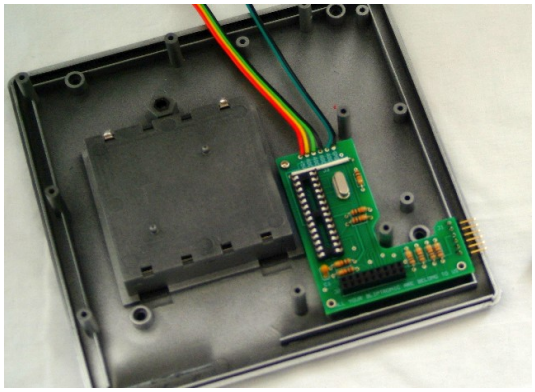
Pin orientation is not important. Mount as pictured.

(J2) 18 Pin Bliptronic ButtonPCB Header

This component fits tightly to the Bliptronics connecting pins. Verify that the component is flush against the PCB before soldering, and verify that it is not crooked in the holes.
Flip and solder 2 corner pins. Verify flush seating (heat pins and re-seat if needed). Solder remaining pins.

(J1) 6 pin FTDI Header

This part will fit in the case better if it is mounted "upside-down." The 6 pins should bend at a 90 degree angle from the bottom of the board, and you will solder the pins from the top of the board. The black plastic joints should be flush with the bottom of the PCB. Solder one end pin . Verify the seating is flush with the board. Solder other end pin. Re-verify. Solder remaining pins.



Ribbon Cable

Strip both ends and Solder 4 ribbon cable wires to the PCB terminals marked ADC0, ADC1, ADC2, ADC3 I've provided a small amount of ribbon cable. Use 2 wires of the multi-color ribbon cable (snipped from the old synth board,) to connect +5V and GND.
there should be plenty to do the job. The remaining 2 wires will be used later.

Test and fit the newly built PCB

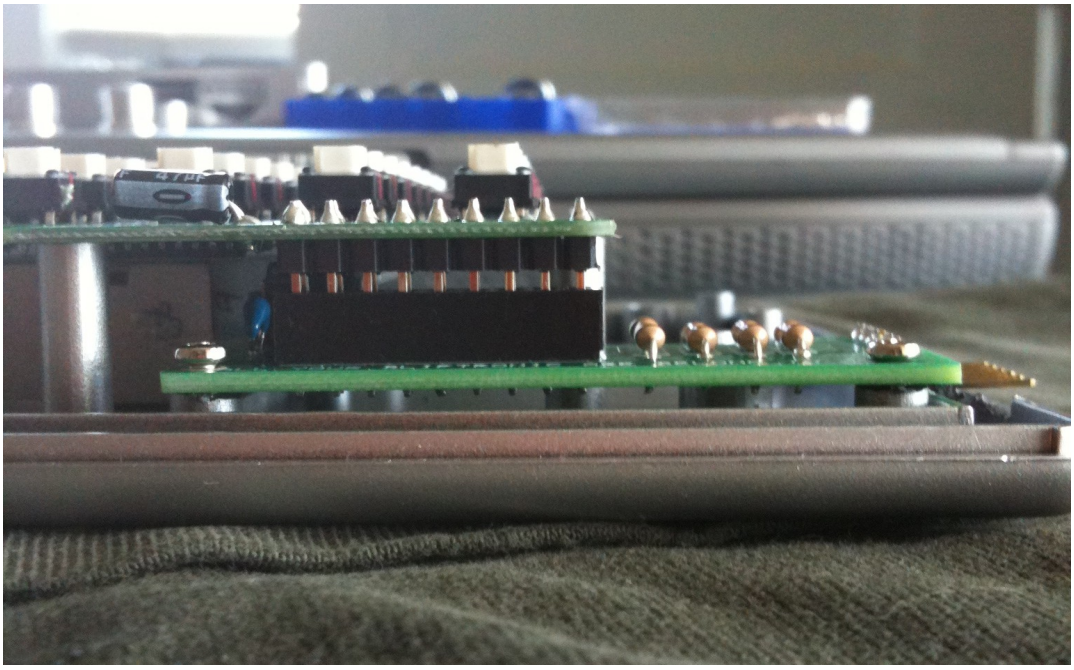
At this point the PCB can be mounted, and tested. First seat the Atmal IC in the 28-pin socket. It is very important that the semi-circle "notch" at one end of the IC lines up with the notch on the socket and on the silkscreen printing on the PCB. Once the legs are correctly aligned, give a firm downward press until both ends of the IC are against the socket. Firm pressure and non-twisting are the key. It is important that the IC is seated as far into the socket as possible.

The PCB will eventually fit on three of the screw posts where the original synthesizer board lived in the Bliptronic's case. The board's hook shape allows for the FTDI pins to reach the edge of the case, while passing around various mounting posts. For testing fasten only one screw nearest the ADC wire holes to hold the board down temporarily.

Once attached, carefully insert the 18 pins on the button-matrix PCB into the 18-pin female header of the new board. Another screw can be used to temporarily stabilize everything if needed.

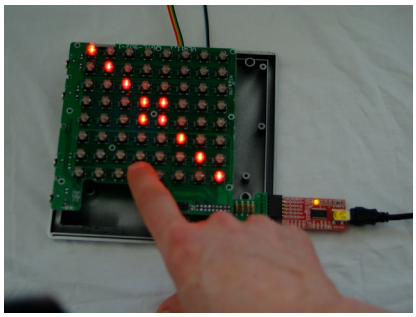
I've found the factory leads on the button board to be a bit long on the 2012 version of the Bliptronic, causing a "bulge" directly over the Atmel IC. In general, the build quality has changed a bit

over the years. If things are still a bit tight (sometimes making some buttons somewhat hard and “unclicky”) the 3 mounting posts for the Bliptronome PCB can be shortened about 1 or 2mm. I've done this successfully with diagonal cutters, letting the screws re-thread themselves through the soft plastic posts. This may cause the 18 pin socket to be slightly unseated which is fine. As long as the pins make contact (no more than a 2mm gap) everything will work well and stay connected. This image is of a March 2012 build with a very new Bliptronic 5000. Note the 2mm gap where the pins are visible.



Cutting these 3 posts may also require nipping a bit of the lower case for the 6-pin FTDI header, (as pictured). More info on that is found in the “case modification” section below.

To test, plug the FTDI cable into the 6 pin ftdi header (smooth side of the female header up). Next plug the USB cable into the board and your computer. The goal here is only to apply power, so driver installation can wait until later. A properly built kit should exhibit a quickly scanning light stripe across the 64 LEDs. (*The small red board pictured below is built into your FTDI cable now, and no longer needed. Folks from the first release of the kit used this board instead of the cable you have).



If you are super anal-retentive (like myself) or have the software already installed, you could proceed to test all of the buttons and LEDs using Max/MSP and the monome_test.mxb patch as outlined in the “Getting Started with the Software” document. My general feeling is if you are on a roll: you built it; You can always take it apart and fix it later. :)

VERY IMPORTANT: Remove the Atmel chip be for continuing. Heat from further soldering can cause damage to the internals of the IC. A tiny screwdriver or blade inserted at either end between the socket and IC will allow gentle prying. Alternate ends, and verify that the pins are straight and aligned once the IC is removed. They can be gently bent to a straight position with little harm.

Case Modification

There are three small areas of case modification needed if adhering to my design. They are outlined below. I've had very good success with a minimal amount of tools and a fair amount of patience. I have no doubt that there are more professional looking methods, but my goals were robustness and ease of installation.

FTDI cable slot : The new circuit board was designed for a recessed access to the FTDI header pins from outside the case. To allow the female header on the FTDI cable to attach correctly, a small rectangle will need to be removed from the top portion of the case.

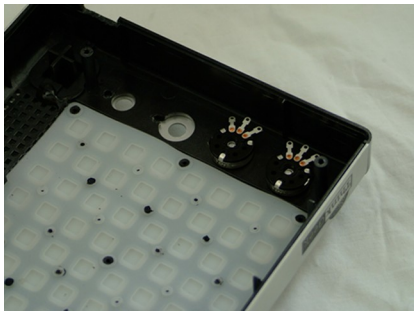
My method of determining this clearance is to mount the PCB loosely as outlined above, and gently lay the top half of the case where it would be attached. I then mark a (roughly) 5mm line on the case with a fine point sharpie/marker or X-acto blade. Once removed I finish drawing the rectangle to be removed using the FTDI cable's female header as a guide for height.

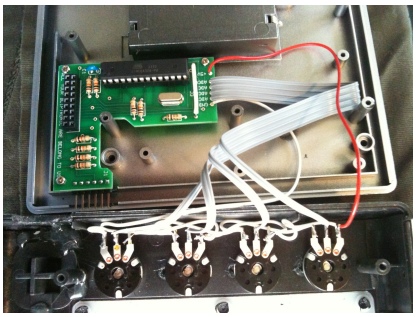
This marked rectangle can now be cut or shaved away using a knife, utility blade, heated blade, or even very small wire nips. Finishing the edge with a small piece of sandpaper and a flat tool such as a file, knife or screwdriver can give a very clean straight edge. On newer Bliptronics, you may need to cut a small 2mm slice from the interior ledge on the bottom case shell as well.

Potentiometer mounting flange holes : Each potentiometer has a small flange of metal on top designed to pin into the case and prevent the entire body from spinning while the knob is turned. For these to work correctly, 4 small holes of the flange's diameter will need to be cut into the case top plastic from the inside. The case's aluminum top will prevent crude drilling or cutting from being visible on the completed unit.

To locate the position of the four holes, remove the nut and washer from a potentiometer. One at a time Place the shaft through the case hole with the 3 solder points facing the closest edge of the case. (see picture) From here you can see how the flange prevents the potentiometer from sitting straight. Mark it's location with a sharpie/marker or blade. On the wider case holes, a completed flange slot will break into the hole. This will not adversely effect it's function.

Despite the brain-frying oder, I've found that a quick stab with a solder iron creates a usable flange hole. You may have better luck and a longer life using wire nips, a hobby knife, a dremel tool, or slowly twisting a drill bit by hand. I've had success with a 3mm or 1/8” diameter bit. The hole needs to go through the plastic so that the metal is visible, to allow proper potentiometer seating. In any method, verify there are no plastic pieces or dust floating around before mounting the potentiometers.





Play button : In my design the rectangular play button is not used. If following this design, I recommend a dab of hot glue or hobby model glue to keep the button from floating around in the case.

After resolving these 3 modifications, remove any dust with a damp cloth, allow to dry, and verify the fit with the installed PCB one last time.

Mount and wire potentiometers

We can now mount the 4 potentiometers, aligning the flange-holes and using the 4 nuts to complete the mounting. Again, verify the 3 solder tabs on each potentiometer are facing the closest case side. DO NOT OVER TIGHTEN, over tightening, or using heavy weight on a wrench will cause the pots to flatten against the plastic case, and squash their internals, making them impossible to turn. The flange pins should prevent random loosening during use, but a tiny amount of hot glue will verify their longevity and be neatly hidden by the knobs. I've found that hand tightening with a 10mm socket (not the whole socket wrench) is more than enough.

Once mounted, you can begin soldering the connecting wires to the ADC points on the board. The photo above will make more sense than my steps... but here they are! I've used the remaining 2 ribbon-cable wires (each cut into 3) for the daisy-chains below.

Step 2: Strip and trim small wires to daisy chain from the 5V solder pin on each pot. Solder. (red wires pictured above)

Step 3: Strip and trim small wires to daisy chain from the GND solder pin on each pot. Solder. (white wires pictured above)

Step 4: Solder the ribbon cable from wire points ADC0, ADC1, ADC2 and ADC3 PCB terminals to the center pin of each potentiometer. It's not necessary to keep the ribbon cable wires attached to each other for function. ADC3 is closest to the “play” button.

Step 5: Solder ribbon cable wires from 5V PCB terminal to the ADC0 pot's 5V solder lug (closest solder lug to the corner of the case) and another from the GND PCB terminal to the ADC3 GND solder lug (closest solder lug to the “play” button).

Test again!

At this point, I'd like to believe the soldering iron can be turned off. The best solution would be to take a break and then turn to the “Getting Started with the Software” document. Everything can be tested “loosely mounted” and with the case split. Just be careful not to yank any wires when flipping the case top back and forth.

Seal the case

Once testing is complete and you are satisfied, here is my installation order. You may find a better technique, but the key here is alignment. Things are precisely measured.

Verify the Atmel IC is firmly seated in the IC Socket.

Mount the Bliptronome PCB using only the top screw (closest to the battery compartment).

Connect/verify the Button Matrix PCB as in testing. Screw in the 2 center screws to verify alignment of screw-holes. Then the screw closest to the 18-pin connector.

Screw in the last 2 screws on the Bliptronic PCB, now that it is properly aligned.

Screw in/ tighten any remaining screws on the button board.

Place the black button mask on the buttons being aware of the two extra edge-cuts that belongs on the bottom and top left.

Verify the white button rubber is smoothly seated in the case top.

Close the case, snapping in the audio jack area first while evenly lowering the knob side.

Snap tightly, and steer the audio jacks into there holes with a small tool, if needed.

Flip it over, and re-thread the 4 screws, then replace the rubber feet.

Flip again, and turn the 4 ADC shafts all the way counter-clockwise. Press on the knobs so they aesthetically point in the same direction.

In an idea world we're done. I've found that the cheaply cast knobs may need to be lightly lifted with a blade, or even tightened with a dab of hot glue or by a small piece of tape fitted to the shaft before pressing. Straight, well mounted knobs should turn smoothly. A “gritty” feel is the result of the knob scraping against the nut, or Bliptronic case. Re-seating the potentiometer or the knob may be necessary.

Final Test

That's it! From here, you may already have enough to move on to the fun part. At very least give the monome_test.mxb one last spin, and the “Getting Started with the Software” document one last browse. --Enjoy!