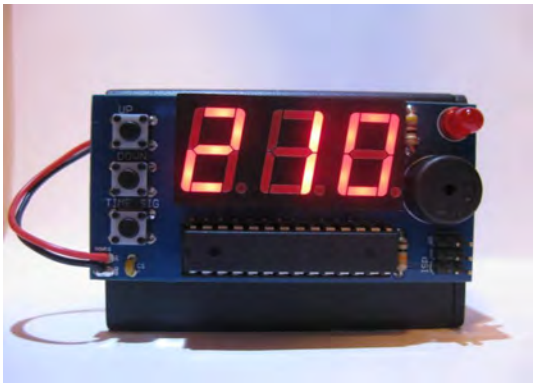
Metronome Kit

PART NO. 2168325



The metronome kit allows you to build your own working electronic metronome. Features include a small speaker, flashing LED, and the ability to switch between several different time signature options. The kit includes a three digit seven-segment display to display the speed in beats per minute and runs off of three AA batteries. This is a great kit for aspiring musicians interested in stepping into electronics. And, since all components are through hole, it's a great way to develop your soldering skills and build something fun at the same time.

Time Required: 1 hour depending on experience

Experience Level: Beginner

Required tools and parts:

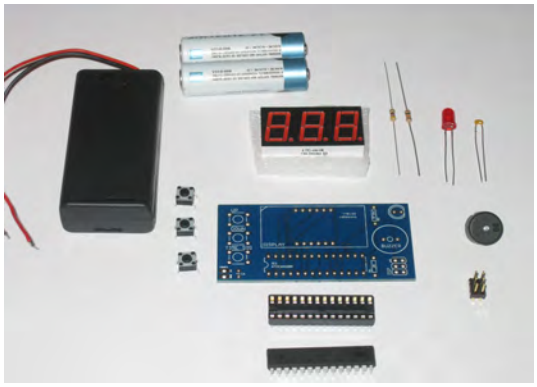
Soldering Iron
Solder
Velcro (Optional)
Wire Cutters
Needle Nose Pliers

Bill of Materials:

Qty	Jameco SKU	Component Name
1	2139111	ATMega328p Microcontroller
Pre-programmed with the metronome code.		
1	333973	Red LED
1	690662	150 Ohm Resistor
1	151116	0.1uF Capacitor
1	2125229	Three Digit Seven Segment Display
3	119011	Tactile Switch
1	2098523	Piezo Speaker
1	115035	ISP Header
1	216144	3 AA Battery Holder
4	2112444	AA Battery
1	2157167	10k Ohm Resistor
1	112300	IC Socket

Step 1 - Verify Parts

Remove all parts from their packaging and lay them out. Be sure that all the components that are supposed to come with the kit are there. (Note the image shows 2 batteries, however, your kit should include 4). Extra components are a bonus.

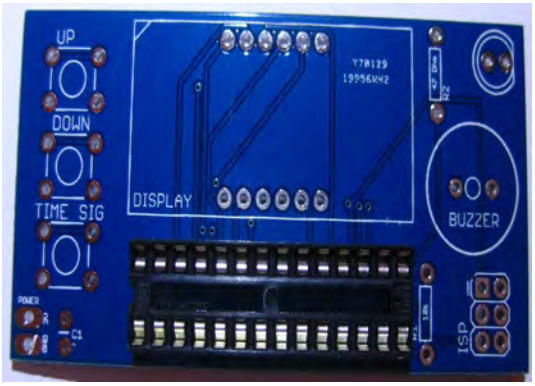


Step 2 - Acquire Tools

To complete this project you will need a soldering iron, solder, wire cutters, and needle nose pliers. Gather theses components and have them ready to use.

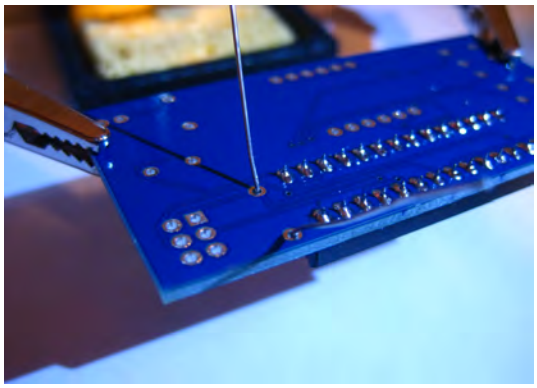
Step 3 - Solder in the Microcontroller Socket

Take the microcontroller socket and place it in the spot marked for it on the Printed Circuit Board (PCB). Be sure that the notch on the socket matches the notch on the PCB. I recommend resting the board on a flat surface and solder one pin at one end and another pin at the other to hold the socket in place. Then solder the remaining pins as you usually would.



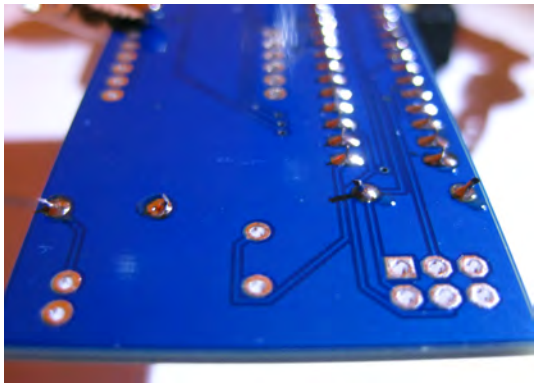
Step 4 - Place Resistor R1

Take the 10k ohm resistor (this is the resistor marked with brown-black-orange-gold stripes) and bend the leads so they are perpendicular to the resistor body. The resistor should almost look like a staple at this point. Try to bend the leads as close to the resistor body as possible. I recommend using the needle nose pliers for this. Once the resistor is bent, insert it into the holes in the area marked for R1 on the PCB. Turn the board over and bend the leads out so the resistor stays in place while you solder, and then solder it in place. Once soldered, use the wire cutters to trim the leads close to the board.



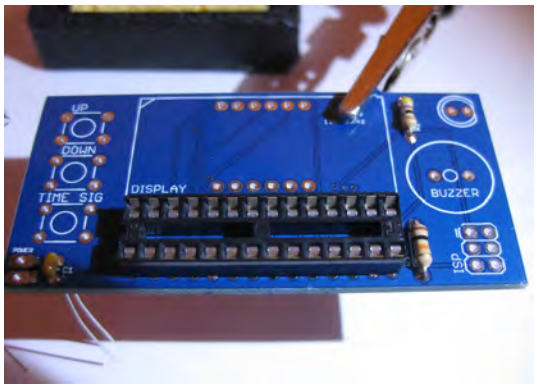
Step 5 - Place Resistor R2

Now take the 150 ohm resistor (this is the one with the brown-green-brown-gold bands) and bend it in the same way as the previous resistor. This is the current limiting resistor for the LED. It prevents too much current from flowing through the LED and damaging it. Place in the holes corresponding to resistor R2 and solder in place and trim the leads.



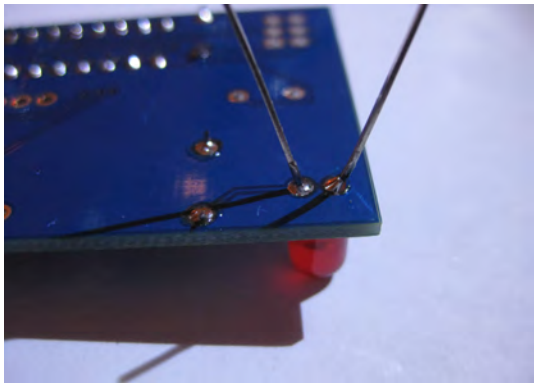
Step 6 - Place Decoupling Capacitor

Take the 0.1 uF capacitor and place it in the holes corresponding to C1 on the PCB. This is the decoupling capacitor and basically helps smooth the power to the microcontroller. This is a ceramic capacitor and therefore is not polarized, so it can be placed in any orientation. Solder it into place and trim the leads.



Step 7 - Place the LED

LED's are diodes which only allow current to flow in one direction, so it is important that it is placed in the correct orientation. Notice the outline of the LED on the PCB. You'll see that the right of the circle is flat. This indicates the cathode or negative side. On the LED there is a flat spot on the lens, but the easiest way to tell the orientation is to look at the length of the legs. The short leg corresponds to the cathode and the long leg, the anode. Insert the LED into the PCB with the short leg in the hole closest to the flat side and solder into place.



Step 8 - Place the Buttons

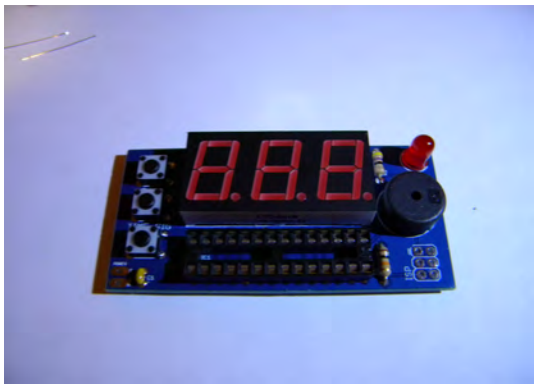
Take the three buttons and place one in each of the locations on the left side of the PCB marked with the square and the labels for Up, Down, and Time Sig. These should slide in easily and stay in place because of the way the legs are bent. Solder them into place.

Step 9 - Place the Piezo Speaker

Take the small piezo speaker and place in the holes in the area marked buzzer. The orientation does not matter, so either lead can go in either hole. If you flip the board upside down and place on a flat surface, it will be easier to solder. Work quickly when soldering this part as to not apply too much heat to the speaker.

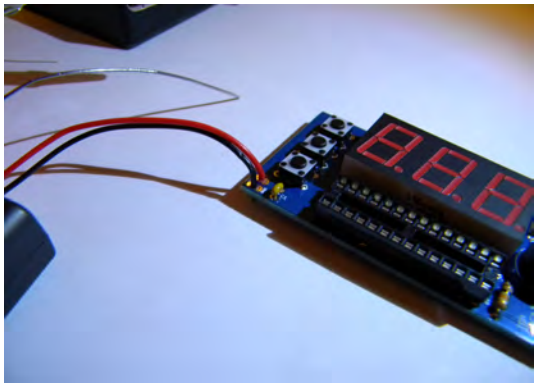
Step 10 - Place the Display

Take the three digit seven segment display and place it in the holes for the display on the PCB. There will be one extra hole on the bottom right side. Don't worry, your display isn't broken, that pin just doesn't exist. It will probably be easiest to solder if you turn the PCB upside down and lay it on a flat surface.



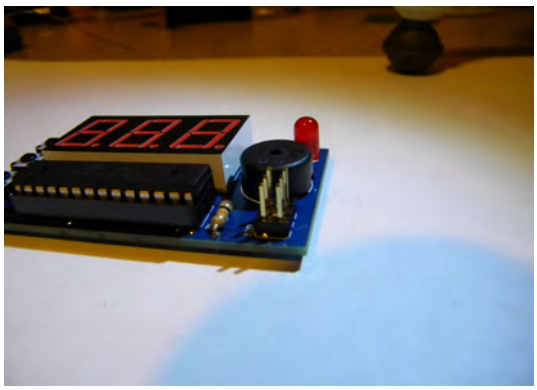
Step 11 - Connect the Battery Pack

Now you're ready to attach the battery pack. The battery pack should have long red and black wires coming from it. If you want, you can trim these wires, for they are quite long and you will not necessarily need them that long. Insert the wires into the two pads located at the lower left corner of the PCB. Be sure the red wire is placed in the hole marked V+ and the black wire is placed in the hole marked GND and solder into place. Trim the wires if they stick out too far past the solder.



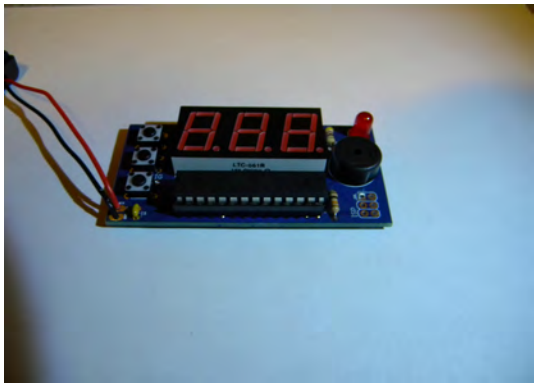
Step 12 - Place Headers (Optional)

If you're interested in modifying the code pre-programmed on the microcontroller then this step is necessary, otherwise, you may skip it. The ISP headers will allow you to interface with the microcontroller. You will, however, need to have a special device to program the microcontroller. These can be easily found at most major electronic distributors. Place the two by three header in the holes marked ISP and solder it into place.



Step 13 - Insert the Microcontroller

It's finally time to insert the brains of this kit. Take a look at the Atmega328p, for you may need to bend the pins so they are more perpendicular to the microcontroller (uC) body. To do this, lay the the uC on its side so all the pins are laying on a flat surface. Holding just the uC body, push slightly to bend the pins perpendicular. Do this on both sides. Then place the uC in the socket, once again paying attention to line up the notches on both the socket and the uC. It will take a bit of force to mount in the socket, so don't be afraid to apply force.



Step 14 - Install the Batteries

Place the 3 AA batteries into the battery holder.

Step 15 - Use

You'll all done. You can go and use your new metronome now or you could add Velcro to the back of the PCB and the battery holder so you can stick the unit together. If you're interested in learning how the code that drives the ATmega328p works or want to try your own hand at programming the device, head over to http://www.instructables.com/id/Electronic-Metronome/step3/The-Software/ and download the code. Please note that if you are planning on modifying the code you will need an AVR programmer to program the ATmega328p.