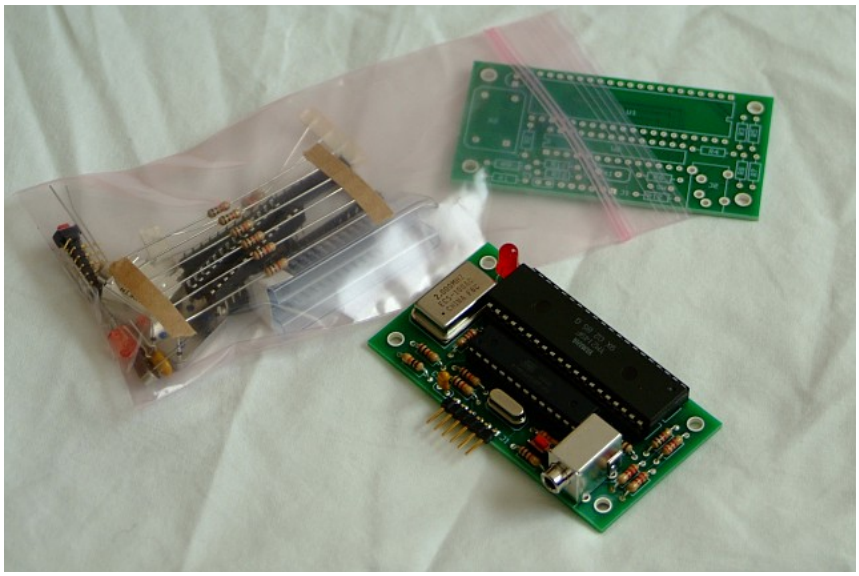
YM_Mini Synth chip synthesizer

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

YM2149 usb/midi synthesizer
Build Instructions Version 1

Building the Kit:
The YM_Mini was designed with ease of construction in mind. Overall, the design takes advantage of software based control, allowing for a very small electronic parts count. This means less cost, and less complexity as a soldered electronics kit.

The small size allows for different possible cases to be modified for fit if desired. Any case needs only access ports for the FTDI cable (providing both power and midi in one cable), and a single button which allows a "panic" stop for all sounds. (This is handy if software is terminated or crashes while a note has been sent to the synth with no related note-off command.) Possible cases include old game cartridges, or small boxes or containers.



Though some additional expense was incurred through use of the FTDI-USB cable, this choice was made to allow future extension to the synth. The 6-pin FTDI port can be used for control by other microcontrollers such as the Arduino platform, or eventually controlled via a planned midi 5-pin jack and power board, that would plug into the same spot allowing untethered use of the synth with hardware-based midi controllers and sequencers.

The kit building process is outlined below in sections, allowing for organization and breaks as needed. The instructions are designed with a beginner in mind, and can likely be quickly skimmed by an expert builder.

Locate/recognize and verify all needed parts

Included Kit Hardware:

C1	.1uf capacitor - 2 legs sort of a gumdrop shape
D1	LED
J1	6 Pin FTDI Header – 6 pins, with plastic base and a 90degree bend
J2	1/8" Stereo Headphone Jack – metal box w/ 6 legs
R1	10k ohm resistor – brown-black-orange-gold stripes
R2	220 Ohm resistor – red-red-brown-gold stripes
R3-R4, R9-12	1K Ohm resistors (x6) – brown-black-red-gold stripes
R5-R8	2.2K Ohm resistors (x4) – red-red-red-gold stripes
SW	button – 2 legs
X1	16Mhz Crystal – 2 legs, small metal canister
X2	2Mhz Oscillator – 4 legs, metal rectangle
U1	40 Pin IC Socket – plastic holder for the YM chip
U2	28 Pin IC Socket – plastic holder for the controller
ATMEL ATMEGA 328 28 Pin IC – the midi brain	
PCB	the circuit board!

You Supply:

FTDI usb-to-serial cable

YM2149F 40 Pin IC – the synth! (provided as an extra kit option if you do not have one pulled from an AtariST, or legacy computer, game, etc...)

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.

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 or YM chip installed in the socket! They are susceptible to heat damage unlike the other components. Place these components only as the final step after soldering.

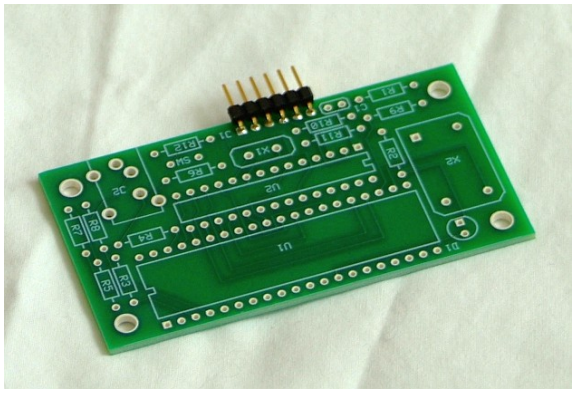
Build order:

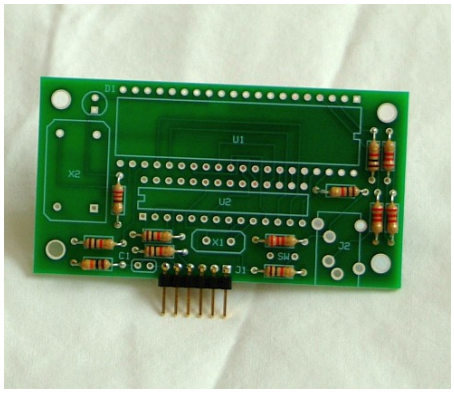
J1 6 Pin FTDI Header

The 6 pins should bend at a 90 degree angle from the top of the board, and you will solder the pins from the bottom of the board. The black plastic joints should be flush with the top of the PCB. Solder one end pin . Verify the seating is flush with the board. Solder opposite end pin. Re-verify. Solder remaining pins.

R1- R12 Resistors

Pin orientation is not important. (It doesn't matter which end is which on a resistor) After soldering, trim the wires short to prevent them from touching any other components. The various resistor values can be determined from the color code stripes as listed above. It is essential that the correct value resistors are in the correct places as numbered on the PCB.





U1 & U2 28 Pin and 40 Pin IC Sockets

There is a semicircle cut in one end of each. Verify that this lines up with the notch on the silkscreen drawing for U1 and U2. This will help us verify correct orientation of the Atmel chip and Synth chip later. Fit the socket to the PCB. 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. Verify all pins are fully soldered into their corresponding holes.

X1 16Mhz Crystal

Pin orientation is not important. Fit to PCB, solder and trim.

X2 2Mhz Oscillator

Note that one of the 4 corners of this component has a point while the other 3 are rounded. This will help determine the orientation of the component and must match the graphic silk-screened on the PCB. The oscillator has small feet underneath and will sit slightly away from the board when mounted correctly.

SW 2-pin button

** Note: this PCB button may be used if no case is desired for the synth. The button orientation is not important and can be mounted directly in the 2 holes marked SW. If a case will be added: any 2 pin button can be used and mounted elsewhere by soldering extension-wires into the 2 holes at SW and soldering the other ends to a button you provide. This will allow any type of panel mount button to be used and mounted anywhere on your chosen case. Any momentary button will work, provided the button is “normally off” when not being pressed. Wire length will not effect the function.

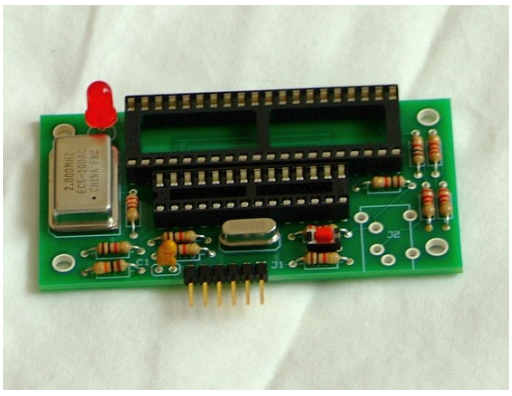
C1 .1uf capacitor

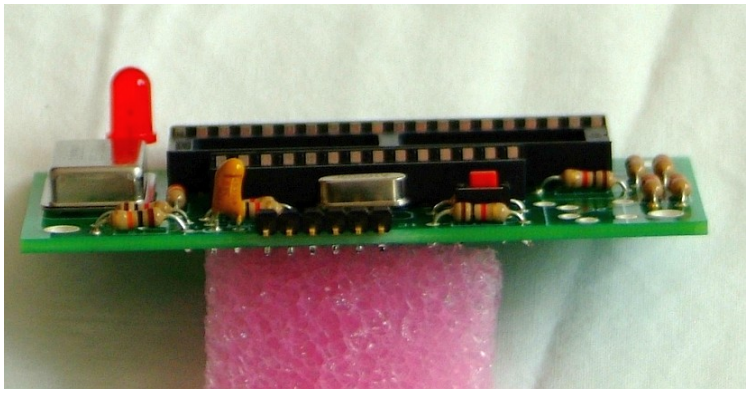
The provided capacitor may be marked with a tiny “+” symbol directly over one leg. If this symbol is present, the capacitor must be orientated such that the “+” leg is in the hole closest to the 6-pin FTDI header.

D1 LED

The specific Led color is not important and may be swapped for an LED of any color if you choose. The leg orientation of the LED is important. Note that the circle drawing on the PCB for D1 is a circle with one flat edge. This flat spot corresponds to a flat spot also on one part of the LED's plastic collar. The flat spot designates the closest leg as negative, and that leg must go into the square hole in the PCB. Though LEDs vary from manufacturer to manufacturer, the shorter of the 2 legs is also a common marker of the negative leg.

The Led could also be replaced with 2 wires if a case mounted LED is desired. Verify orientation when doing this, as a backwards LED will not correctly light. (See SW for the same concept)





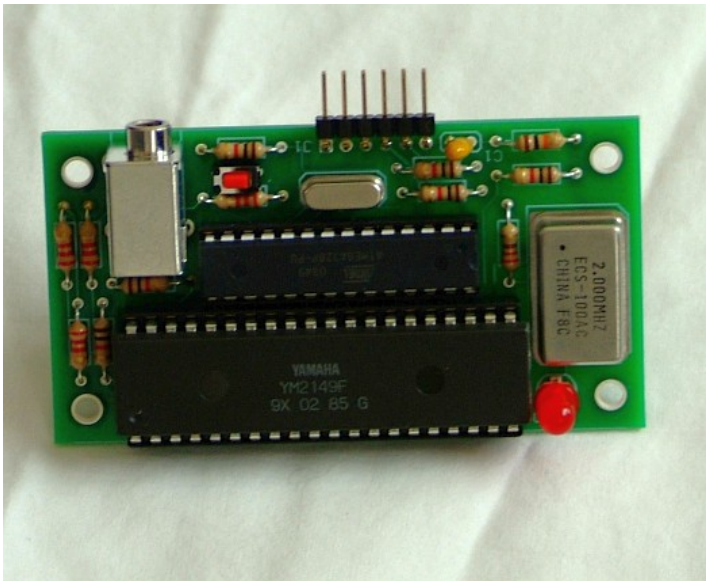
J2 1/8" Stereo Headphone Jack

The stereo jack can be a difficult fit into the 5 holes. The legs may need to be slightly bent to correctly fit. The 3 straight legs are the essential pins as they carry the audio signal, and must have a good solder joint. The crimped legs are designed to only secure the case to the PCB and flooding the entire hole with solder on these legs is not essential for correct function.

Inserting the IC chips

Once soldering is complete, and all joints are visually verified, the Synth IC and Atmel chip can be inserted into their corresponding sockets. These chips each have a small dimple on the top directly over “pin 1” of the IC. In the case of the Atmel chip, this dimple is also close to a semi circle on one end. Each socket has a notch in one end. “Pin 1” of the IC must be closest to this notch when inserted, or the synth will not function correctly. Be aware that the Atmel and Synth chip face opposite directions and the PCB drawings have notches on opposite ends to mark proper orientation.

You may need to slightly bend the legs to make a proper fit in the socket. A slight rocking motion against a flat table-top can bend all pins on the IC a small amount simultaneously while keeping the pins properly aligned. A small air gap between the IC bottom and the socket is completely normal and the chip will function correctly as long as all legs connect tightly to their corresponding metal sockets.



Testing the Synth Kit

At this point the synth can be tested. Please see the second document labeled “YM_MINI_Instructions.pdf at http://www.straytechnologies.com/resources/ This document will lead you through proper software and driver setup. The synth will go through a blink sequence when power is supplied, given proper construction.

If the synth does not blink correctly or function, visual inspection of all component placement is the first step in trouble shooting. Make sure all components are properly orientated as mentioned above, and that all solder joints are shiny and do not bridge to other pins. Help and troubleshooting suggestions can be had at the website forums at:
http://www.straytechnologies.com/forum/

Feel free to email me with any questions at wil@straytechnologies.com

enjoy!