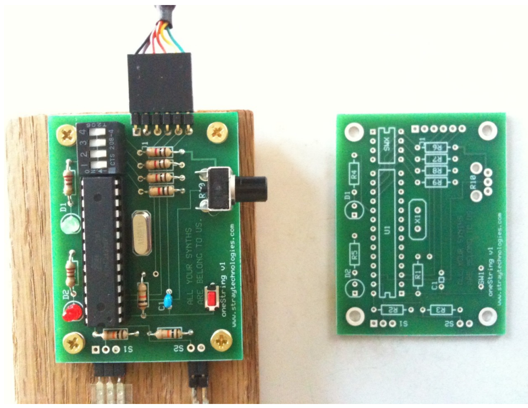
oneString Controller

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

open source USB ribbon midi controller
Build Instructions

This document is a guide to building the oneString USB controller.
See the associated "Getting Started" document for a bit of history, theory and discussion of the project beyond the hardware device.

If you have a tested working PCB and only need to mount the works to an instrument body, feel free to jump down to the section marked "STEP FOUR: Mounting the Sensors and PCB."



Building and Modding (and a bit of poorly veiled philosophy):

The oneString controller was designed as an open source / open hardware controller with attention given to low cost and ease of modification. As such, this circuit and the related firmware could have many changes made to turn it into a unique and personalized instrument. The instructions below lead to a basic and highly functional version that can serve as a finished device, or the beginnings of a more complex project.

With these ideas in mind, the circuit is dependent on USB via a specific FTDI USB-to-Serial cable, for both power and transmission of midi messages to a host. This limits the potential for traditional 5-pin din midi, as it would require more parts, a more expensive design, and a complete power circuit (likely a wall wart and power filtering circuit). Though the oneString can't plug directly into your DX-7 or other analogue synth, I feel that the cost savings, simplicity, and flexibility of computer based synthesis make this more than a fair trade, and leave this inexpensive project with unlimited potential as an expressive device.

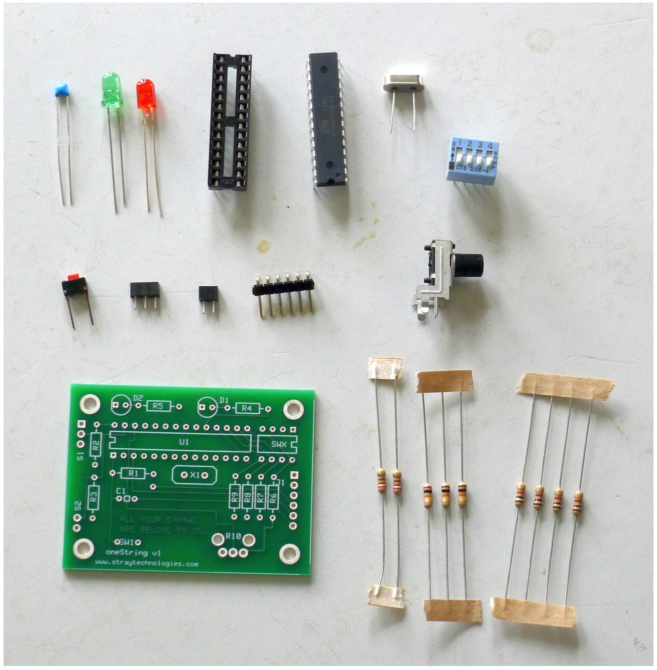
The base description below also suggests an open PCB mounted on a easily-obtainable piece of wood which can be bought as-is from your local big-box hardware store for less than $5USD. I have been performing in beer-soaked clubs, and traveled via train, plane, and subway with open circuits like this for several years now, with no real significant problem or lack of reliability. Plus, the open-hardware aesthetic generates a lot of attention.

I hope some builders take the time to create and share designs for cases, covers and different bodies for the oneString. The potential for truly unique and beautiful instruments are endless, especially considering the new generation of open prototyping platforms and companies like www.ponoko.com and www.bigbluesaw.com. If you want to share such a design, please let me know and I will be glad to publish it on my site or set up a section on the wiki for the design.

Other potential modifications are described inline in the instructions below and in the last section called "Possible Advanced Modifications." It might help to review the full process below and that section, before beginning the work.

STEP ONE : Locate/recognize and verify all needed parts

Included Kit Hardware:



C1 .1uf capacitor
D1 Green LED
D2 Red LED
J1 FTDI Header
R1,R2,R3 10k ohm resistors
(stripes = brown-black-orange-gold)
R4,R5 220 Ohm resistors
(stripes = red-red-brown-gold)
R6,R7,R8,R9 1k Ohm resistors
(stripes = brown-black-red-gold)
R10 5k Tuning Potentiometer
SW1 reset button (2 pin)
SWX 4x Dip Switch (8 pins)
U1- socket 28 Pin dip socket
U1 ATMEL ATMEGA 328 controller
(the brain!)
X1 16Mhz
S1 3 pin sensor connector
(male or female depending on your sensor)
S2 2 pin sensor connector
(male or female depending on your sensor)
Printed Circuit Board (PCB)
– note silkscreen markings on the “top” surface

Additional items provided in the “full” DIY kit

FTDI usb-to-serial cable: http://www.sparkfun.com/products/9718
SpectraSymbol SoftPot Position sensor: http://www.spectrasymbol.com/softpot
2-pin Pressure sensor: http://www.sparkfun.com/products/9375

You provide:

wood or acrylic base: 3”x .5” x 24” recommended (2” x 24” minimum)
(Lowes has a pre-cut unfinished oak board this exact size : item# 1195)

Tools:

solder, soldering iron, wire strippers, pliers, phillips-head screwdriver, small blade, rubber cement

Some modifications or layouts would be easier with a bit of flat ribbon cable, or other parts. Plan ahead, and decide on your layout and the parts needed before you start.

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

STEP TWO: 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 from the bottom
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.

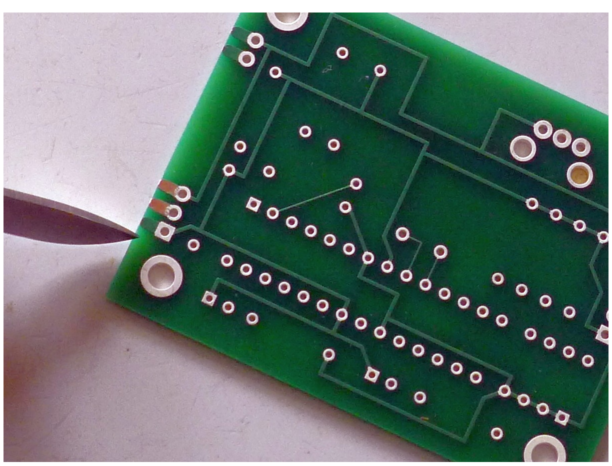
NEVER solder anything on the board with the Atmel controller IC installed in the socket! It is susceptible to heat damage unlike the other components.

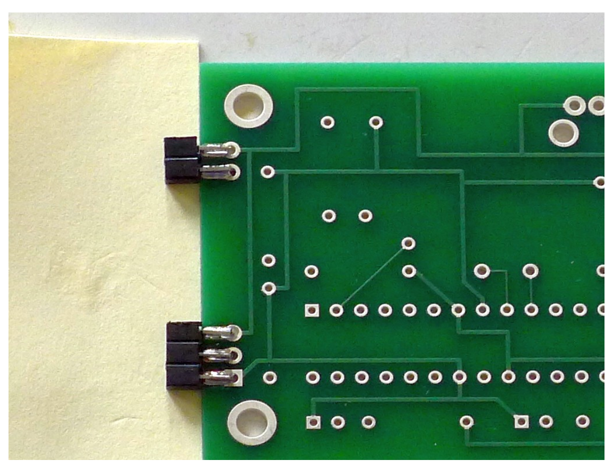
Before proceeding, verify which connector type is needed for your two control sensors. The default layout (with a supplied parts kit) assumes that female sockets will be used. Other mounting styles can be used as described below. The PCB has been designed for multiple potential layouts.

BUILD ORDER:

S1 & S2 3-pin connector & 2-pin connector

In the default layout, these small connectors will be soldered with their pins laying down sideways against the 2 & 3 grouped stripes on the PCB's bottom side edge. In this way, the male pins from the strip sensors can plug into these small black sockets and lay flat. Note that the green "resist" coating on the board covers these stripes by default. The coating will need to be gently scraped with a blade or small knife until the copper strips are visible and shiny from the solder holes to the board's edge. (See pictures for scraping and orientation.)





It may be easiest if the newly scraped copper pads are "tinned" before applying the part (coated with a little solder). This will allow the part to be heated and quickly connected to the board. An alligator clip, or "third hand" may make this part easier to solder. I've found good success in placing 6 sheets from a memo pad or other paper under the parts to support them at the correct level while soldering. Fear not! This is the most difficult part of the build... It's all easier from here on!

If your position sensor is supplied from elsewhere and has a female socket, a 3-pin male header can be installed in the 3 holes in the same way that the FTDI header is installed. Another possible layout would allow you to solder in 3-wire and 2-wire ribbon cable to the sensor connector holes. This would allow changing the layout of your overall instrument or even using different types of analog sensors. See the final section "Possible Advanced Modifications" for a list of ideas.

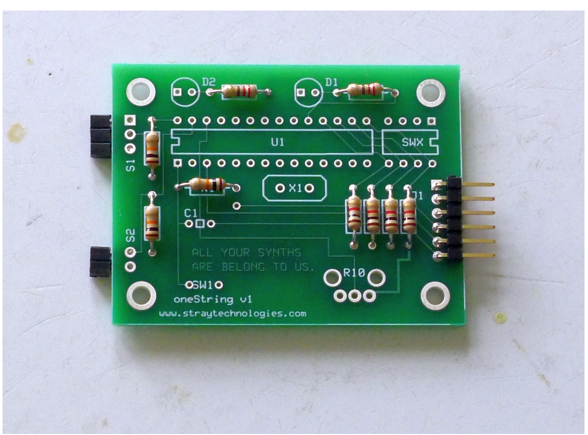
J1 6 pin FTDI Header

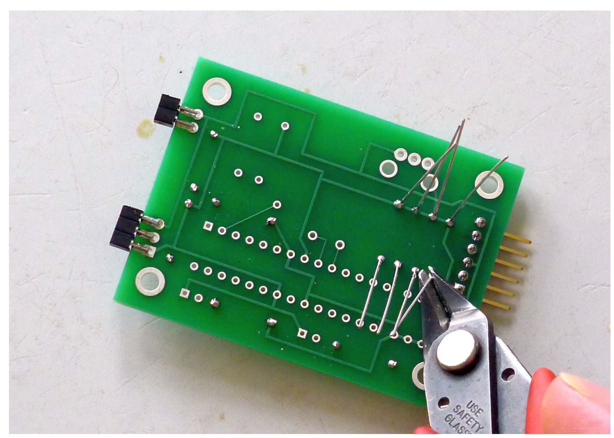
The 6 pins should bend at a 90 degree angle from the top of the board, and you will solder the pins from the bottom of the board. The black plastic joints should be flush with the top of the PCB. Solder one end pin. Verify the seating is flush with the board. Solder other end pin. Re-verify. Solder the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

R1-R9 Resistors

Verify the correct value resistors are in the correct spaces, by comparing the stripe colors to the stripes listed in the parts list. Pin orientation is not important. After soldering, trim the wires short to prevent them from touching any other components.

R1,R2,R3	10k ohm resistors (stripes = brown-black-orange-gold)
R4,R5	220 Ohm resistors (stripes = red-red-brown-gold)
R6,R7,R8,R9	1k Ohm resistors (stripes = brown-black-red-gold)



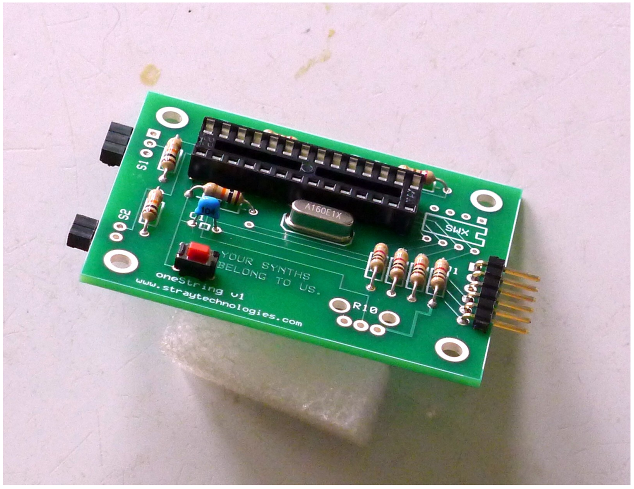


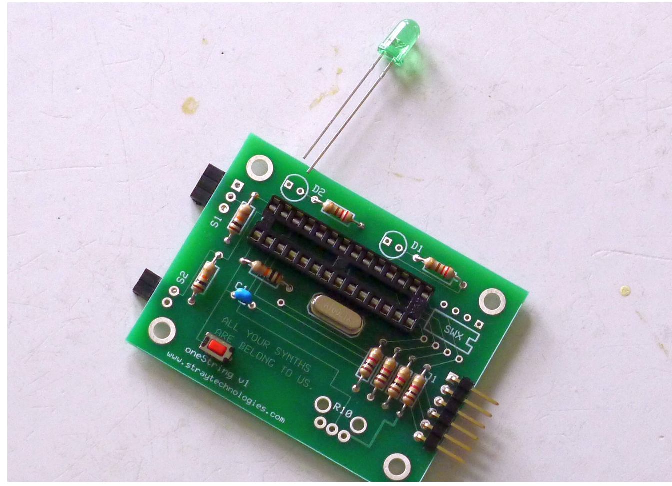
X1 16Mhz Crystal

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

U1 28 Pin IC Socket

There is a semicircle notch cut in the center of 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. Fit the component, gently straightening any pins as needed for all 28 to line-up with their respective holes. Flip and solder 2 of the corner pins. Verify flush seating (heat pins and re-seat if needed). Solder remaining pins. A missed pin may mean missed electrical flow later. Socket leads are generally not trimmed after soldering. DO NOT CONNECT THE ATMEL CHIP YET!





C1 **Capacitor**

Pin orientation is not important. You may need to slightly bend the pins to span the holes. The component does not need to be flush with the board, and can have air space below it.

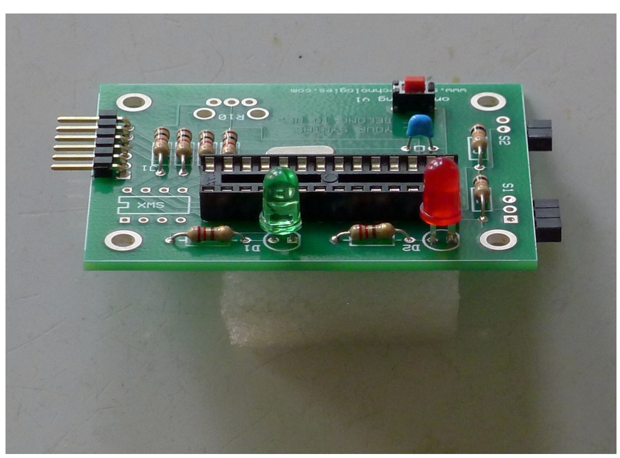
SW1 **2-pin button**

Pin orientation is not important. Keep flush with board for mechanical strength.

Under normal operation, this button will be used very little. This part can be replaced with a “panel mount momentary switch” (button) available from many electronics stores (such as Radio Shack part #275-1571). If this is done, the button leads can be soldered to SW1 solder holes, and solder those wires to the panel mount button when mounting your case. Again, orientation is not important.

D1-D2 **green and red LEDs**

LEDs are *polarized*, meaning the direction they are mounted matters for correct operation. In general LEDs have a short leg and a long leg. This short leg must go in the hole closest to the flat edge of the semicircle silk-screened in each position (square pad hole). Some LEDs may also have a flat side which will line-up with the silkscreen pattern. Like the reset button, these LEDs could be replaced with panel mount LEDs that can be attached to wire leads and connected to the same solder holes. Be VERY aware of orientation when making these extensions, as reversed LEDs will not light correctly. Allowing an air-gap under the LEDs is fine (and sometimes preferred) instead of mounting flush. If mounting flush be careful not to overheat the pins, as LEDs can melt or become damaged from the soldering process, if heated for an extended period.



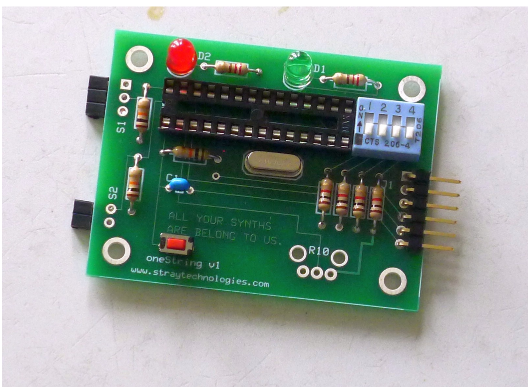
SWX 4x Dip Switch (8 pins)

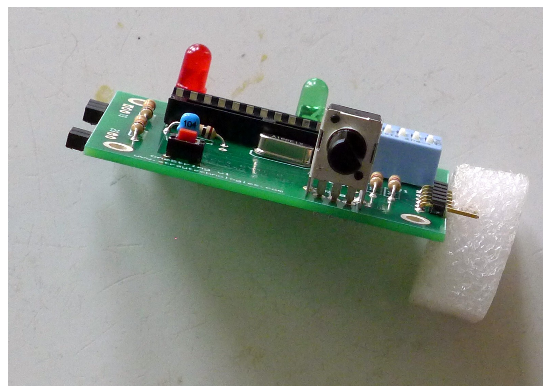
Mounting this switch is similar to mounting the 28-pin dip socket. It's easiest to solder 2 corners first, and verify a nice fit before continuing. The dip switches should be oriented with the “1” (one) switch closest to the 28-pin socket, and the “4” (four) switch closes to the PCB's edge. This is oriented so that the “on” position of the switches is closest to the LEDs. Some dip switches come with a piece of plastic protecting the sliders, that can now be removed.

R10 5K Tuning potentiometer

When pressing this part into the PCB, firm pressure will allow the part to be flush with the board. The two larger holes support the potentiometer body, and should be completely filled with solder for good mechanical strength. The three small pins are the electrical contacts.

This item could be replaced with a “5k panel mount “linear” potentiometer” (such as Radio Shack 271-1714). If this is done, attach 3 wires to the 3 small potentiometer holes, and solder them to the potentiometer when mounting the case. The 2 larger holes in the R10 area are not used in this scenario.





Adding the ATMEL IC for testing

Before seating the ATMEL chip in the 28-pin socket, verify that the 28 pins are straight, aligned, evenly spaced, and do not bow excessively from the body. Slightly rocking the pins against a static-free surface can help align the pins, and a small pair of needle-nose pliers can gently straighten pins with little detrimental effect to the device.

There is a semi-circle at one end of the IC's rectangular body that helps us with orientation. The pin directly to the left of this may have a tiny circle above it notating this as “PIN 1” on the IC. The semi circle will align with the notch in the IC socket for perfect connection. The IC will not function if reversed in the socket.

Let the IC sit on the socket, and verify that all 28 pins are evenly nestled in the metal socket ports. A slight rocking motion can help adjust them a bit. If everything is aligned, the IC can be slowly pressed straight down into the socket with a thumb or index fingers. A correctly seated IC will be even in the socket, parallel to the PCB, and live less than a millimeter from the socket's top.

Prepping the Sensors

Most of the pressure sensors I've seen have “solder tabs” as connectors instead of pins. These may be a bit thin to make good contact in the socket. If your pressure sensor has these, it is a simple process to “tin” the 2 solder tabs with a thin even coat of solder. This will add thickness to the two pins, and help make a firm connection when connecting it to the female socket S2.

Though not necessary, I have also found that nipping about 2mm length from the pins on the position sensor makes for a prefect fit on a 24” board. Lay everything out on the body before making this decision.

STEP THREE: Testing the Circuit

The health of your completed circuit can be tested a bit without the installing drivers or mounting the hardware in its final position. However, you may wish to jump to the “Getting Started” document (a separate pdf from the www.straytechnologies.com Resources Wiki) and install the drivers and basic software at this point.

In either case, the board can be minimally tested without the sensors, by plugging in the FTDI (USB-to-serial) cable and powering the unit. The FTDI cable should first be plugged into a USB port to provide power. The 6-pin female end can be connected to J1 (6-pin male header) verifying that the black wire (of the 6 visible colored wires) on the cable header is closest to the dip switches. This usually means “flat side up” on most cables. For other types of FTDI connectors verify that the pin closest to the dip switches is connected to GND of your FTDI connector.

In a perfect build, the red LED should light steadily, with the green led flicking on for a split second followed by a brief pause, and three steady fast blinks. This is the instrument's way of telling us that the Arduino firmware is installed correctly on the chip and ready for action. A quick press of the reset button (SW1) should repeat the blink pattern. If all is good, unplug your device, and move on to mounting the sensors and PCB on the instrument body.

This test verifies the firmware, basic power, crystal, reset switch, and LEDs only. Further testing can happen with software and after the mounting process.

If anything other than this happens: unplug everything, take a break, than hop down to the troubleshooting section of this document or the wiki. You built it, so ANYTHING is repairable!

VERY IMPORTANT: If any parts need re-soldered remove the Atmel chip be for continuing. Heat from further soldering can cause damage to the internals of the IC, even when working in a different area of the circuit board. A tiny screwdriver or blade inserted at the 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. Again, they can be gently bent to a straight position with little harm.

STEP FOUR: Mounting the Sensors and Printed Circuit Board

The mounting of the PCB and sensors to your board or instrument body can seem a little tricky. I suspect better methods will come about as more people build these instruments. Please check the wiki for the most recent methods, hints and discoveries about the process. For now, here is my best description of simplifying the process for a standard layout and mount.

As pictured at the top of this document, the PCB will be mounted with 4 small screws. The sensors connect to the female connectors at the bottom of the PCB, and run the length of the instrument. Keeping everything straight, means a firm connection and nice aesthetic. These sensors are fairly hardy, so don't be afraid to bend them a tiny bit as you apply them to the instrument.

My best suggestion is to plug-in the sensors (without removing the sticky-back covering), and lay the works out on the instrument to verify and mark the best fit. Once you are happy with the layout and everything lays flat, a piece of masking tape can be used as an edge marker on the wood. Tape a straight line perfectly against the edge of your loosely placed position sensor to use as a guide-line while adhering the sensor. A tiny pencil mark could work just as well. This is also your opportunity to mark the screw-holes for the four PCB mounting screws with a small pencil or awl. The sensors and PCB can be temporarily disconnected at this point. Before doing so, you may want to jump to the "Getting Started" document and do a full test, while things are loosely connected on a clean flat non-conductive surface. Remember: intermittent loose connections may cause testing problems that don't exist in a finish-mounted instrument. Passing the "blink-test" is still your best sign of instrument function.

As a perfectionist, I would borrow a drill, and pre-drill tap holes in your instrument body for the PCB mount screws. I recommend #4 x 3/8 screws which couple nicely to a 5/64" tap hole in wood or soft acrylic. It may be possible to self-tap in softer mediums, though the possibility of cracking the body may exist with drier woods. It would also be possible to mount the PCB with a nut and bolt system. The hole size is .125" (1/8 inch or a bit over 3mm).

Once a decision is made on all of this, disconnect the sensors and mount the PCB. The female headers should be close to flush with the instrument body in order to make the sensors sit flat. Up to 1/8" air gap is fine, as there is some flex in the sensor leads. DON'T over-tighten the board mounts. This could result in warping of the PCB or even mini cracks in solder points.

The next step is to mount the long position sensor. The trick is to peel back the mounting tape a bit at a time, while maintaining alignment with your marking tape or pencil line:

Clean all surfaces (and your hands) of dust, oil or dirt. Plug the sensor into the PCB. Verify alignment. Lift the sensor lightly and peel back an inch or so of the protective plastic, exposing the adhesive from the PCB end. If you softly press down while pulling the strip toward the opposite end of the instrument body, you should be able to slowly maintain alignment as you go. Once a bit of adhesive is committed, the rest will follow alignment – so put your real effort into the beginning of the process.

You may be able to lift the whole works and re-align the sensor ONCE, but any sharp angular yank on the sensor could result in cracks in the resistive material printed on it. I use a soft (and overspent) credit card to release any last air-bubbles under the adhesive. The area from the adhesive to the PCB can have quite a bit of flex in it and still operate fine. Don't panic if there is a slight lifting toward the PCB. This won't hurt anything, as long as the sensing area is adhered.

The pressure sensor may "appear" to have a similar adhesive with a black cover over the round area. MOST DO NOT. I found it most simple to plug in this sensor bend it upwards and apply a small thin amount of rubber cement to the bottom of the disk, and the point of contact on the instrument body. Allow it to air dry for a minute, and then press down allowing the two tacky cement surfaces to contact. Let it dry with mild pressure before testing or you may experience some slipping. Any excess rubber cement can be rubbed away into those rubber-cement boogers you played with as a kid.

STEP FIVE: Driver Installation and Final Testing

If you're happy with the instrument at this point, it's time to go to the “Getting Started” document, install drivers, give everything one last test, and make some loud sound.

Troubleshooting

Most symptoms in a new circuit board result from missed soldering points, bridged soldering points (solder touching two contacts) or “cold solder joints” that appear nearly correct, but do not transmit electricity. The first thing to do when troubleshooting is take a break. THEN check for these three things with clear eyes, good light, and maybe even a magnifying glass.

“cold solder joints” can be identified as small bulbs of solder that didinot fill in holes and gaps correctly. The are almost always a dull gray color instead of the proper shiny silver color as a good solder joint appears. They are very easy to correct by simply reheating the offending pin and pad until the solder pulls into a concave form.

Further troubleshooting can be categorized into areas depending on the symptoms. Some basic problems are below, but worse problems can likely be sorted out with some deduction and a visit to the forums at www.straytechnology.com. If you post a problem, try to post as much descriptive info as possible and maybe even a picture.

SYMPTOMS:

Red LED Doesn't Light:

Verify polarity (direction of the LED). Potential areas to check are all connections of D2, R5, J1 (pin 6 and 4), and the 28 pin socket.

Green LED Doesn't Blink:

Verify polarity (direction of LED). Potential areas to check are connections of D1, R4, the 28 pin socket. This could also be a symptom of firmware being incorrectly loaded on the ATMEL IC. Check the troubleshooting and update sections of the Getting Started document for more info on this possibility.

RESET switch does nothing.

Areas to verify are the connecting pins on SW1, R1, C1 and the 28 pin socket

Potentiometer or sensors do nothing.

Check all connections of the offending sensor, and the 28 pin socket. For S1 and S2, check R2 and R3 respectively as well.

Possible Advanced Modifications

As discussed, there is a huge potential for modification. This brief discussion will serve as a starting point for ideas and which could then be shared on the wiki as people dig deeper into this instrument. Modifications could generally be separated into three categories with some minor overlap:

Aesthetic: cases, improved body materials, paint, stencils, body shape.

This can all happen on an individual basis, and allows for some wide-open customization. One only needs to do a google image search of “Diddley Bow” or “Cigar Box guitar” to see a mass amount of potential borrowed from our devices analog cousin.

Interface: sensor types, sensor positions, added buttons, knobs, or new sensors

As the “Arduino” sees most of these sensors as an “analog input” there is potential to replace any sensor, use different shape sensors, and relocate them using ribbon cable. A simple example would be to use a long pressure sensor (I've tested *Sparkfun.com SEN-09674*) directly underneath the softPot sensor and rerouting it with a small run of 2-wire ribbon cable folded under the PCB. Though not all settings would be useful, this would create one pressure+position sensitive strip similar to a traditional 80's era ribbon controller, or a long 1D mouse-pad.

Function: special effects, new settings, new uses accompanying new sensors

The heart of these possible changes will be in software. I expect to catalog different versions of the “Arduino” firmware on the wiki as designs progress.