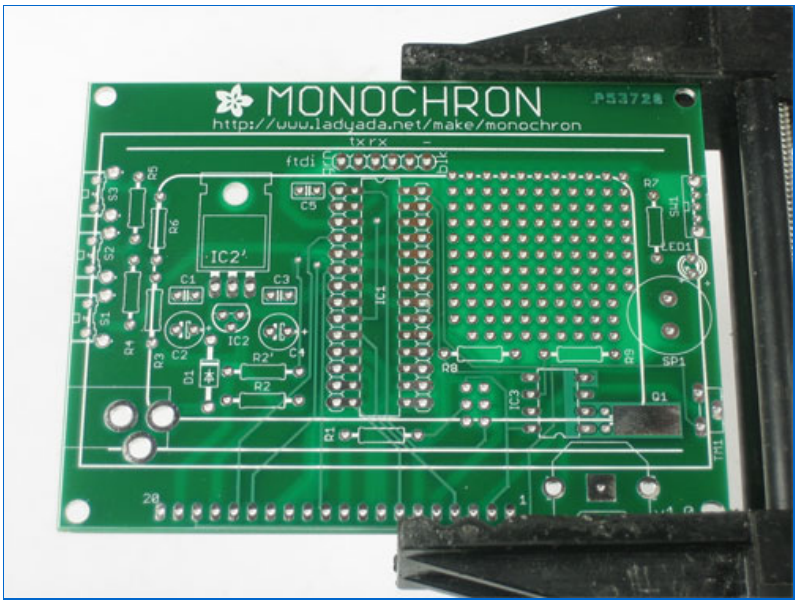
MONOCHRON

Assembly instructions

May 17, 2011 20:07

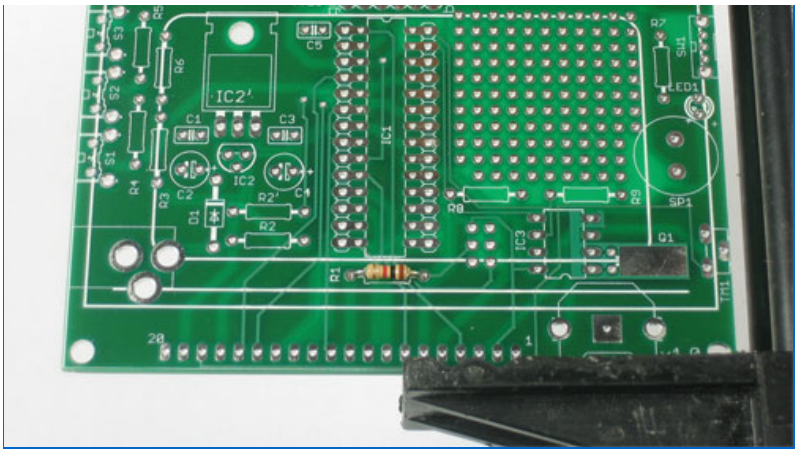
Here are the step by step instructions...just want to remind you not to solder in the 6-pin FTDI header or ISP header!

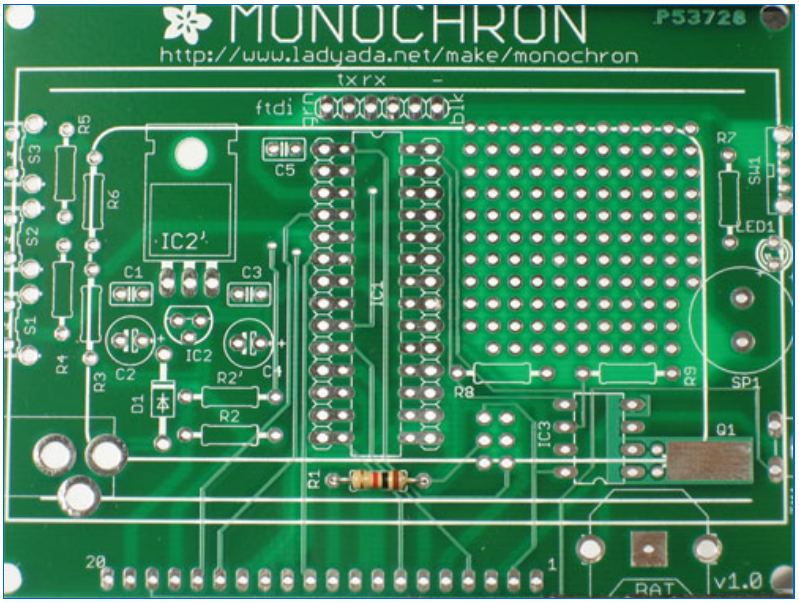


Get ready by checking all your parts against the Bill of Materials (parts list). Once you are sure you have everything, prepare your workspace by heating up the soldering iron, wetting the sponge and arranging your tools and parts so they will be convenient.

OK! Lets start! First thing we will solder in is the resistor **R1**, which has a 1Kohm value. You can tell which one this is because it has Brown Black Red Gold stripes on it. Bend this resistor into a staple as shown. Then slip it into the spot in the lower middle, right over the matching silkscreen that is labeled **R1**. Resistors are not 'polarized' so you can place them either way and they work the same. The resistor should sit right up against the PCB

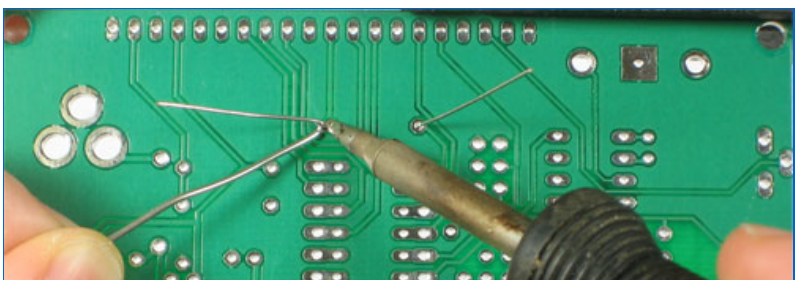
This resistor is used to allow the microcontroller to be programed with an In-System Programmer while the LCD is connected (they share a pin)

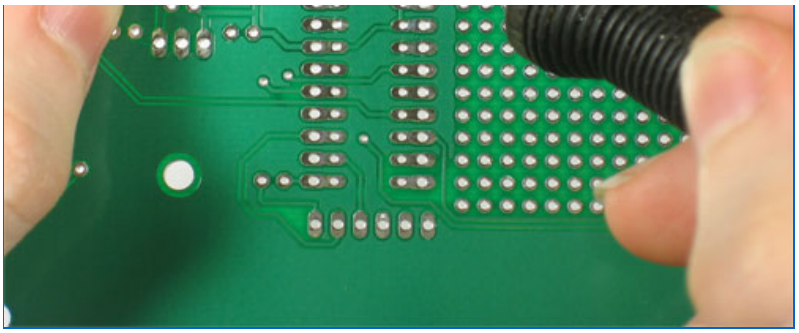


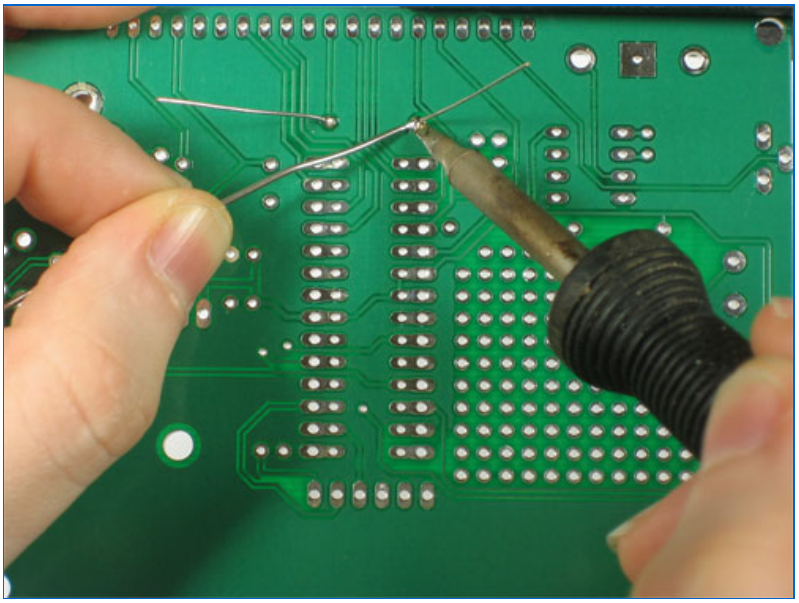


Bend the leads out of the staple so it holds the part in place and flip over the PCB.

Using your soldering iron tip, press and heat both the pad (the silver ring around the hole) and lead (wire) at the same time for 2 or 3 seconds. Then poke the end of the solder in to create a nice solder joint. Do this for both leads.

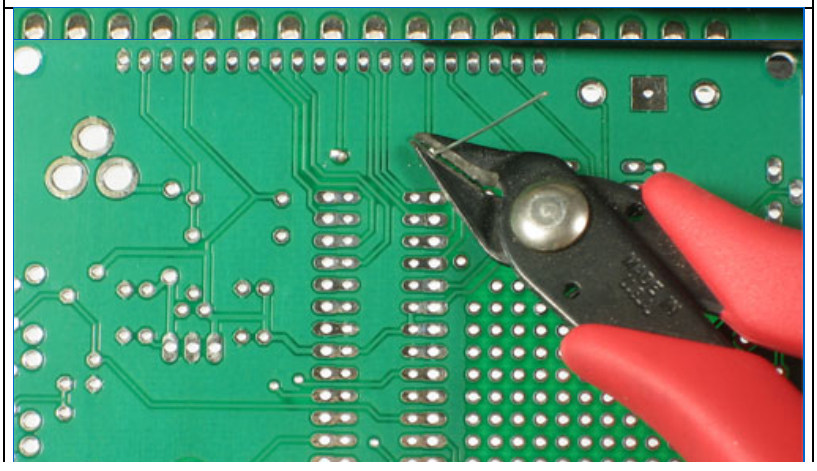


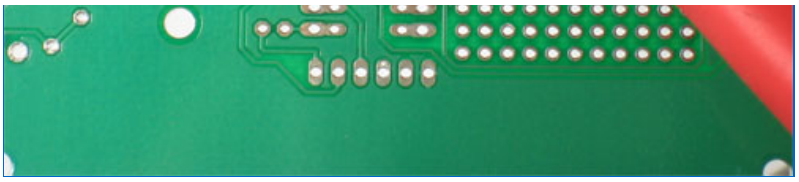


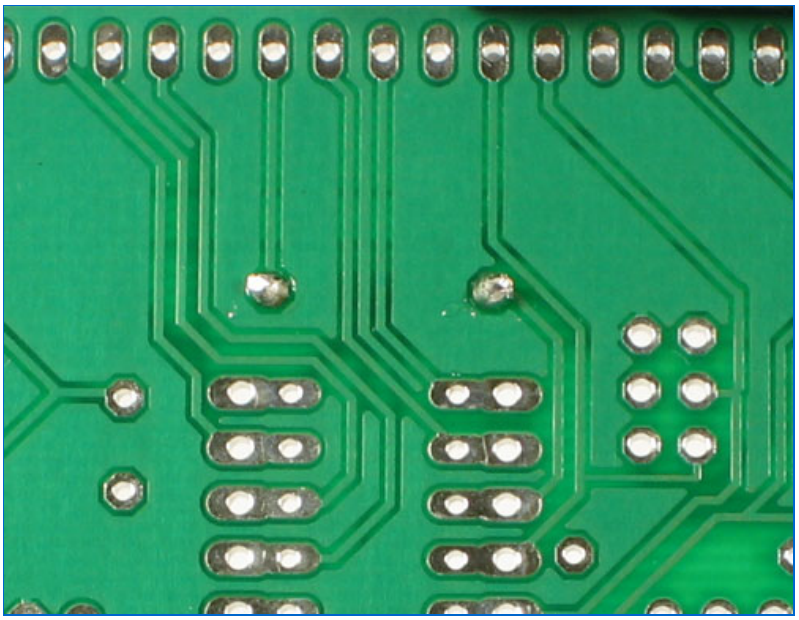


Using your diagonal cutters, cut off the long leads just above the solder joint.

You should have nice shiny solder points that are rounded and fill the entire pad.



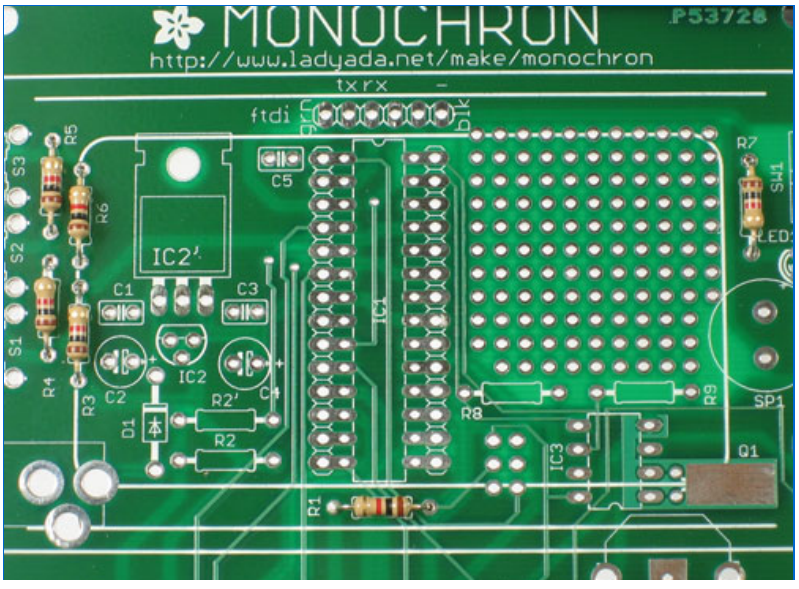


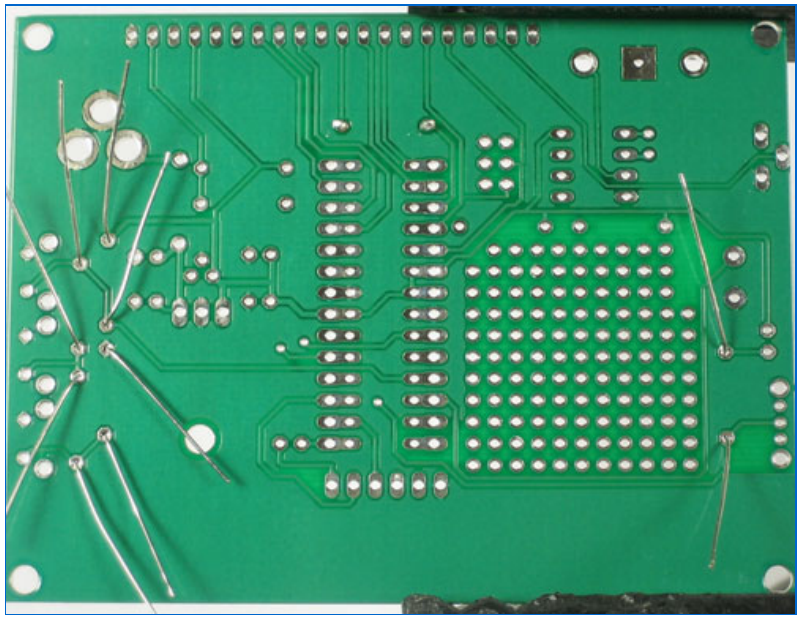


Now we will continue by placing the remaining 5 1K resistors.

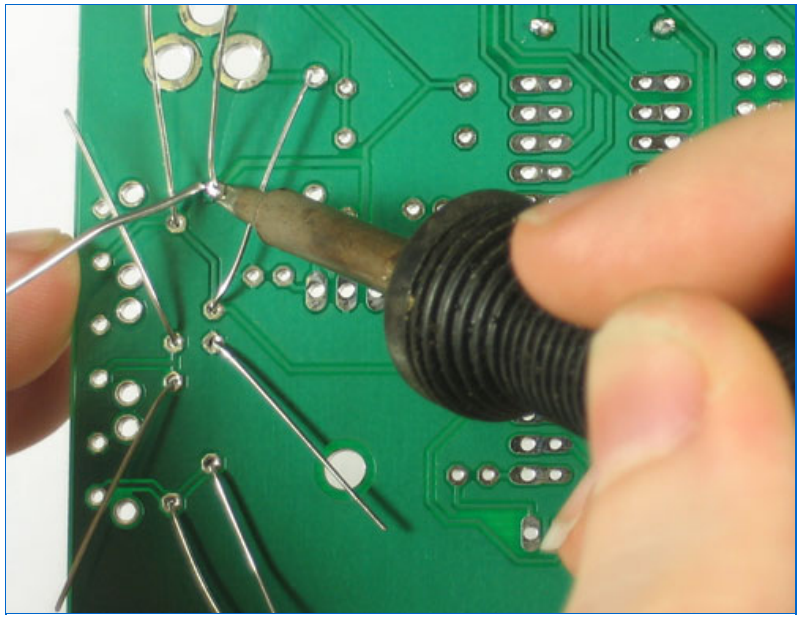
Place **R3**, **R4**, **R5**, and **R6**. These resistors make up a multiplexed button-reading circuit that allows the chip to listen to 3 buttons using only one analog pin.

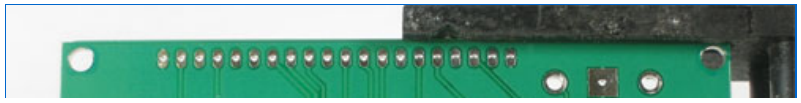
Place **R7** this is the current limiting resistor for **LED1** (the alarm indicator)

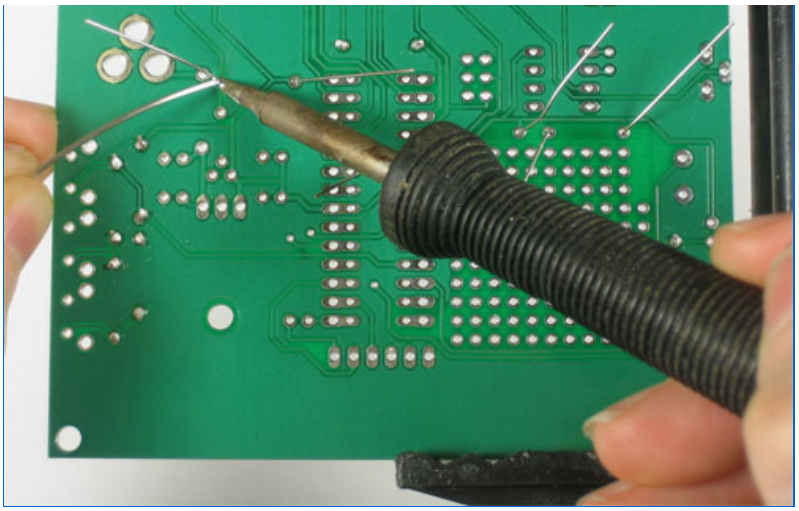


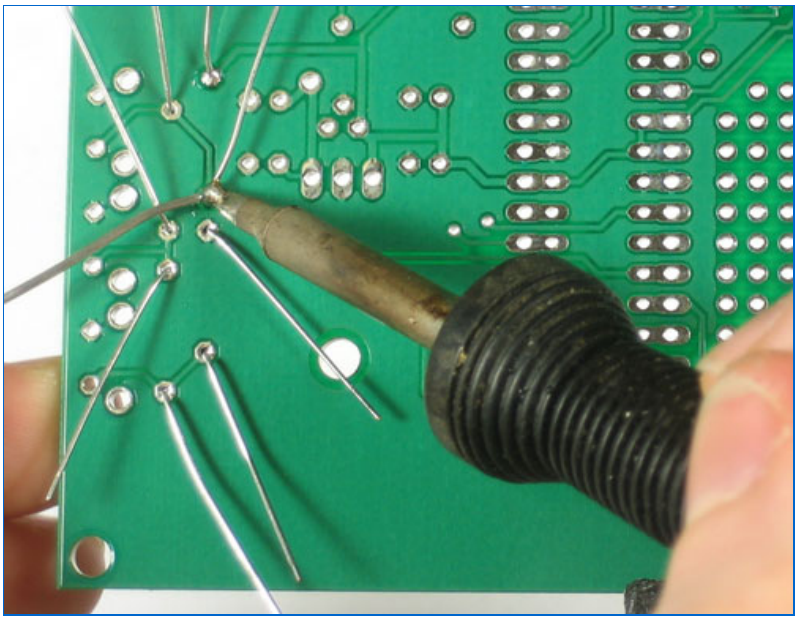


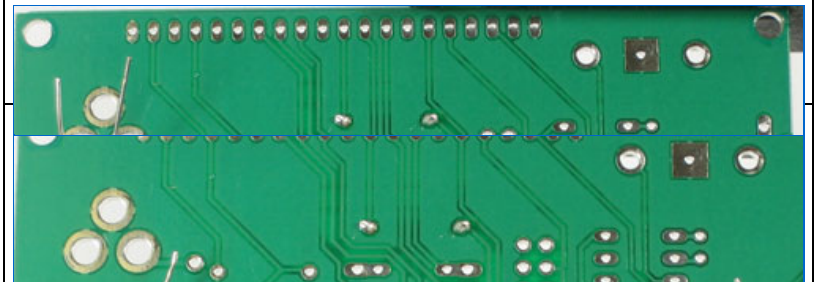
Flip over and solder the resistors just like before, making sure to heat each joint and fill it with solder completely.



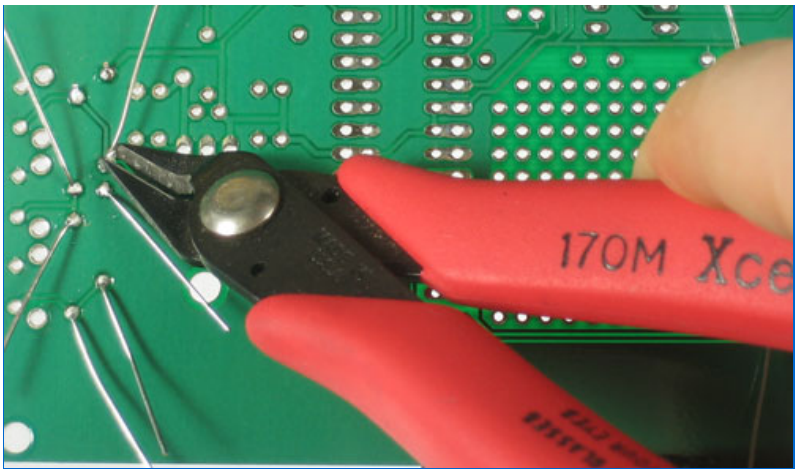


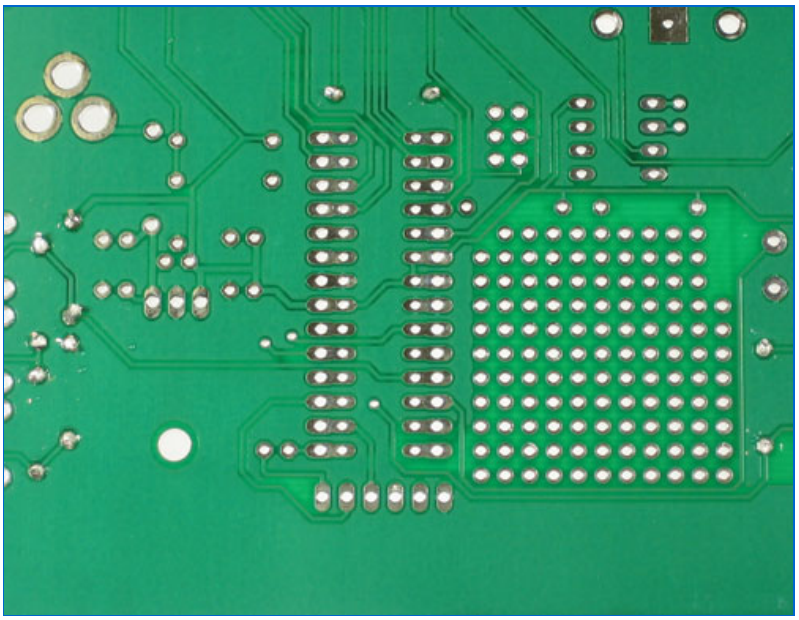






Then clip all of the leads





Next we will solder in last 3 resistors.

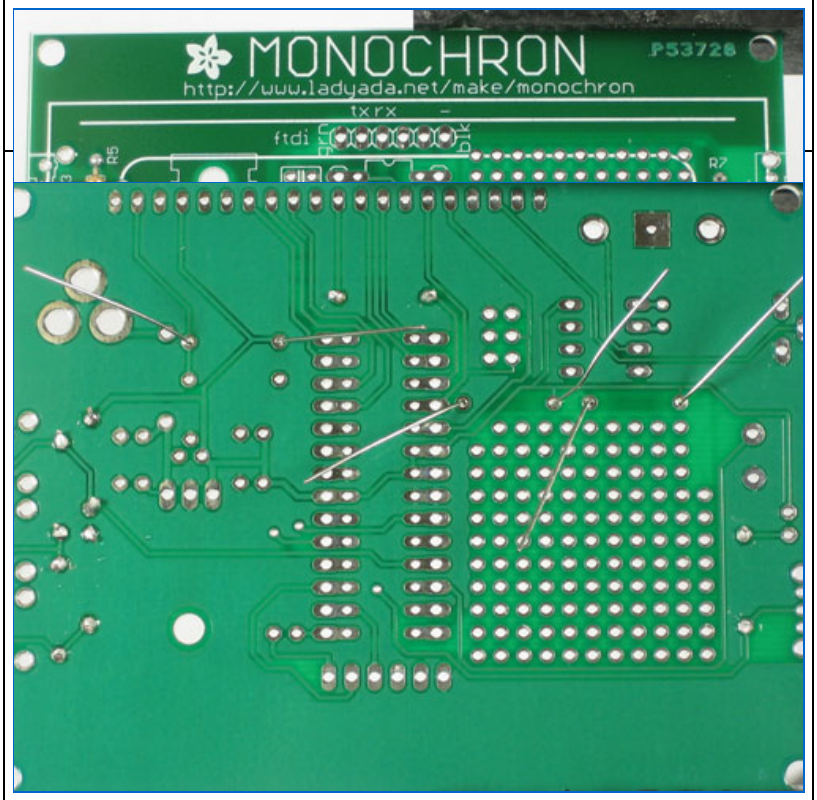
R8 and **R9** are 2.2K (red red red gold) resistors that are used as i2c pull-ups for the real time clock (the datalines require pull-up resistors).

R2' is a 100 ohm resistor (brown black brown gold) that sets the brightness for the LCD.

Please note that this resitor can be soldered into **R2** (fixed brightness) or **R2'** (software controllable). In these photos we soldered them into **R2** but we suggest you use **R2'** which will allow you to control the brightness easily (it turned out the

LCD backlight is very quite a bit

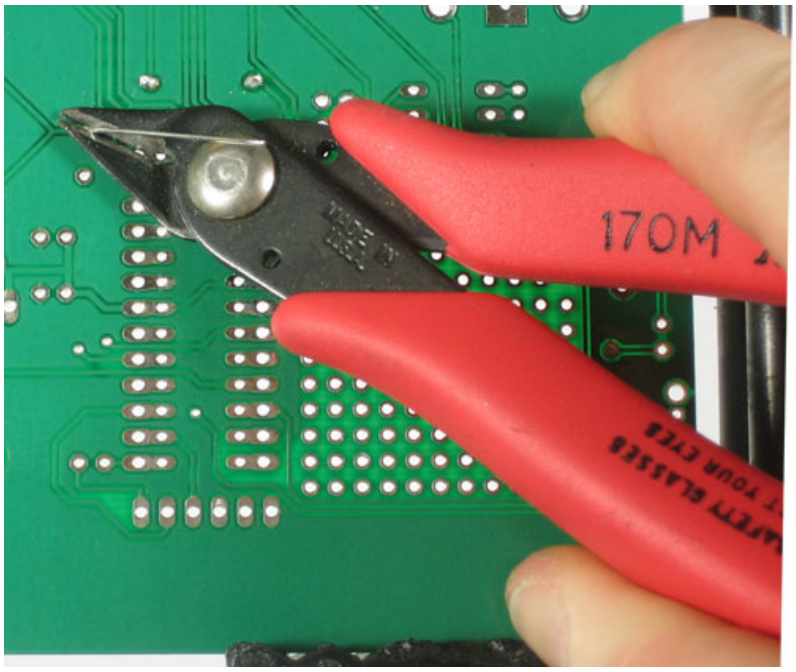
LCD backlights vary quite a bit from one to the other). **So please ignore the incorrect resistor location here and the use R2' slot!**



Flip over the PCB and solder in the resistors.

(Don't forget to use **R2'** not **R2**!)

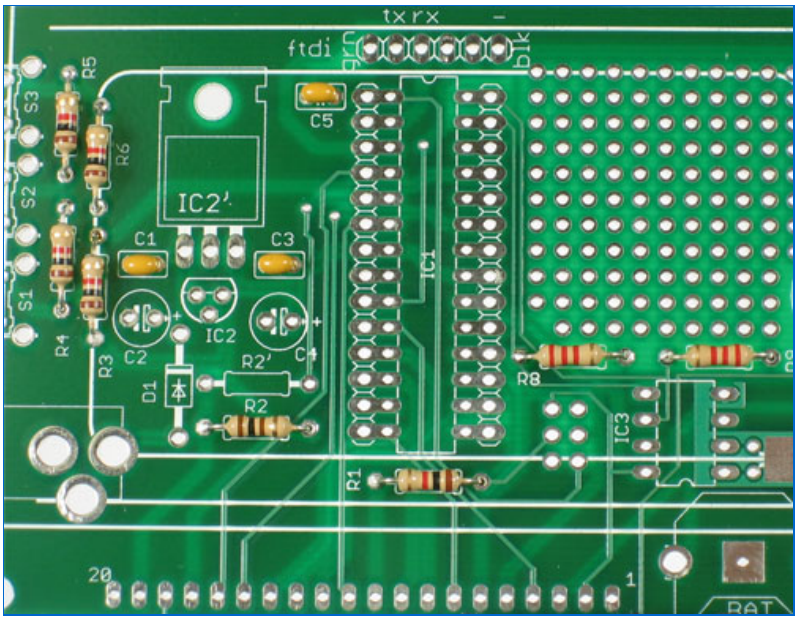
Then clip them short

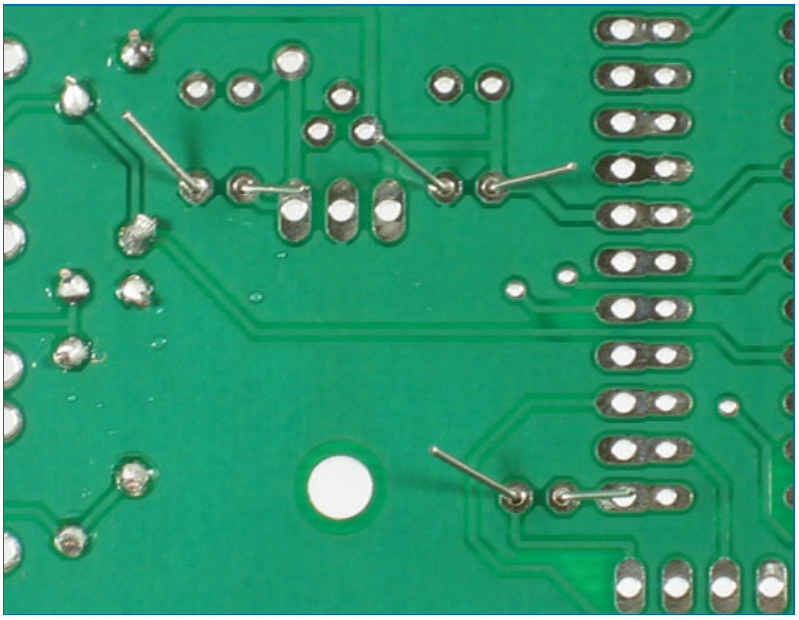


Now that we are done with the resistors its time to solder in the ceramic capacitors. Capacitors are used to smooth out power supply ripples and also to block DC voltages.

C1 is the input (9V) filter capacitor, **C3** is the output (5V) filter capacitor. **C5** is used to program the chip using an FTDI chip, it blocks the DC 'reset' pin and turns it into a pulse.

Ceramic capacitors, like resistors

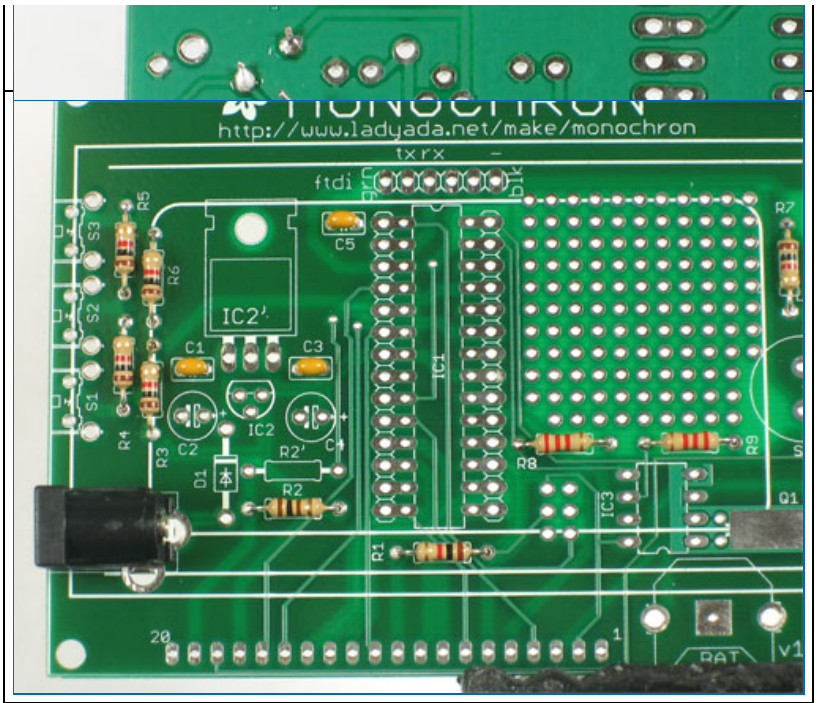




Ceramic capacitors, like resistors, are not polarized so they can go in 'either way'

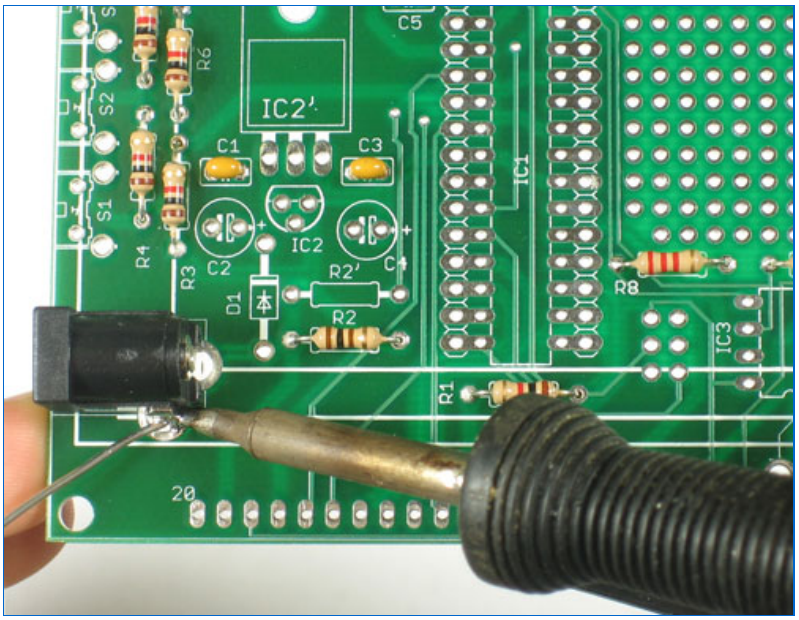
Flip over the board and solder in the 3 ceramic capacitors

Then clip the leads (not shown)



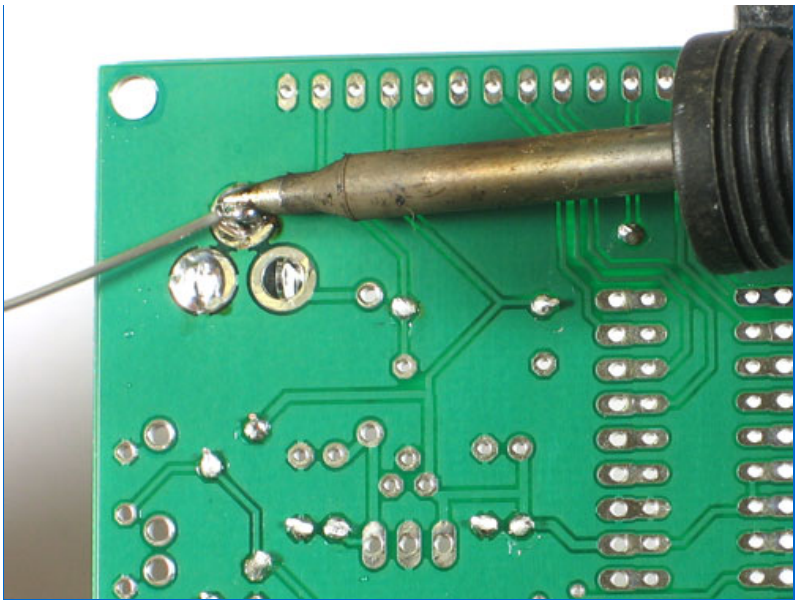
Now we are ready to build the power supply. The power supply is what takes 9V from the outside world (from power plug) and regulates it down to 5V that is safe for the microcontroller to run at.

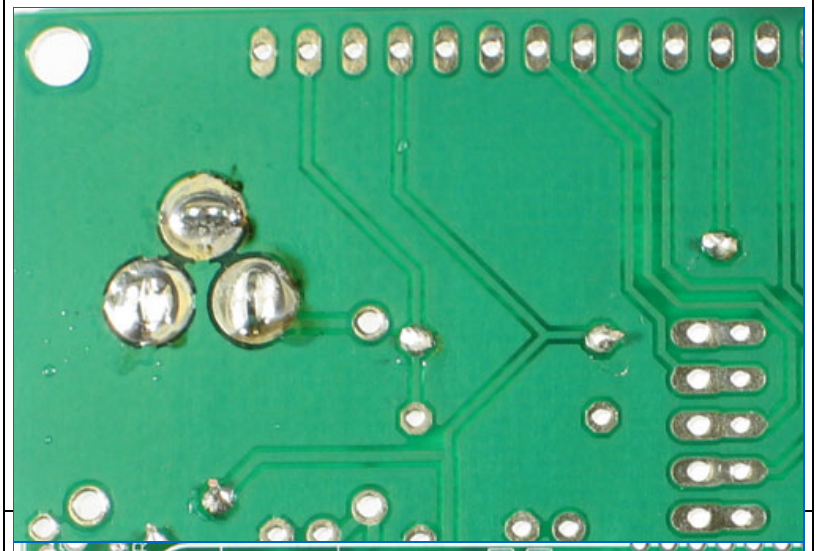
The first part is the 2.1mm DC jack. This is the mechanical connection. It fits in the lower left hand side of the PCB and you can snap it into place (may need a bit of a squeeze) - make sure it sits flat against the circuit board!

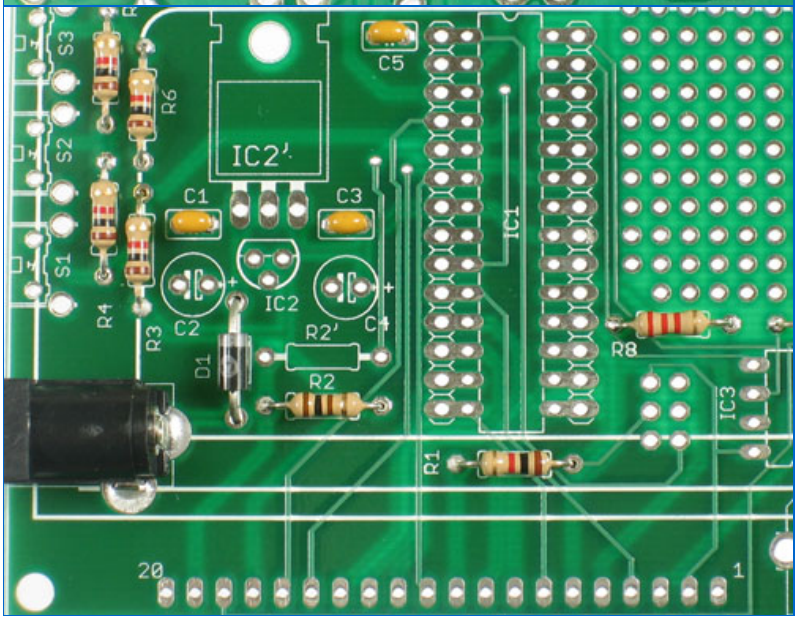


If it doesnt snap in place, you can keep it from falling out by 'tack' soldering it a little from the top on one of the pins

Now flip over the PCB and solder the three big pins. Make sure to heat up the pads and pins for a few seconds before shoving tons of solder in there. This is a mechanical solder connection so you really want to get the big round pads filled up



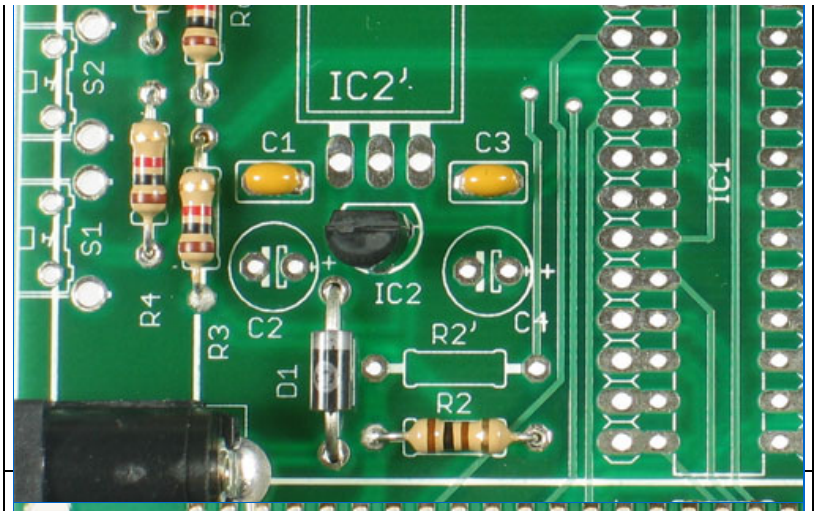




Next is **D1** the 1N4001 protection diode. Diodes are semiconductors, in particular a diode will only pass current in one direction. This protects the power supply and micro from if someone plugs in a power supply that is AC or negative polarity. Instead of going poof, the diode will block the negative voltage.

Because diodes only conduct in one direction, its important to put it in right. Look for a white stripe at one end, this is the **cathode**, now look on the PCB, there is a silkscreened diode and one side also has a white stripe, make sure to match those up!

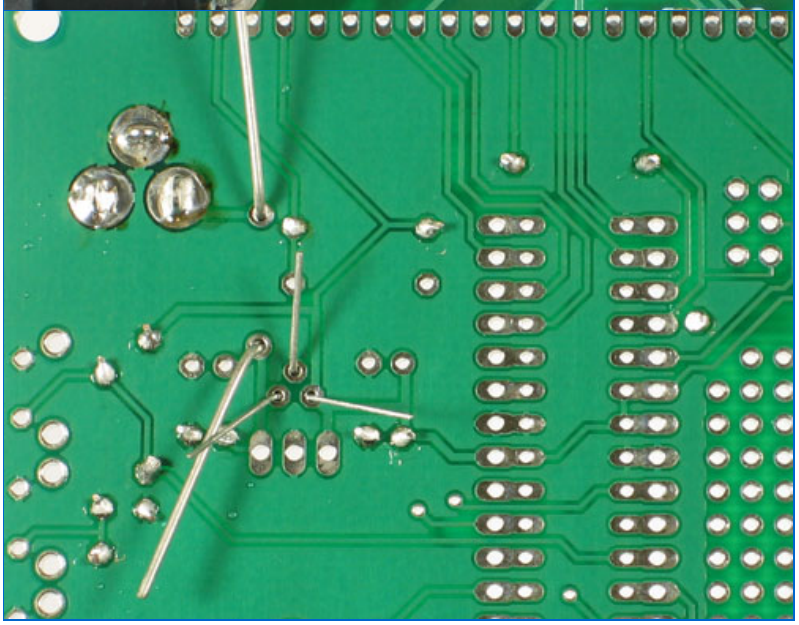
After the diode is the 7805 voltage regulator **IC2**. The 7805 regulators up to 17VDC down to a nice steady 5V. Because we are using a good quality LCD with an LED backlight, we dont need a big



honking power supply chip. The TO-92 package 7805 can supply about 100mA

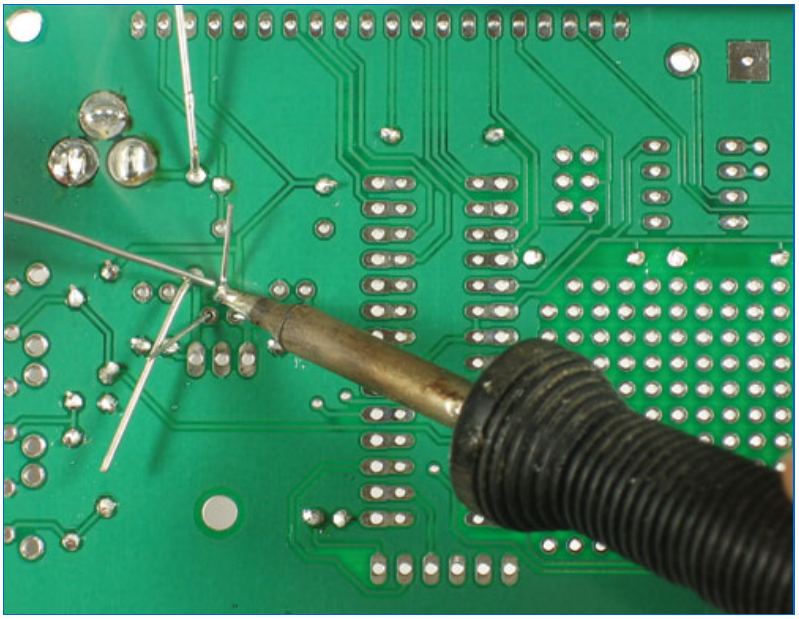
However, if you end up doing mods or want to use an LCD with an EL backlight, you may end up using **IC2'** which is the TO-220 package version and can provide 1.0 Amp of current!

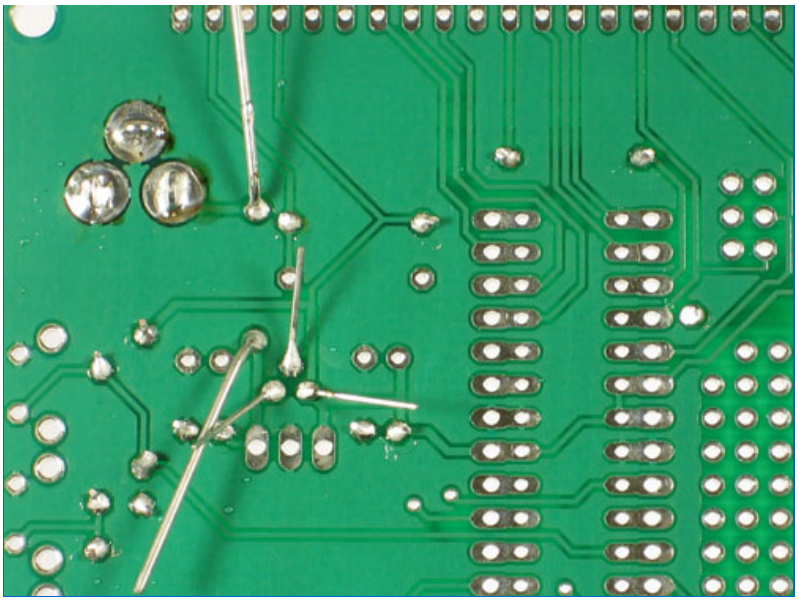
OK, so the regulator has a flat side and a round side, the three pins go in the pads, and the outline on the silkscreen should match the shape of the regulator



Now flip over the PCB and solder in the diode and regulator

Clip the leads (not shown)





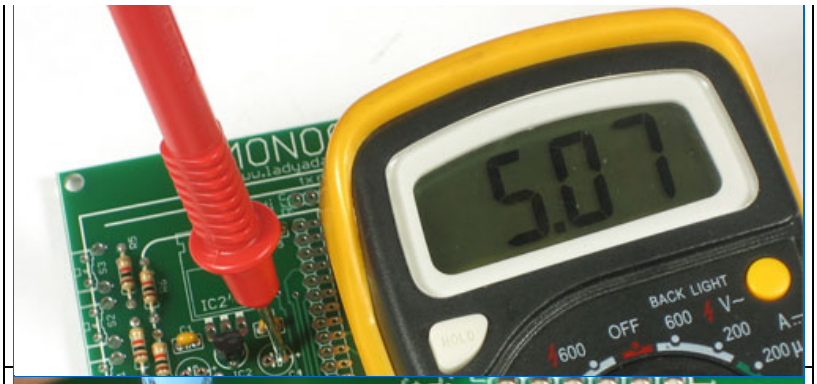
Now we are ready to do our first test. Clear off your table and make sure there are no wires or bits that could short out the PCB.

Place the PCB down and insert the power supply plug into the jack all the way. The jack should not move at all, if you soldered it enough. Now using your multimeter in voltage mode, measure between the two points shown (you can zoom by clicking on the picture)

You should get 4.9 to 5.1VDC

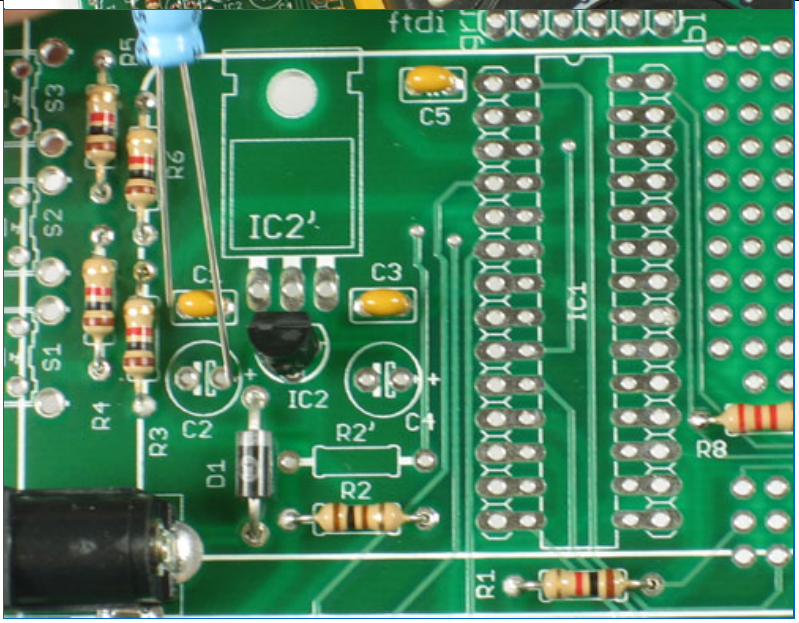
If you **dont** then stop, check your work. Make sure your multimeter has a fresh battery in it, and post in the forum if you can't get it to

display 5V. If the voltage is too



high or low, it could damage your kit, you should not continue.

Once you are done, unplug the power supply and put away the meter.

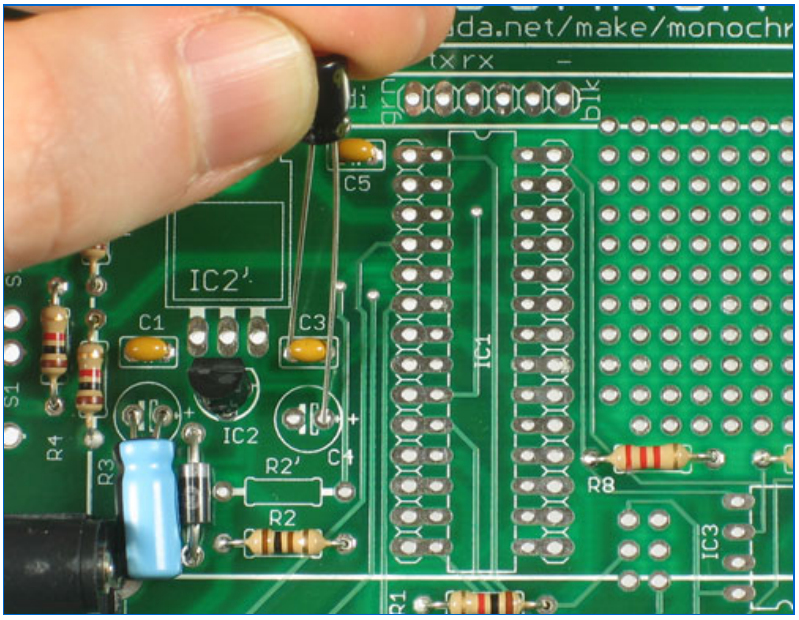


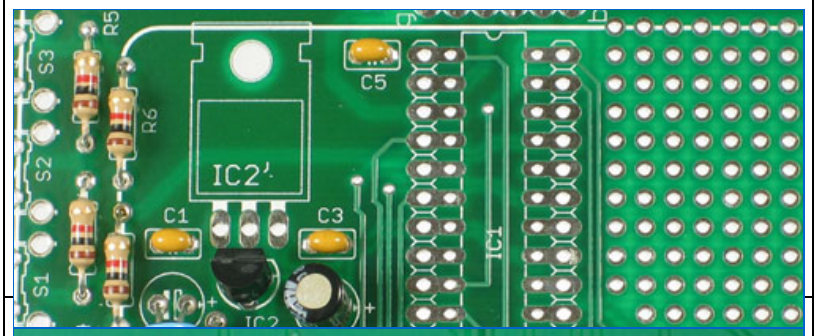
Now we will solder in the two electrolytic capacitors **C2** and **C4**

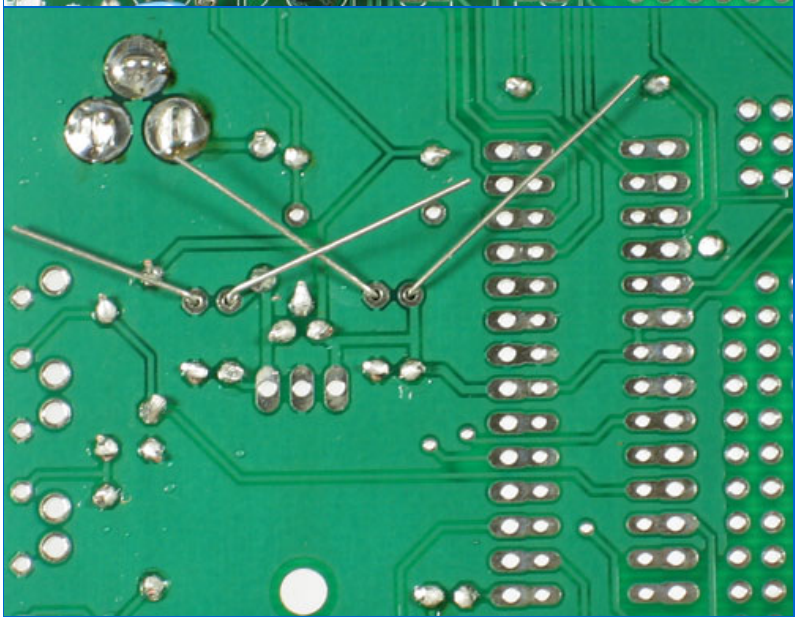
C2 is the 47uF/25V electrolytic capacitor. This capacitor smooths out any large ripples in power coming into the kit. Electrolytic capacitors are *polarized* which means they must be placed correctly or they wont work at all. If you look at the capacitor you'll notice one leg is longer than the other, this is the positive (+) lead. Make sure this lead goes into the pad silkscreened with a +. See left for how to place the capacitor.

Bend **C2** down so that it doesnt stick up so much.

Next is **C4**, a 100uF/6.3V electrolytic, which is the capacitor which helps reduce noise on the regulated 5V supply. It is electrolytic so make sure its placed correctly.

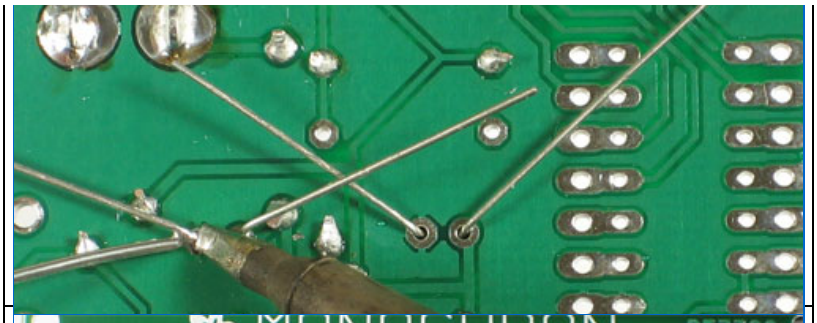


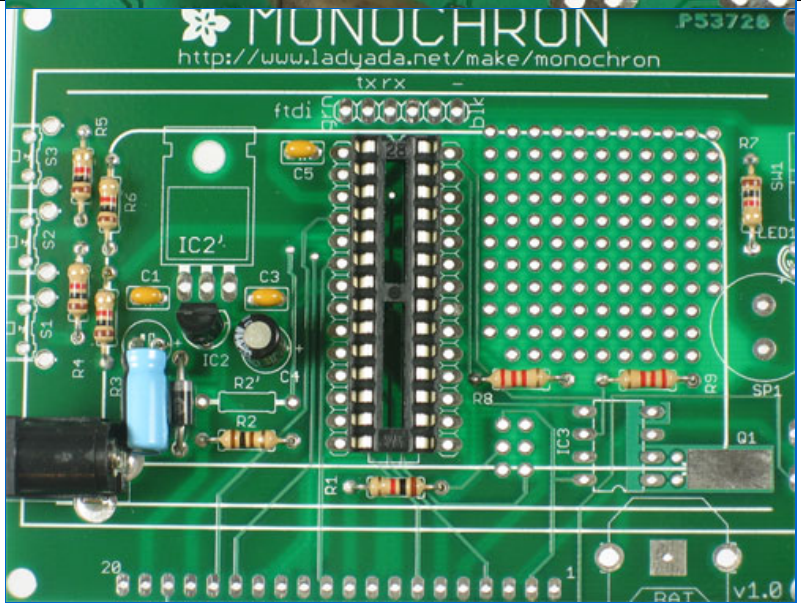




Once the electrolytic capacitors are placed correctly, bend out the leads and solder them in place.

Use the diagonal cutters to clip the leads short

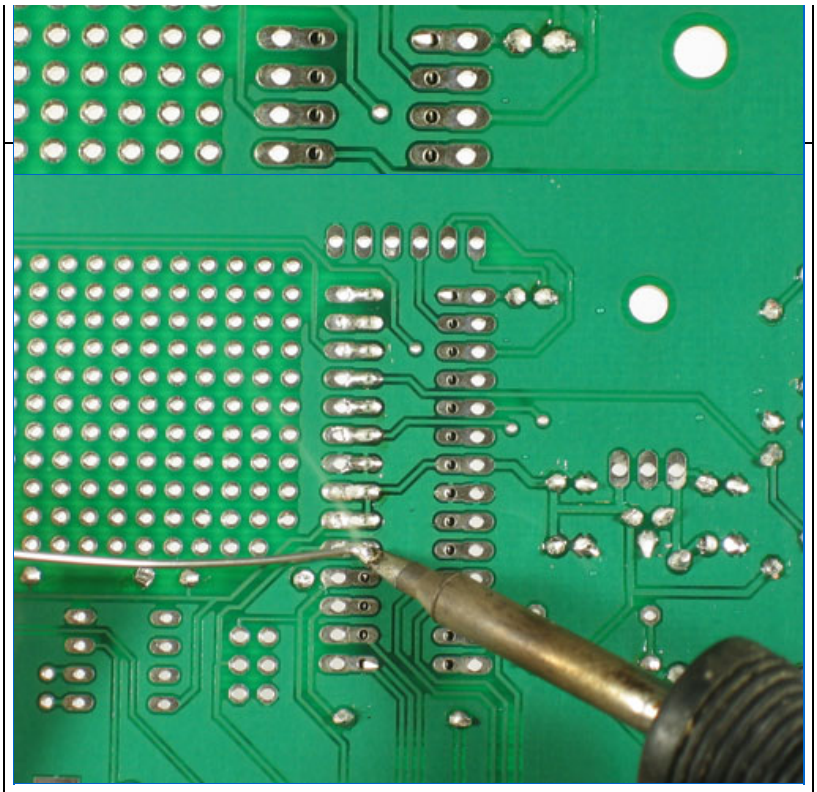


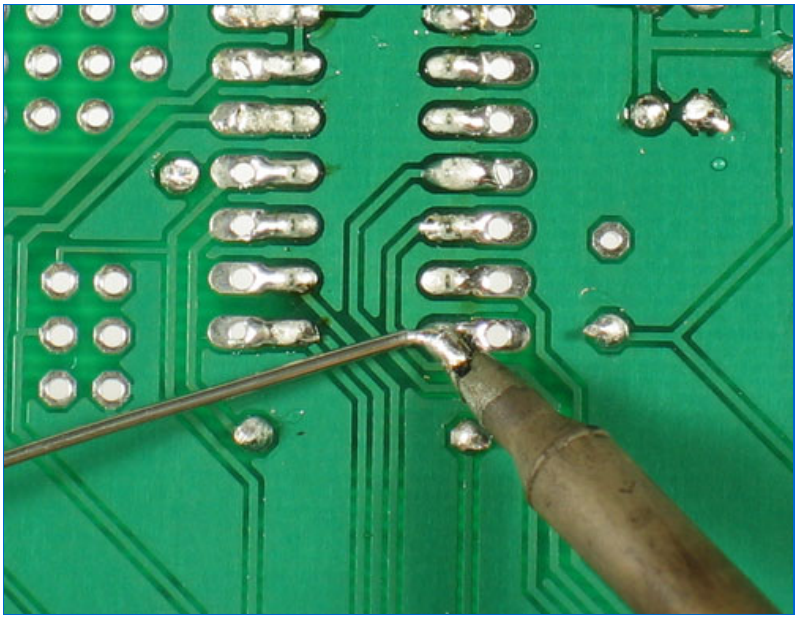


Now place the IC socket. The socket protects the microcontroller chip and allows it to be replaced if necessary.

The socket has a U-shaped notch in one end. Make sure that this notch matches the U-shaped notch in the silkscreen, see the image to the left if you're not sure. If you end up putting the socket in backwards, don't fret. Its not essential that it is in right, but it will help you if you have to replace the chip.

You can keep the socket in place with tape or if you have long fingernails, by bending over two of the little legs to hold it in place.

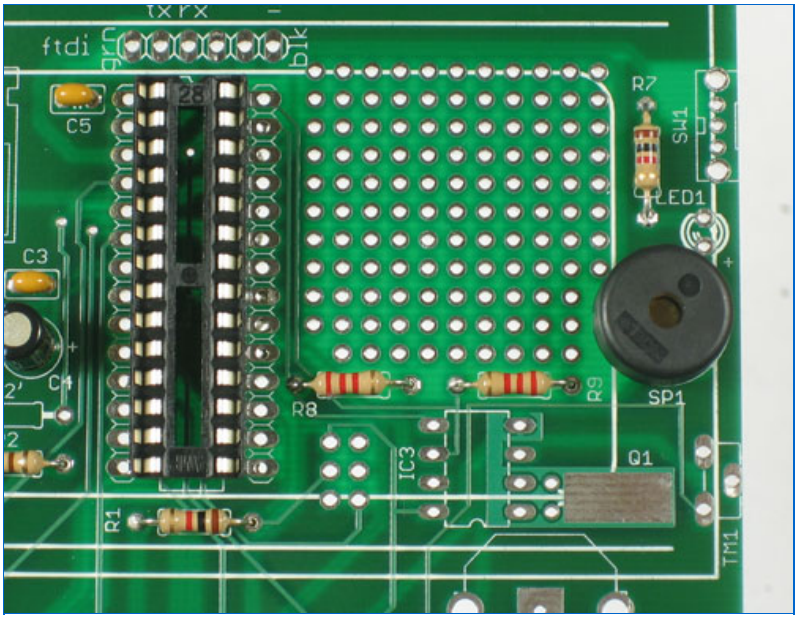


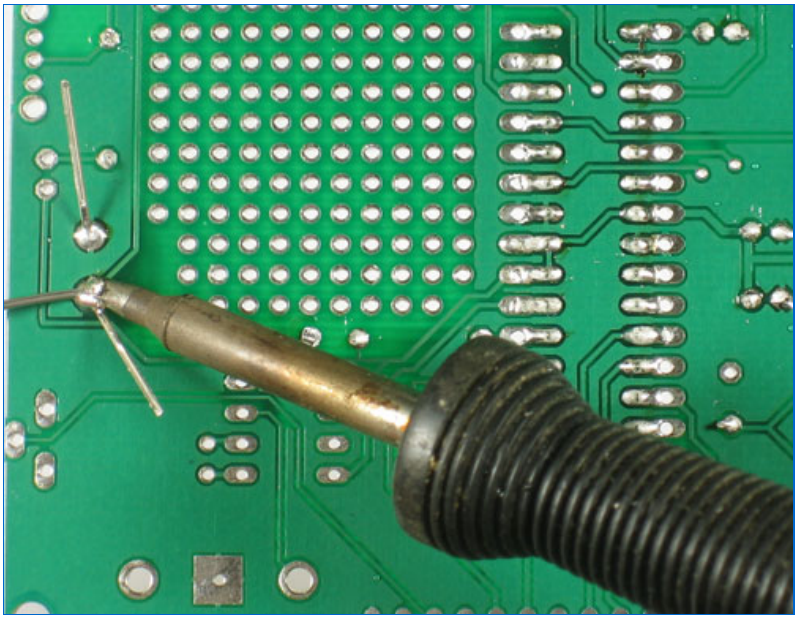


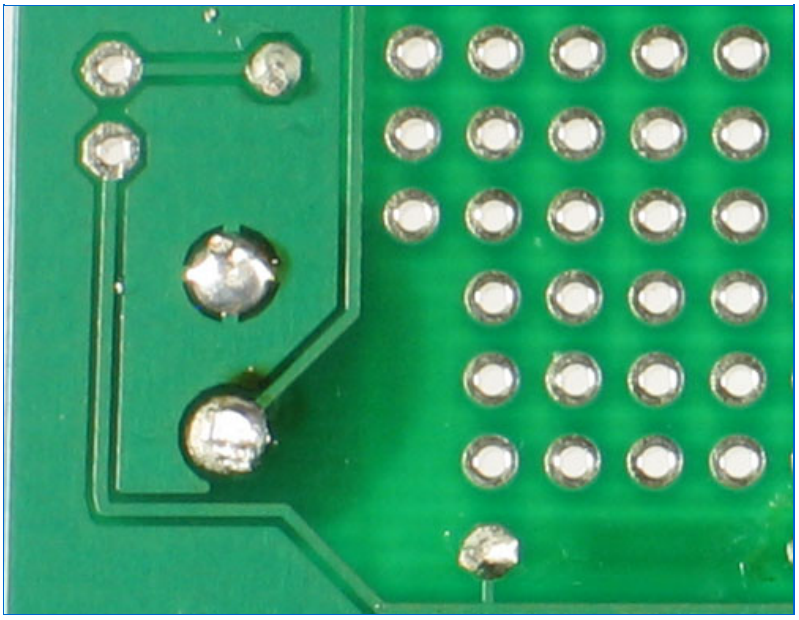
First solder in 2 opposite corners. Then solder the rest of the pins

Then do not need to be clipped as they are already quite short

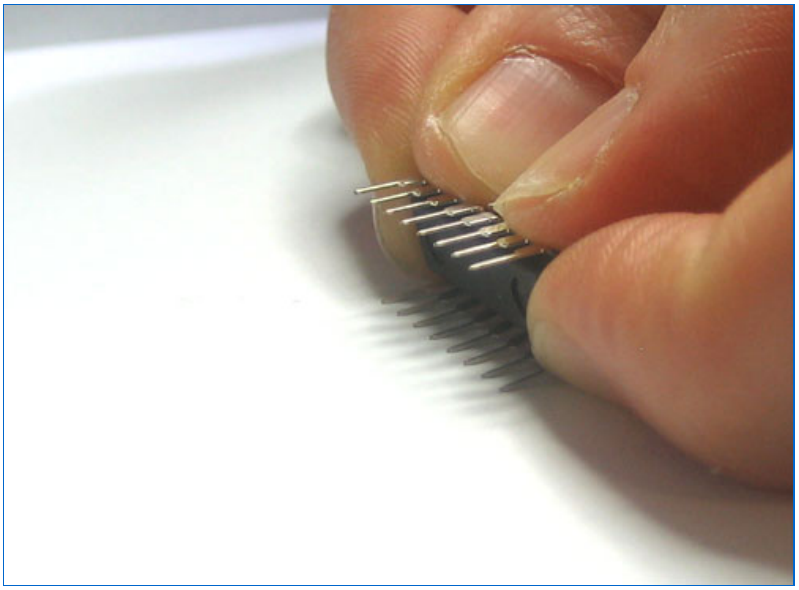
Place the piezo beeper **SPK**. This is the alarm noise-maker! It is non-polarized and can go in either way.

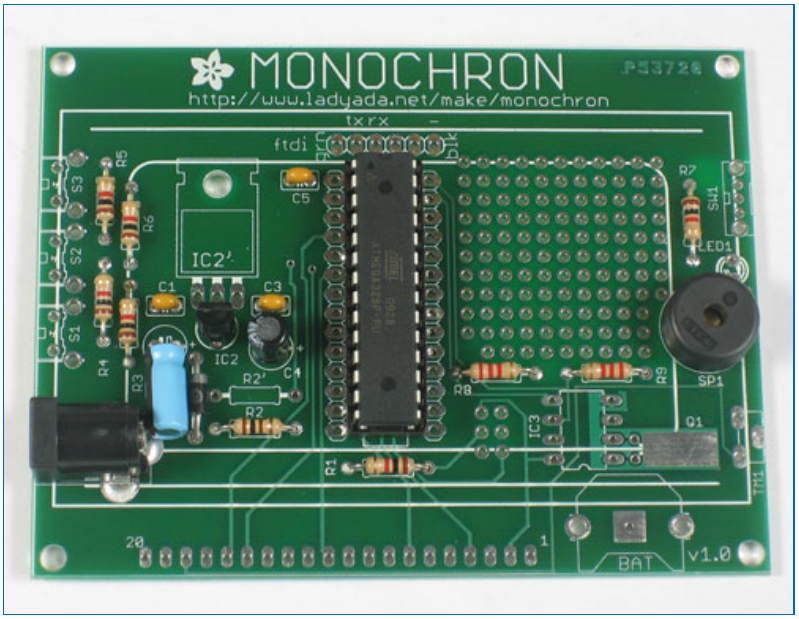






Now it is time to insert the processor chip! Carefully remove it from the packaging. You'll have to bend the pins in a little to make them fit nicely into the socket. I grab both ends and rock the pins against a tabletop. (The image shows a smaller chip, but the idea is the same). Once the legs are parallel, locate the U-shaped notch in one end. Make sure that this end goes into the notched-end indicated





on the silkscreen (and, hopefully, the socket as well)

Double check the chip is in right!

Now making sure that all the legs are lined up, and not bent or twisted, press the chip into the socket. It should seat itself easily without a lot of force.

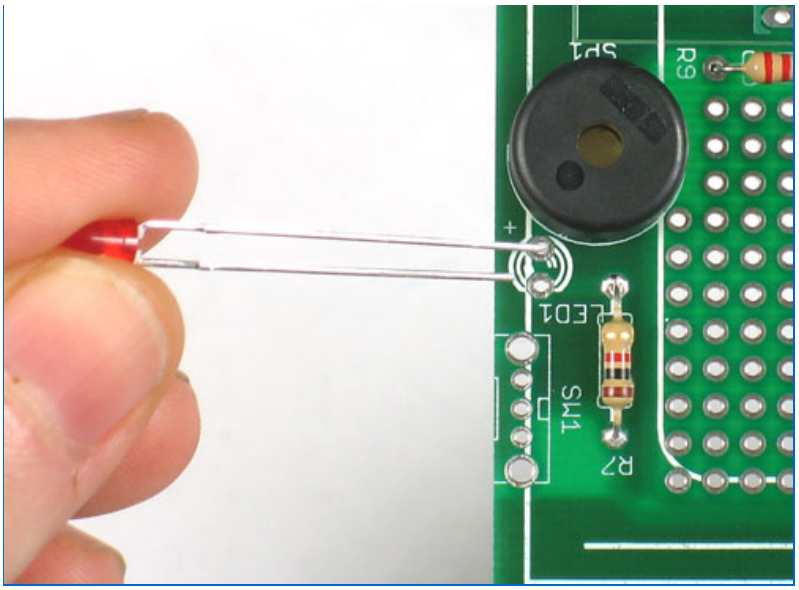
Beep! Beep!

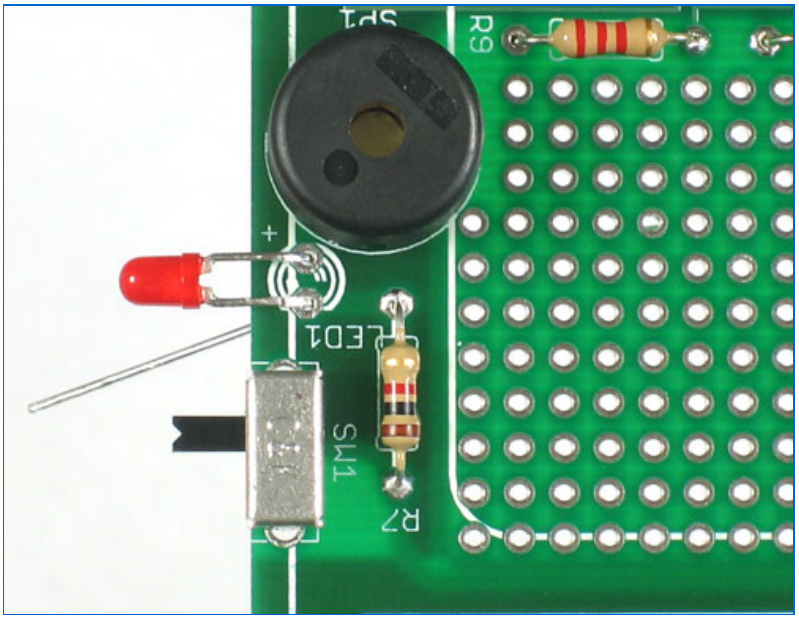
Now its time for another test. Clear off your table and plug in the power supply. You should hear repeating double beeps

If you aren't getting beeps, check the power supply, are you getting 5V still? Is the chip in right? If you are struggling, post in the forums for help.

Once you are satisfied, unplug the kit and continue

Now its time to place the small



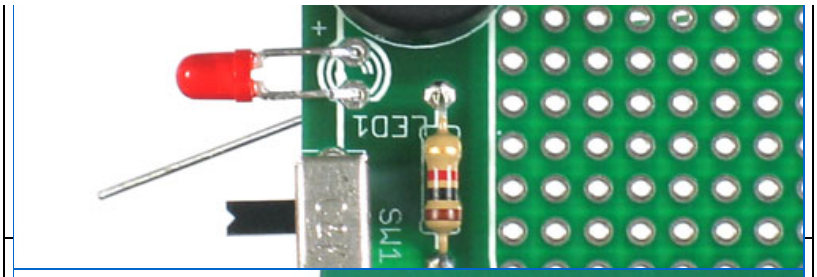


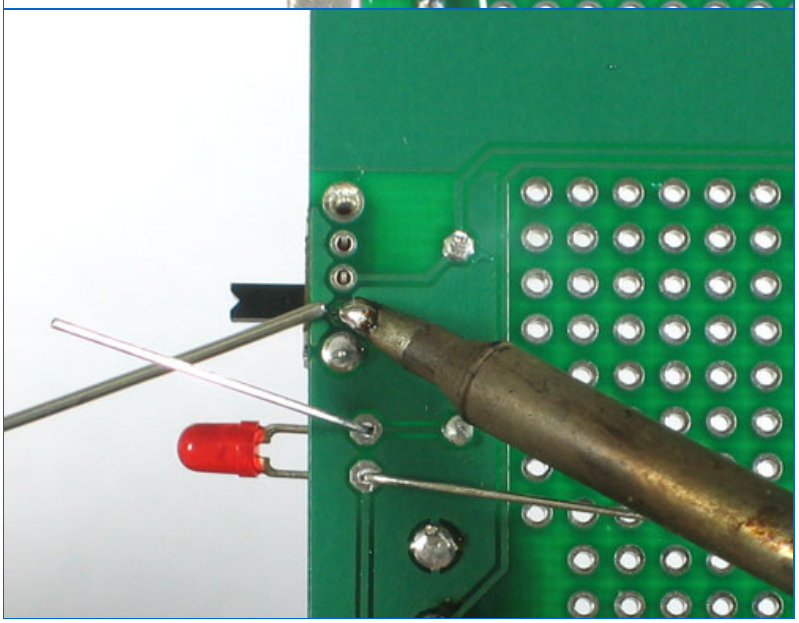
indicator LED **LED1**. LED's are not symmetric and must be placed correctly in order to work. You'll notice one leg of the LED is longer than the other. This is the *positive* leg. The positive leg goes into the hole with a **+** next to it. In the picture shown, its the left hole.

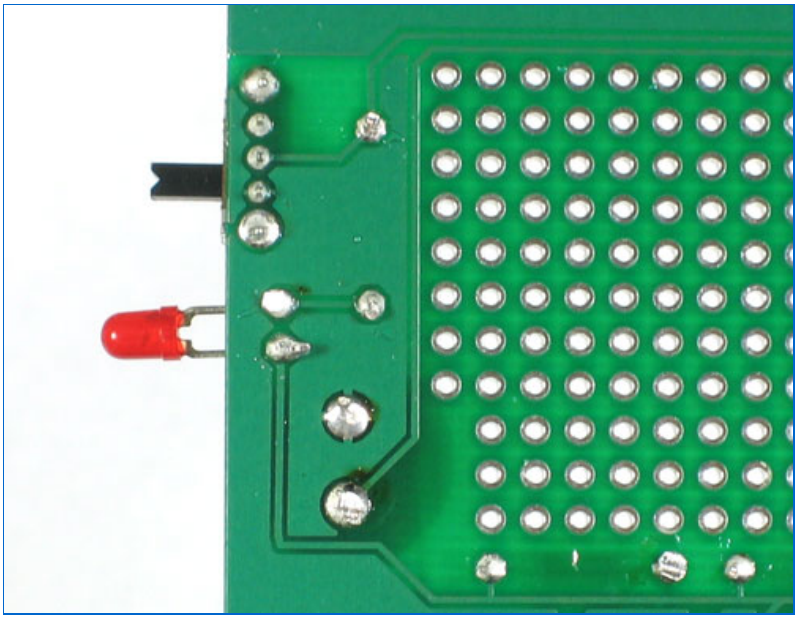
Insert the LED into the correct location, and bend the body out so it will stick out of the enclosure a bit. Beads out to keep it from falling out when you turn the PCB over.

Now you can also place **SW1** which is the alarm switch. Make sure the switchy part faces out, it should snap in place.

You may want to tack solder the switch from above to keep it in place. Make sure the switch is sitting flat against the PCB.





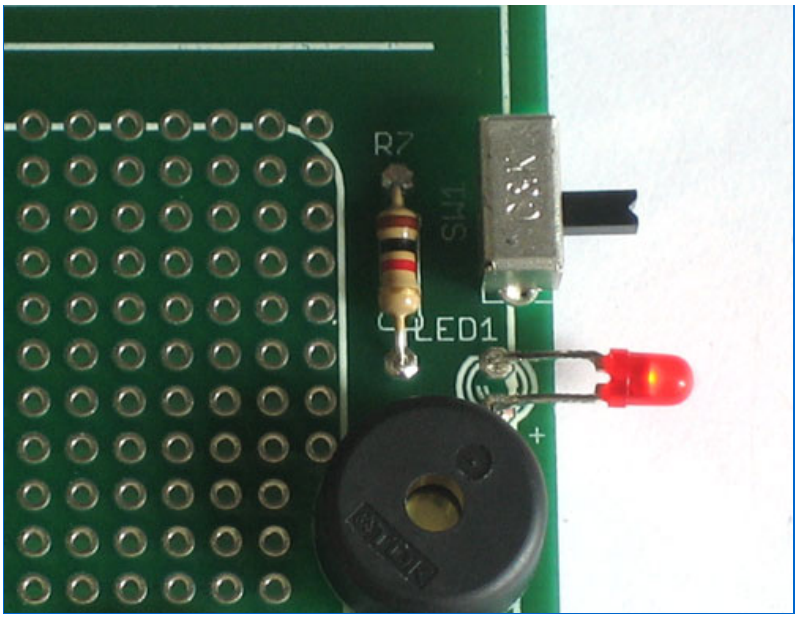


Solder in the LED and switch, watch out because the switch has some smaller pins in the middle. Dont use tons of solder.

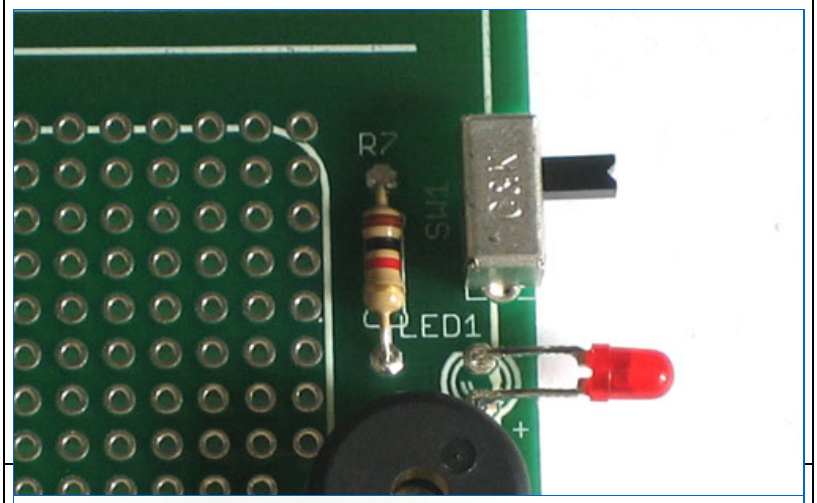
Clip the leads of the LED. The switch doesn't need to be clipped

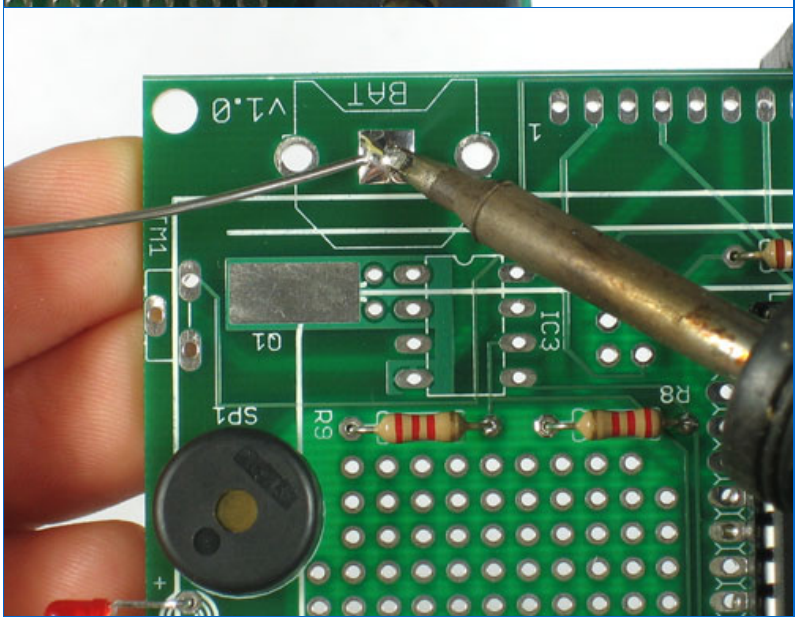
Now you can do a test of the LED and switch if you'd like. Clear off your desk and power up the clock. It will still beep, but now you can flip the switch back and forth and

see the LED turn on and off!



Once you are satisfied, lets continue



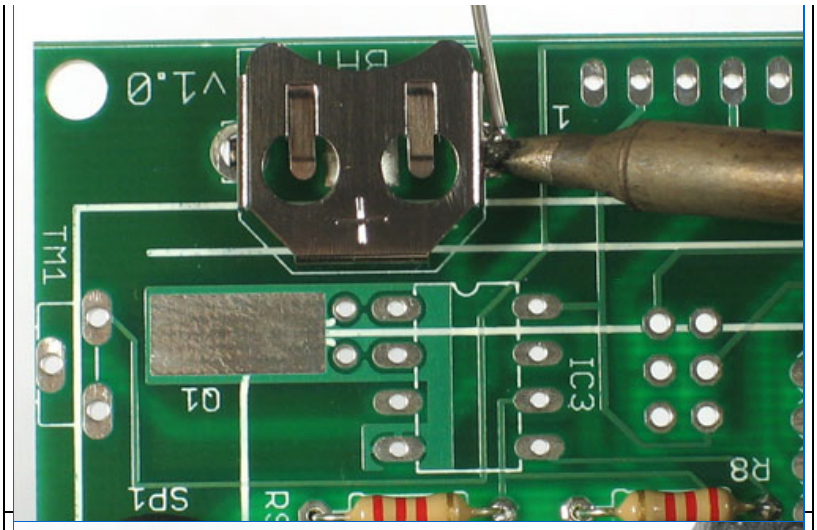


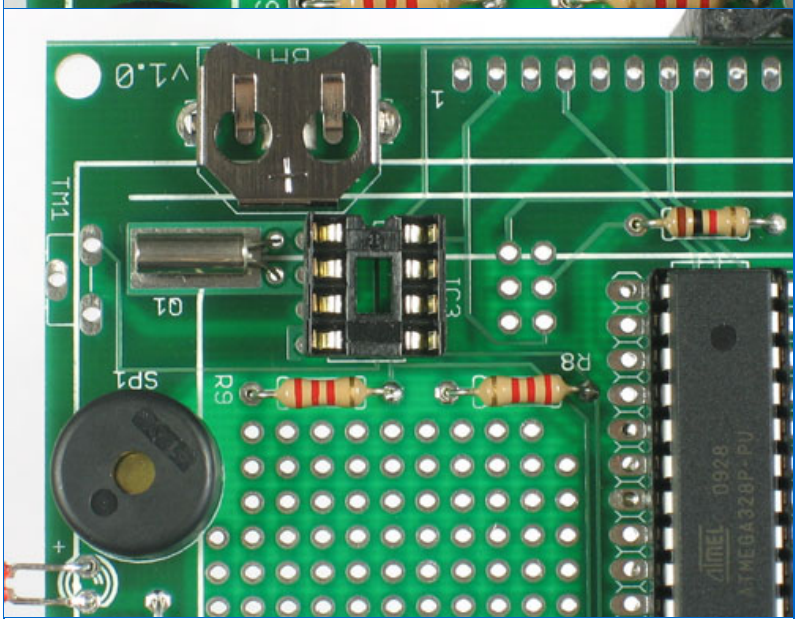
Now we will solder up the real time clock (RTC) circuit. The RTC is what keeps time when the power is out, its a very very very low power microcontroller and crystal that will keep time for years on a tiny coin cell. This way you can mod your clock with ease and not have to reset it after power loss.

First, melt a tiny bit of solder onto the center tab of the battery holder **BAT**. This will make good contact with the battery.

Now place the 12mm coin battery holder

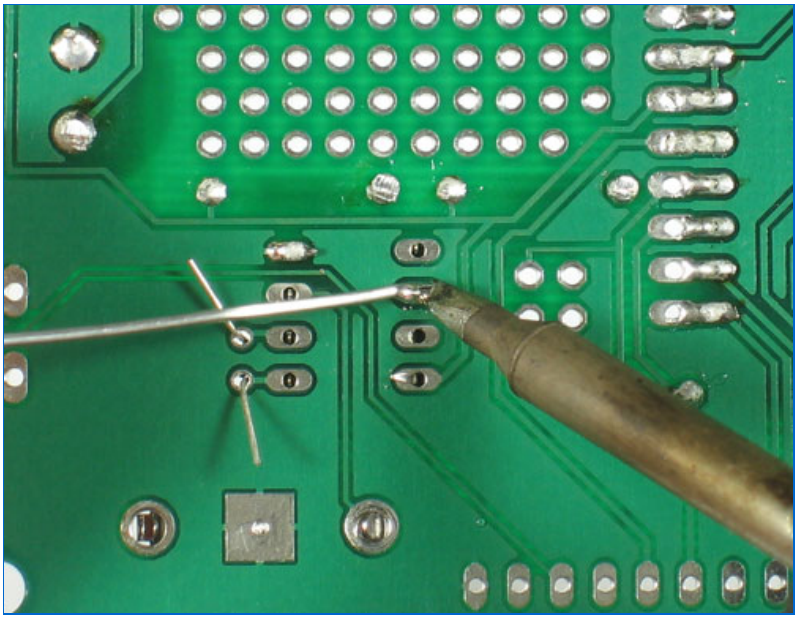
Tack solder one side so it doesnt fall out when you flip over.



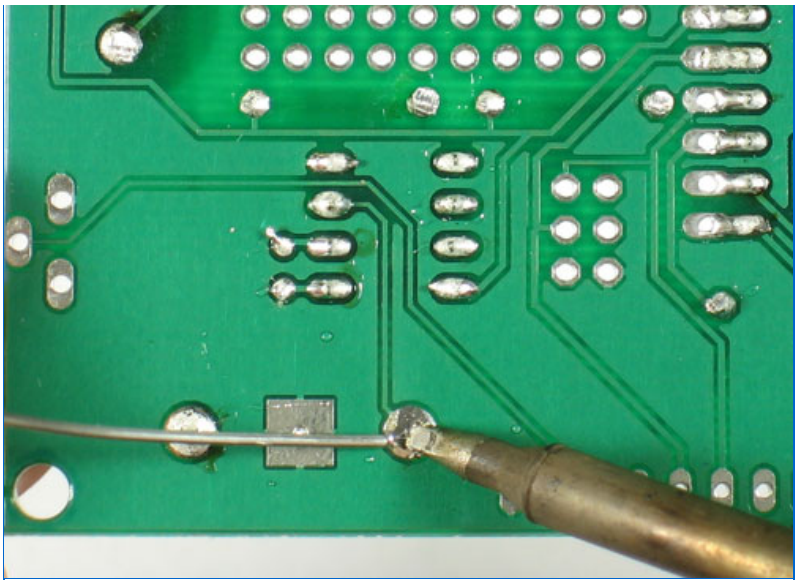


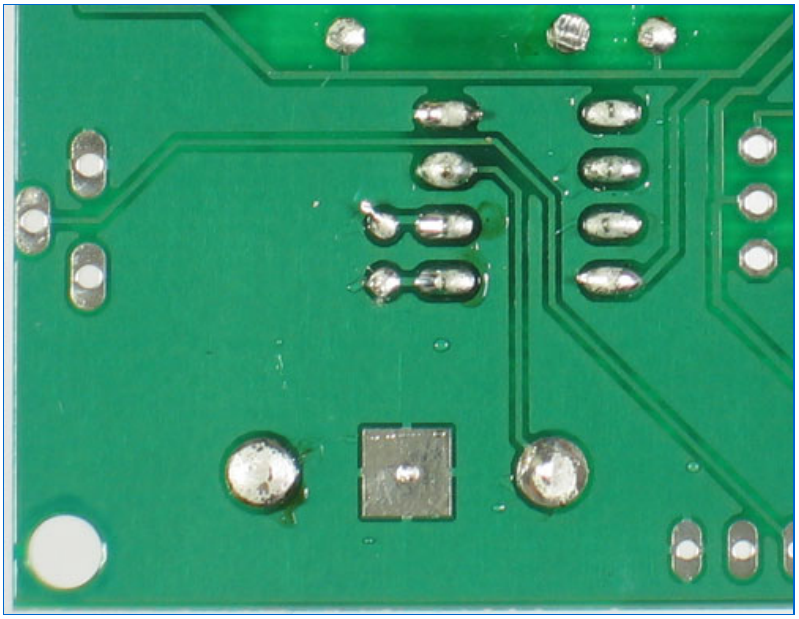
Now place the 8-pin socket (watch the U-tab) and 32.768 KHz watch crystal **Q1**. The socket is to protect the chip and the crystal is the same as whats in your watch or clocks.

The crystal sits on top of a pad but dont solder it to the pad! Just let it chill out there, sitting flat against the PCB



Solder the socket and crystal, then clip the crystal leads



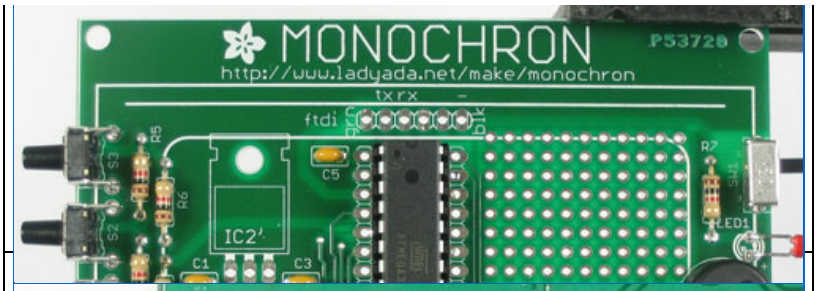


Now that the RTC is done, we will solder in the 3 interface switches **S1**, **S2**, and **S3**. These are what allow you to set the time and alarm and date and all that great stuff. they will snap into place, try to get them as flat to the PCB as possible.

All the way to the right we will also solder in the 10K 'trimmer' potentiometer **TM1** which is used

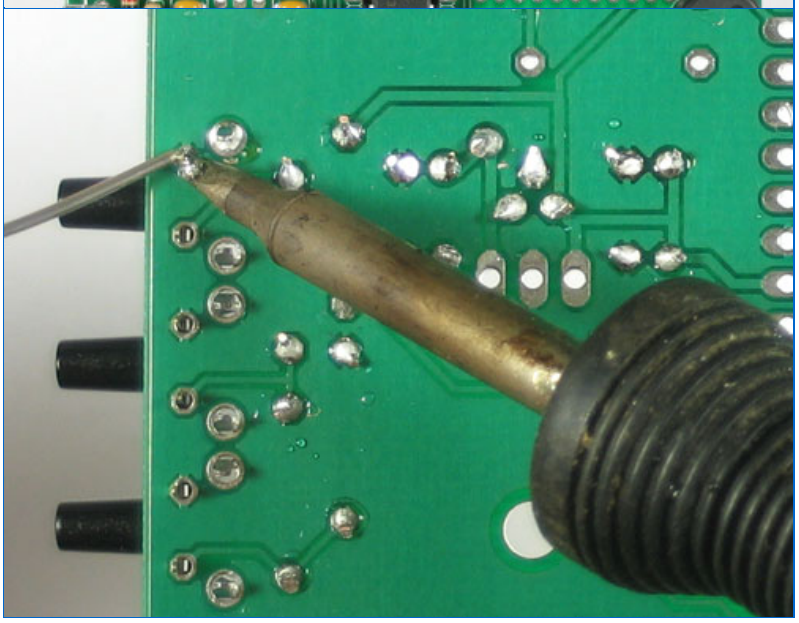
to set the contrast of the LCD. It

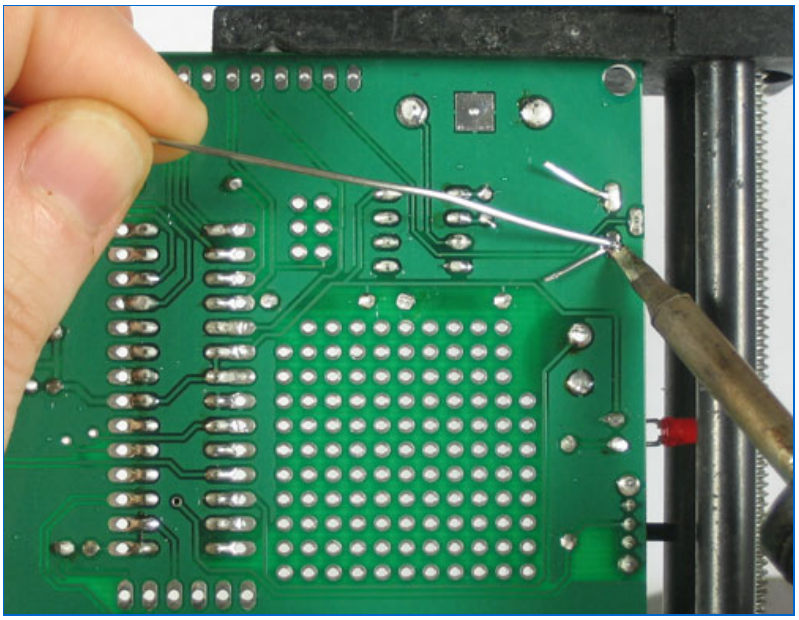
also sits flat against the PCB



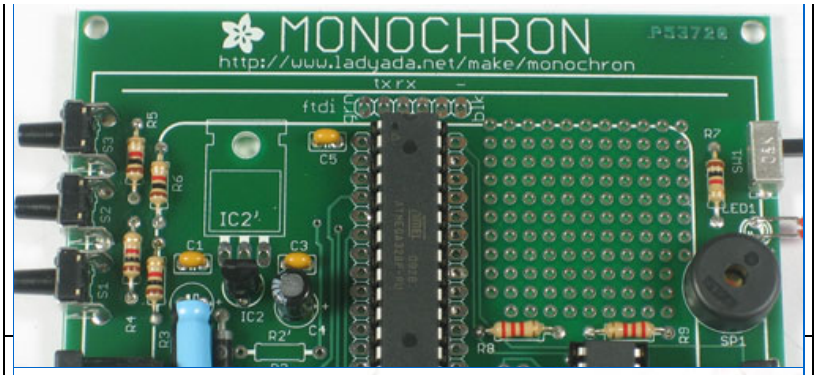
Solder in the three switches, checking they are sitting flat.

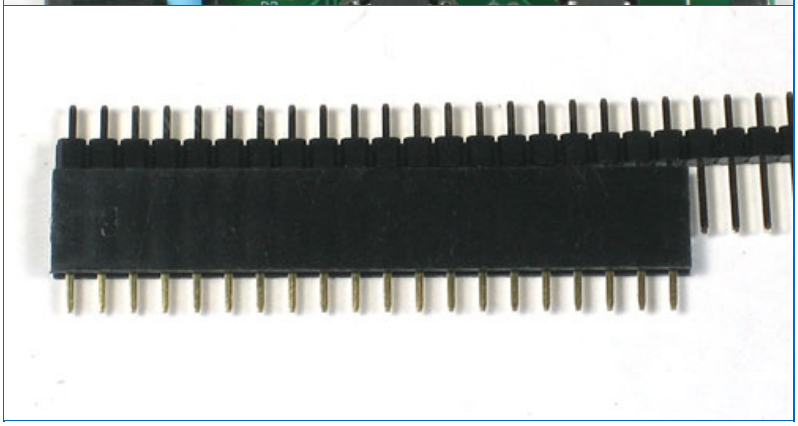
Then also solder and clip the trimmer pot.

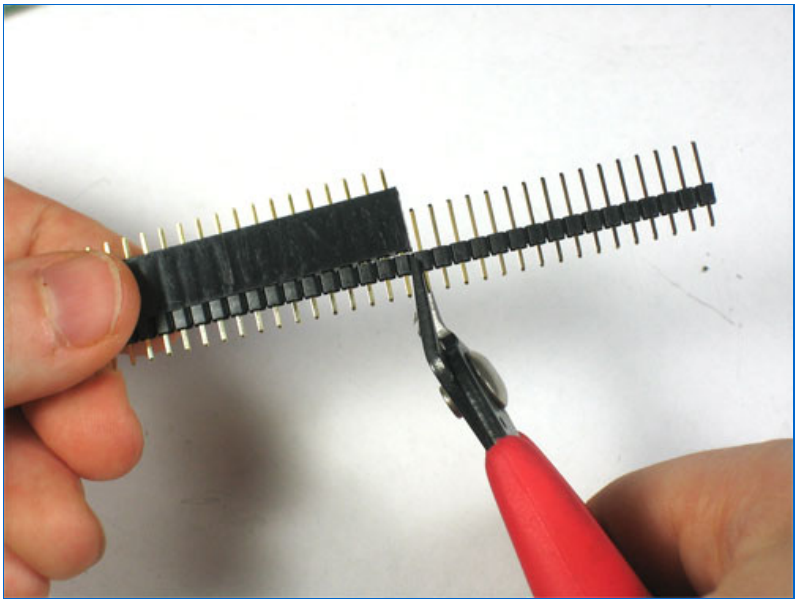


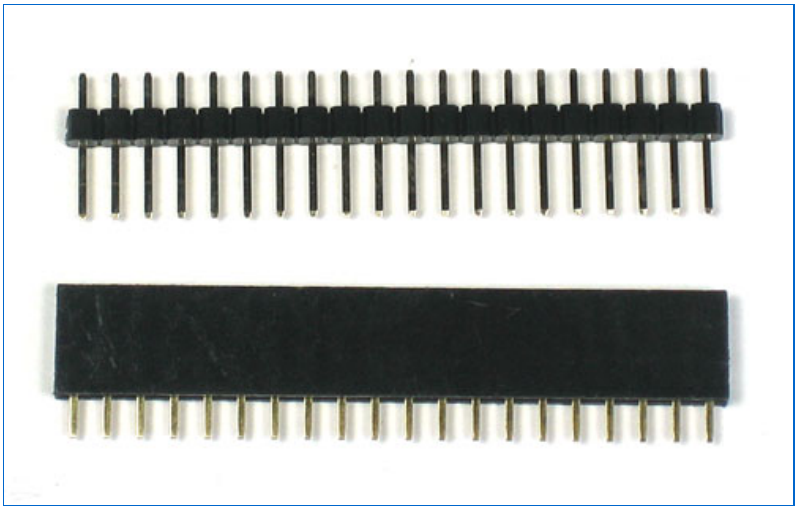


Insert the DS1307 RTC chip into the socket, making sure to align the U notches



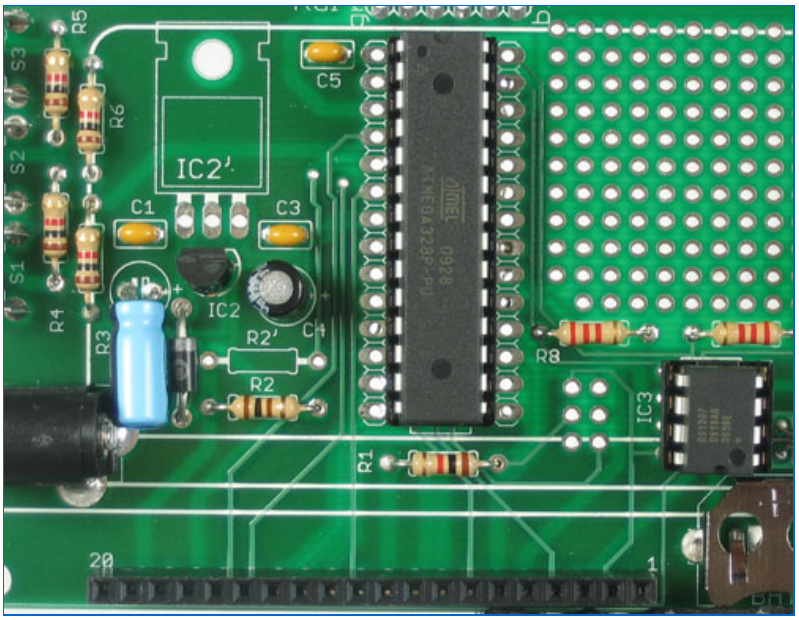


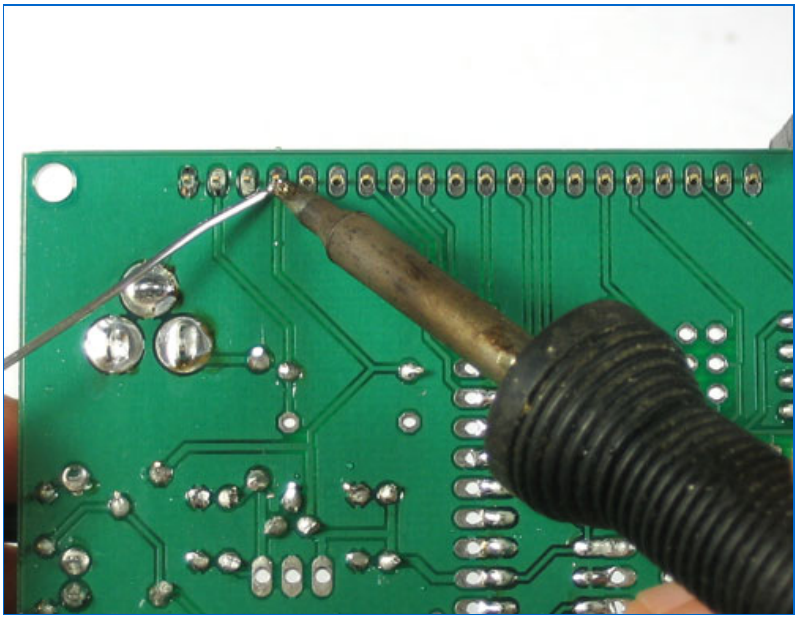




Now we're onto the LCD part. Find the 36 pin male header and 20 pin female header. Use the 20 pin header to help you cut down the 36-pin down to 20.

You can use diagonal cutters, or pliers to break apart the header





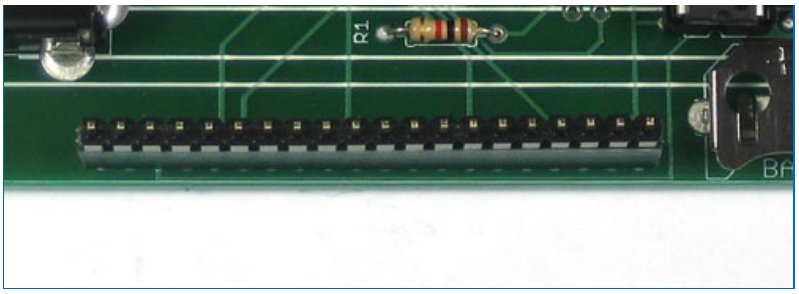
Place the female header strip with the sockets up in the PCB

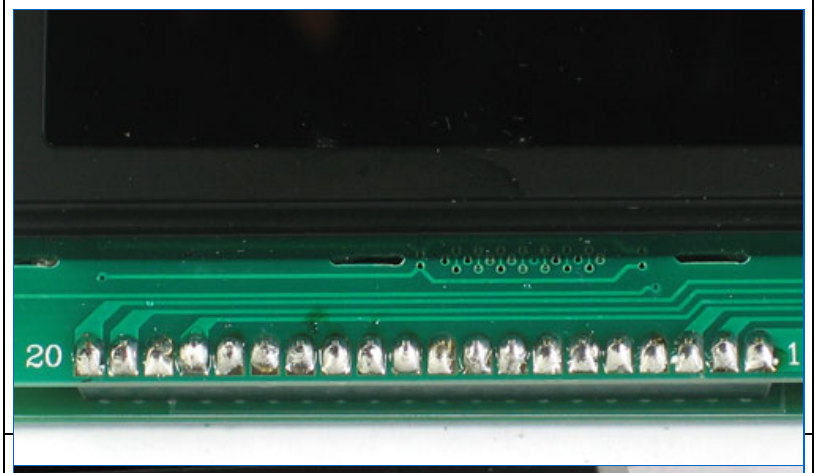
You may need to tape it so you can turn over the kit and solder it in place. Solder a few pins and then see if you need to bend the header so its perpendicular!

Once the female header is solidly in place, slip the male header into the sockets so that the short pins stick up.

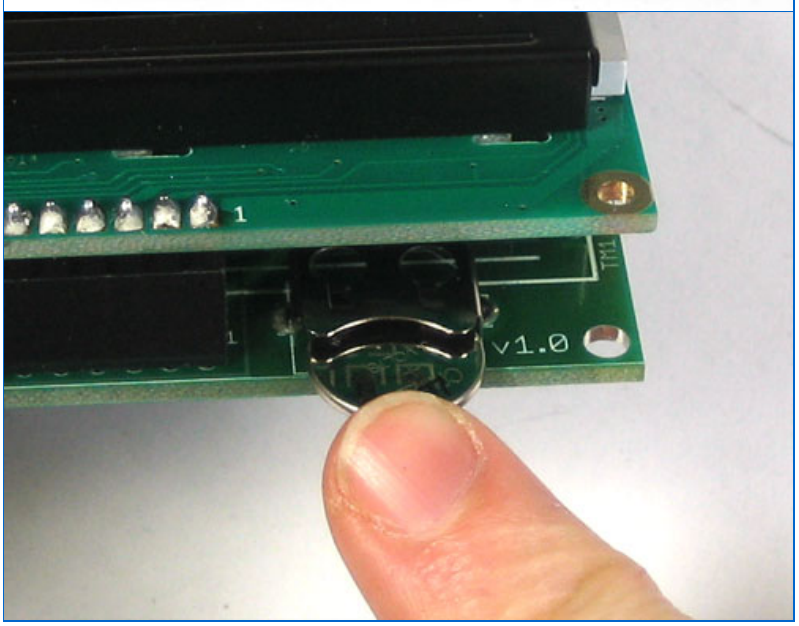
Then place the graphical LCD on top and solder the pins. You may want to stuff something underneath the LCD so it is parallel to the PCB

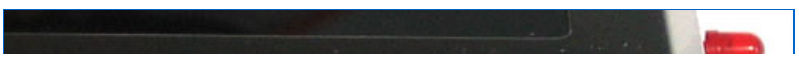
the LCD so it is parallel to the PCB

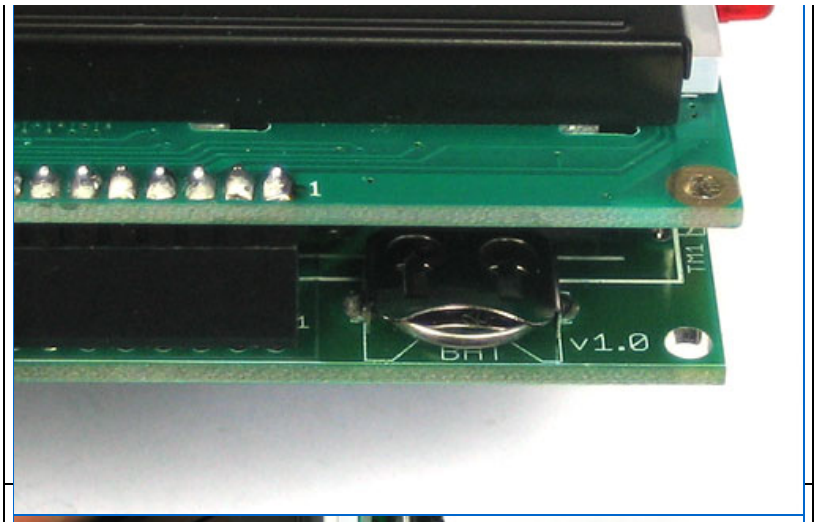


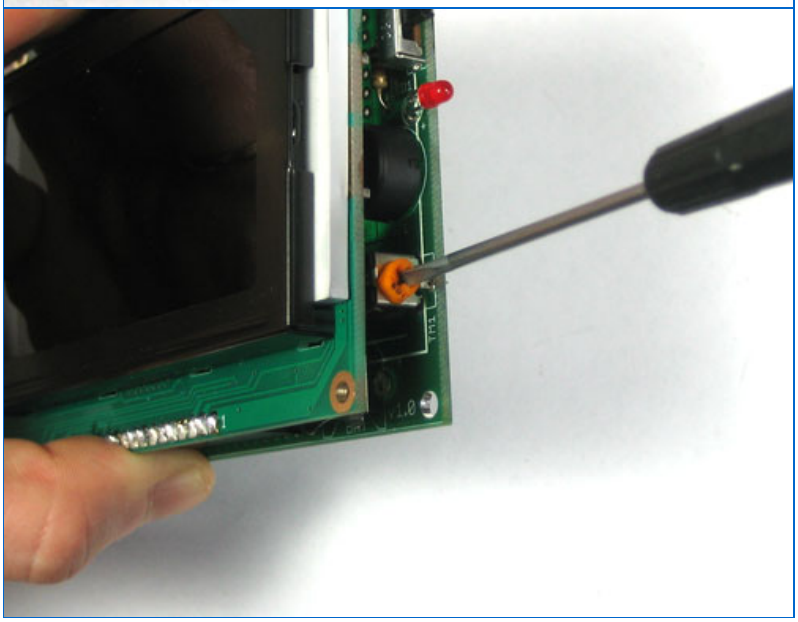


Now insert the coin cell, + side goes up. You must place the coin cell because otherwise the RTC will be erratic and the clock will act oddly.



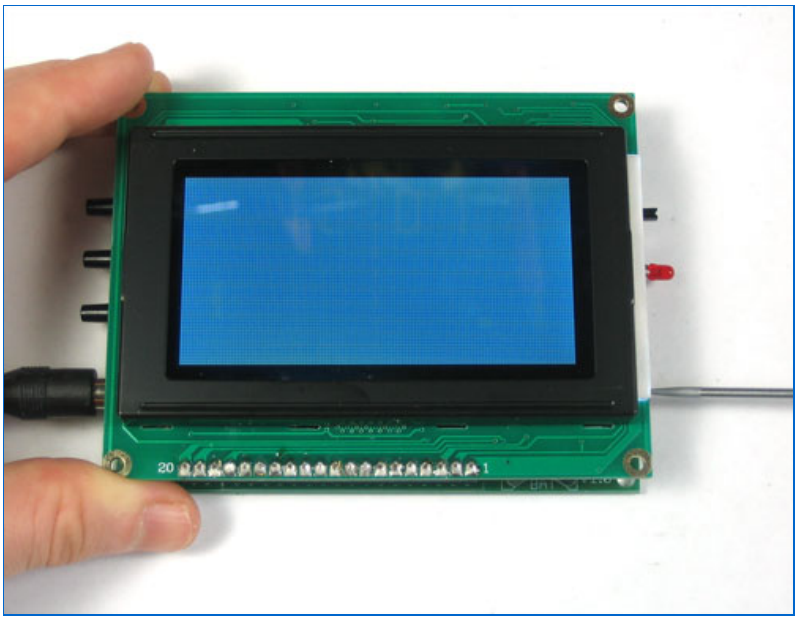


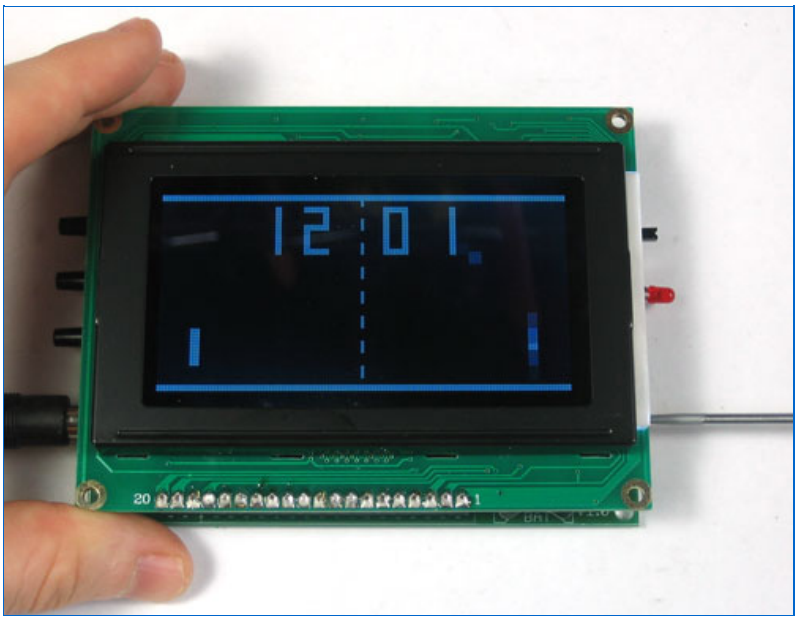




The trim pot on the side is used to adjust the contrast.

Plug in the kit and turn the pot with a small screwdriver until you see the animated display show up!





Great! Now that you have your kit up and running its time to build the enclosure, please go to the next step of the instructions