Chipsmall Limited consists of a professional team with an average of over 10 year of expertise in the distribution of electronic components. Based in Hongkong, we have already established firm and mutual-benefit business relationships with customers from,Europe,America and south Asia,supplying obsolete and hard-to-find components to meet their specific needs.

With the principle of “Quality Parts,Customers Priority,Honest Operation,and Considerate Service”,our business mainly focus on the distribution of electronic components. Line cards we deal with include Microchip,ALPS,ROHM,Xilinx,Pulse,ON,Everlight and Freescale. Main products comprise IC,Modules,Potentiometer,IC Socket,Relay,Connector.Our parts cover such applications as commercial,industrial, and automotives areas.

We are looking forward to setting up business relationship with you and hope to provide you with the best service and solution. Let us make a better world for our industry!

Contact us

Tel: +86-755-8981 8866 Fax: +86-755-8427 6832
Email & Skype: info@chipsmall.com Web: www.chipsmall.com
Address: A1208, Overseas Decoration Building, #122 Zhenhua RD., Futian, Shenzhen, China

Bulbdial

LED Shadow Clock Kit

Assembly Guide

The Bulbdial Clock kit is based on an original design concept by David Friedman of IronicSans.com and developed at Evil Mad Scientist Laboratories.

The Bulbdial Clock works like an indoor sundial. Three rings of LEDs cast shadows of different lengths, forming three moving hands on the clock face.

This guide covers the procedures for assembling the Bulbdial Clock and its accessories. Please exercise appropriate safety practice while soldering.

An open-source hardware+software project. For design files, source code, & additional documentation, please visit: http://wiki.evilmadscience.com/Bulbdial

Support: http://www.evilmadscientist.com/forum/

Distributed by
Evil Mad Science LLC
http://evilmadscience.com/

Evil Mad Science

Kit versions 1.0-1.1
Manual v. 1.1a

STEP 0: Tool Checklist

Essential tools: Needed to build the kit:

1. Soldering iron + solder

A basic soldering iron meant for electronics, with a reasonably fine point tip. We recommend one of this design-- a "pencil shape" soldering iron (not gun!) with a base that holds the iron and a wet sponge. A tip in good condition (a "tinned" tip) should get shiny when hot-- able to melt and wet to solder.

While you don't need an *expensive* one, the iron *can* make a big difference in the time needed to build the kit. (Seriously. If you use one that is old and busted, or a $10 radio shack iron, or that thing from the dollar store, you should expect to spend at least twice as long soldering!)

Our recommendation for a low-cost iron:
model WLC100 by Weller, about $40.

You'll also need some solder. Thin *rosin-core* solder (roughly .020 - .040" in diameter) is the most common type for electronic soldering, and is the only choice that is appropriate for electronic kits. Either standard (lead-bearing) or newer "lead free" solder types will both work just fine.

2. Angle flush cutters

Small nippers for for clipping loose wire ends close to the circuit board.

e.g., Sears Craftsman

3. Electrical power

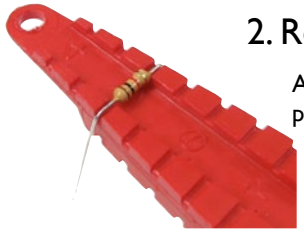
The Bulbdial Clock kit comes with a regulated universal-input power supply that accepts worldwide voltages and puts out 5 V at up to 150 mA. If you're using your own external power, make sure that it's regulated, provides 4.5 to 5 V DC, and is rated for at least 100 mA. (3 "AAA" cells can do the job, for example.) Please be careful: Inappropriate voltage or polarity can cause permanent damage.

(International users may need to supply a plug adapter to fit the prongs into local outlets.)

Suggested, but not required

1. Small pliers with smooth jaws

Very helpful for final LED alignment. The jaws should be smooth so that you don't scratch the LEDs. Needlenose pliers with masking tape over the jaws can be a good solution.



2. Resistor lead forming tool

Allows fast, neat bending of resistor leads. Not many parts like this in the Bulbdial kit, but if you're obsessive....

And for the adventurous...

1. USB-TTL Cable

FTDI model TTL-232R or equivalent. A "smart" converter cable with a USB interface chip inside. One end hooks up to your USB port, the other to the clock. This allows you to program the Bulbdial Clock through the Arduino development environment (http://arduino.cc/).

Besides programming, the cable can also be used to set the time on the Bulbdial clock through a serial sync program.

The 5V version of the cable (TTL-232R-5V) can be used to provide USB power to the Bulbdial clock.

2. Computer, Internet access, USB port....

All of the software and source code that you'll need to reprogram the clock is available online for free. You'll need a reasonably recent vintage computer (Mac, Windows, or Linux) and internet access.

Additional information is available at the project page:
http://www.evilmadscientist.com/go/BulbDialKit

STEP 1: It's the BOM.

Your kit came with a *bill of materials*: an up-to-date list of what's in your particular kit. The exact items may differ between versions of the kit, for example between RGB and monochrome options.

Bulbdial

Bulbdial Clock Kit:: Bill Of Materials
Standard 3-color version

Evil Mad Science

Congratulations on your new Bulbdial Clock kit! This document lists everything that comes with the kit. If anything is broken or missing, please let us know right away and we'll get you squared away.

To put your kit together, you'll need the build instructions.
Please visit: http://www.evilmadscientist.com/go/bulbdialkit

Main Kit

Line	PCB location	Description	Needed	Kit Qty
1	-	Blue PCB	1	1
2	-	Green PCB	1	1
3	-	Red PCB	1	1
4	JP1, LED1-10	Zerohm Jumper Wires ("Resistors with one black stripe")	12	13
5	R11	Resistor, 10 k Ohm, 1/6 W (the small one)	1	1
6	R1-R9	Resistor, 24 Ohm, 1/4 W	9	11
7	R10	Resistor, 68 Ohm, 1/4 W	1	1
8	Quartz	Oscillator Crystal, 16 MHz, 20 ppm	1	1
9	C1,C2	Capacitor, 18 pF ceramic. (Marked with black stripe)	2	2
10	S1,S2,S3	Tactile button switch, Right angle, long actuator	3	3
11	J2	Header, male, 6-position SIL, right-angle	1	1
12	M1	Header, female, 4-position SIL, vertical	2	2
13	C3,C6	Capacitor, 0.1 µF ceramic	2	2
14	U1	ATmega168-20PU microcontroller, pre-programmed	1	1
15	J1	Power jack, 2.5 mm	1	1
16A	(See instrux)	Blue LED, 5mm, Ultrabright, 20°	30	31
16B	(See instrux)	Pure Green LED, 5mm, Ultrabright, 20°	30	31
16C	(See instrux)	Red LED, 5mm, Ultrabright, 20°	12	13
17	-	Custom LED Bending Guide, 6 mm plywood	1	1
18	-	Hex Standoff M/F, Aluminum, 3/8" long, 4-40 thread	8	8
19	-	Hex Standoff F/F, Aluminum, 7/8" long, 4-40 thread	4	4
20	-	Printed Clock face, 1/16" acrylic	1	1
21	-	Gnomon Spike, Nickel-plated, 3/8" x 4-40 thread	1	1
22	-	Button socket cap screw, Stainless, 3/16" x 4-40	1	1
		Hex wrench, alloy steel, 1/16"	8	9
		Button socket cap screw, Stainless, 3/8" x 4-40	1	1
		150 mA, Universal input, Center positive	4	5

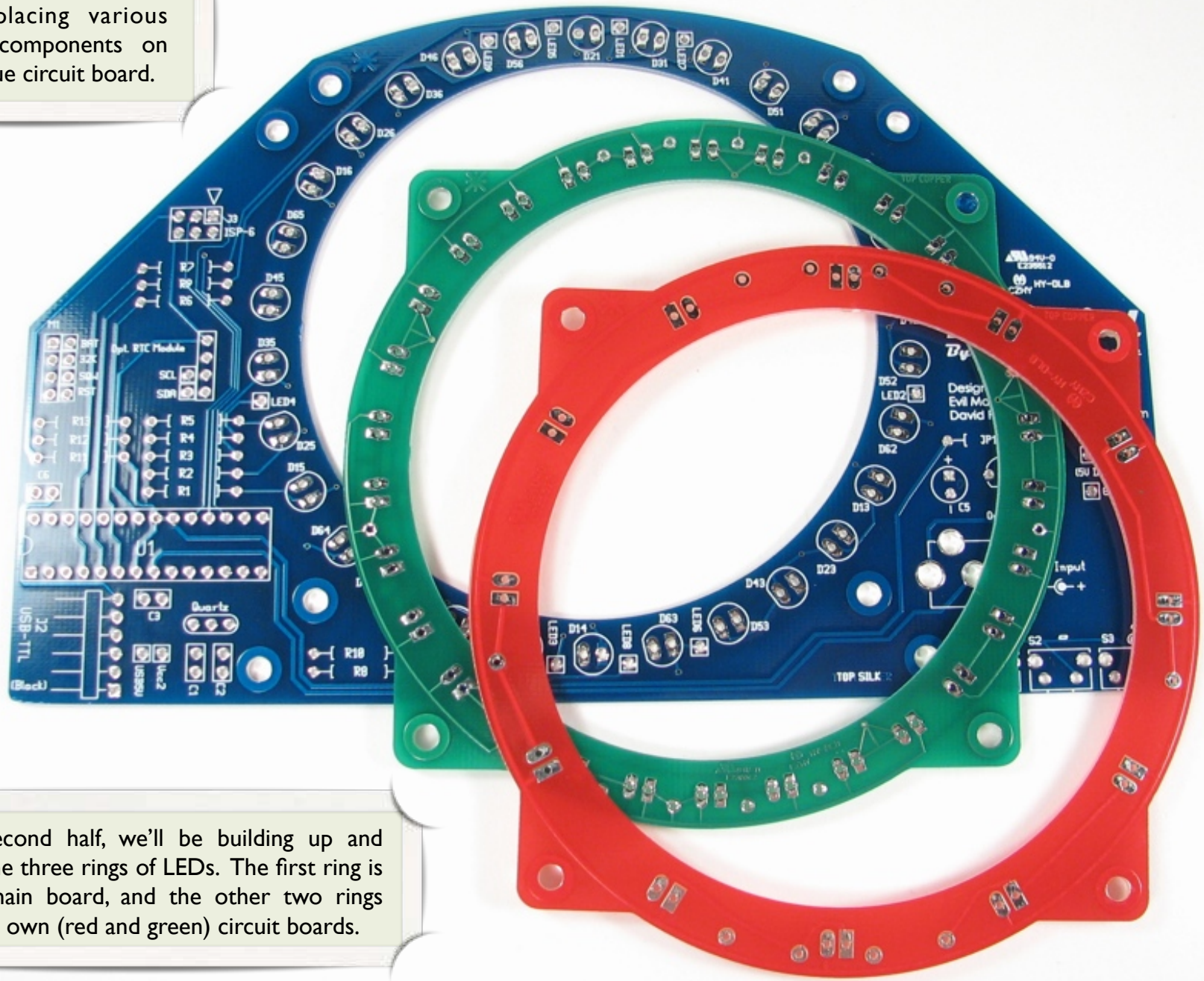
In the instructions, we refer to components by their line item number on the bill of materials. For example, **#14** is the ATmega168 microcontroller.

In some places, the assembly procedure differs between kit versions.

In particular, watch out for *RGB ONLY* or *MONO ONLY* instructions, which are specific to the 3-color or monochrome versions of the kit. If you aren't sure which you have, please consult your *BOM*.

STEP 2: The Plan

In the first part of the build, we'll be placing various electronic components on the large blue circuit board.

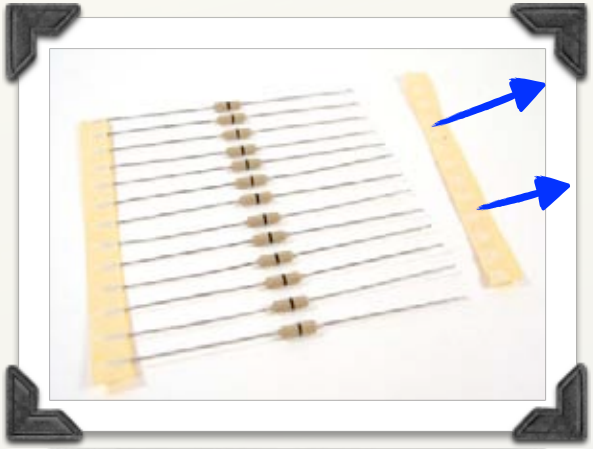


In the second half, we'll be building up and aligning the three rings of LEDs. The first ring is on the main board, and the other two rings have their own (red and green) circuit boards.

STEP 3: Placing the first component

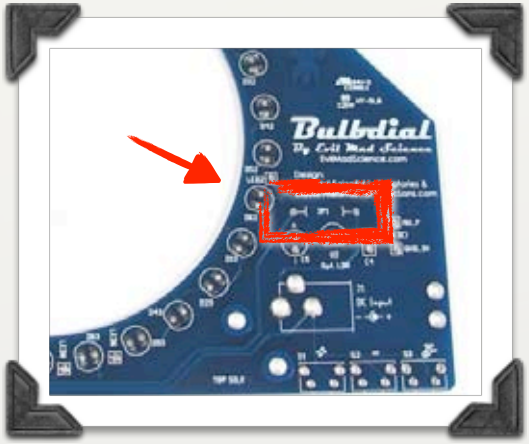
Our first component is a "zerohm" jumper, **#4.** It goes in location JP1 on the main circuit board. (And for this first one, we'll take it slowly.)

A zerohm jumper looks like a resistor with one black stripe.



If they are taped together, pull the tape straight off.

Take one and bend it, like so.



Now, find location JP1.

Insert the jumper there...

... and press it flush to the board.

(Next... Soldering!)

STEP 4a: Some hints on soldering

As the old Heathkit manuals say, "it is interesting to note" that the vast majority of problems reported with soldering kits turn out to be due to unreliable solder connections.

Before we go further, here's a quick refresher, with our suggested procedures for adding components to the circuit board. These procedures apply to most components in the kit.

Adding components to the circuit board

(0). Pre-form the leads of components if needed.
(For example, like the jumper in step 3.)

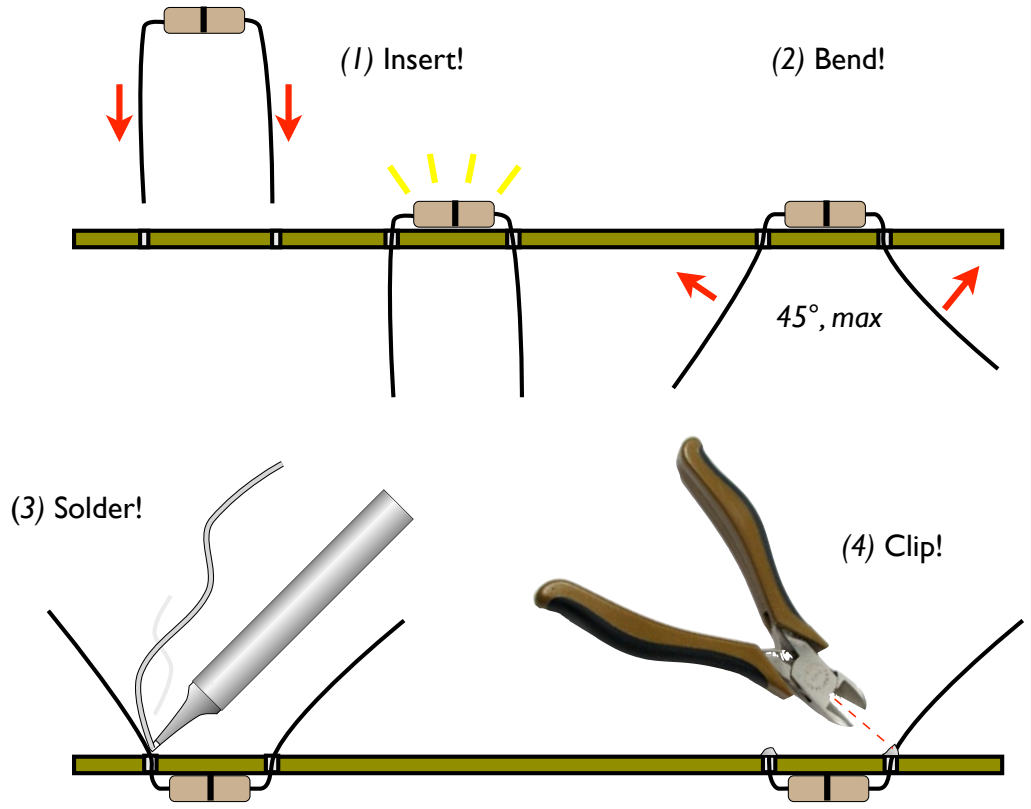
(1). Insert each component into the circuit board, from the top, at its given location. Push it flush to the board (Note that some components, like the chip and LEDs, need to be inserted with a particular orientation.)

(2). If your component has flexible leads, *gently* bend the leads out, up to 45°, to hold it in place while you solder.

(3). One at a time, from the back side, solder the leads of the component to the circuit board.

- Your tip *needs* to be *shiny* (tinned). If not, melt some fresh solder against it and *quickly* swipe clean on a wet sponge.
- Place the solder against the joint that you wish to connect.
- Touch the iron to the solder and joint for about one second. Count it out: "one thousand one."
- The solder should melt to the joint and leave a shiny wet-looking joint. If not, let it cool and try again.

(4). If the component has long and/or or flexible leads, clip off the extra length, close to the board. (But not so close that you're clipping the board itself.)

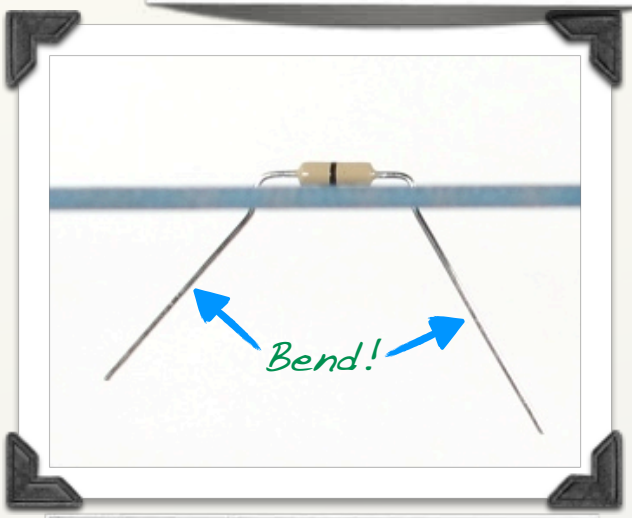


TO BE CONTINUED...

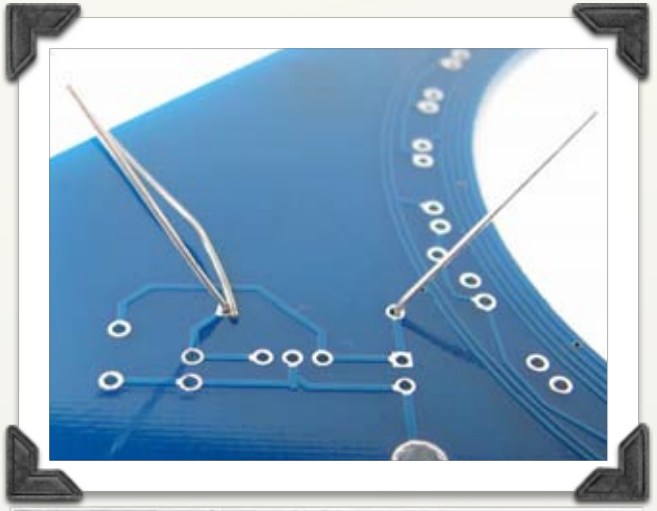
STEP 4b: Soldering that first component

(Where we perform those steps that we just described.)

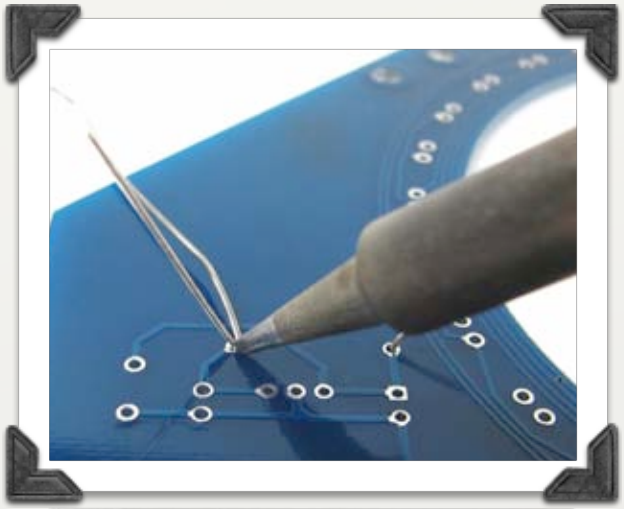
We've already inserted the jumper at location JP1.



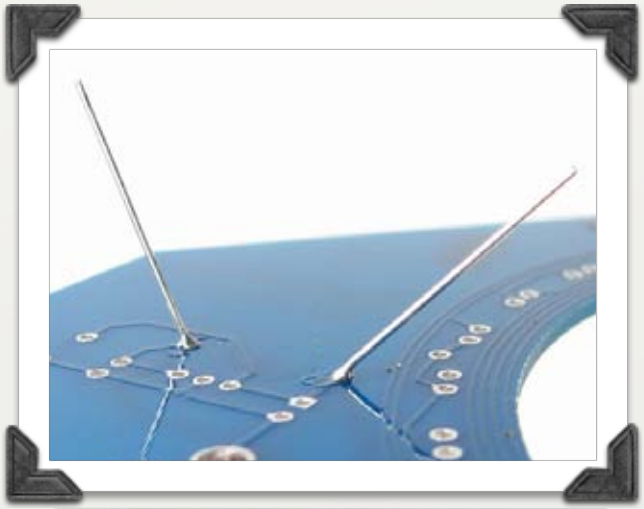
Now bend out the leads to hold the jumper in place while we solder



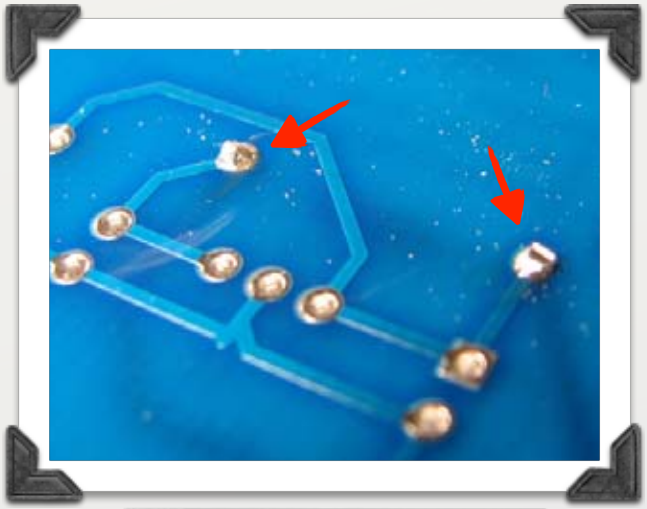
Flip the board over and get ready to solder.



Solder both pins to the board.



Inspect the solder joints; make sure that they look shiny and wet.



Clip the excess leads short.

STEP 5: One or two resistors...

Now that we've stepped through the basics, we can proceed at a faster pace.

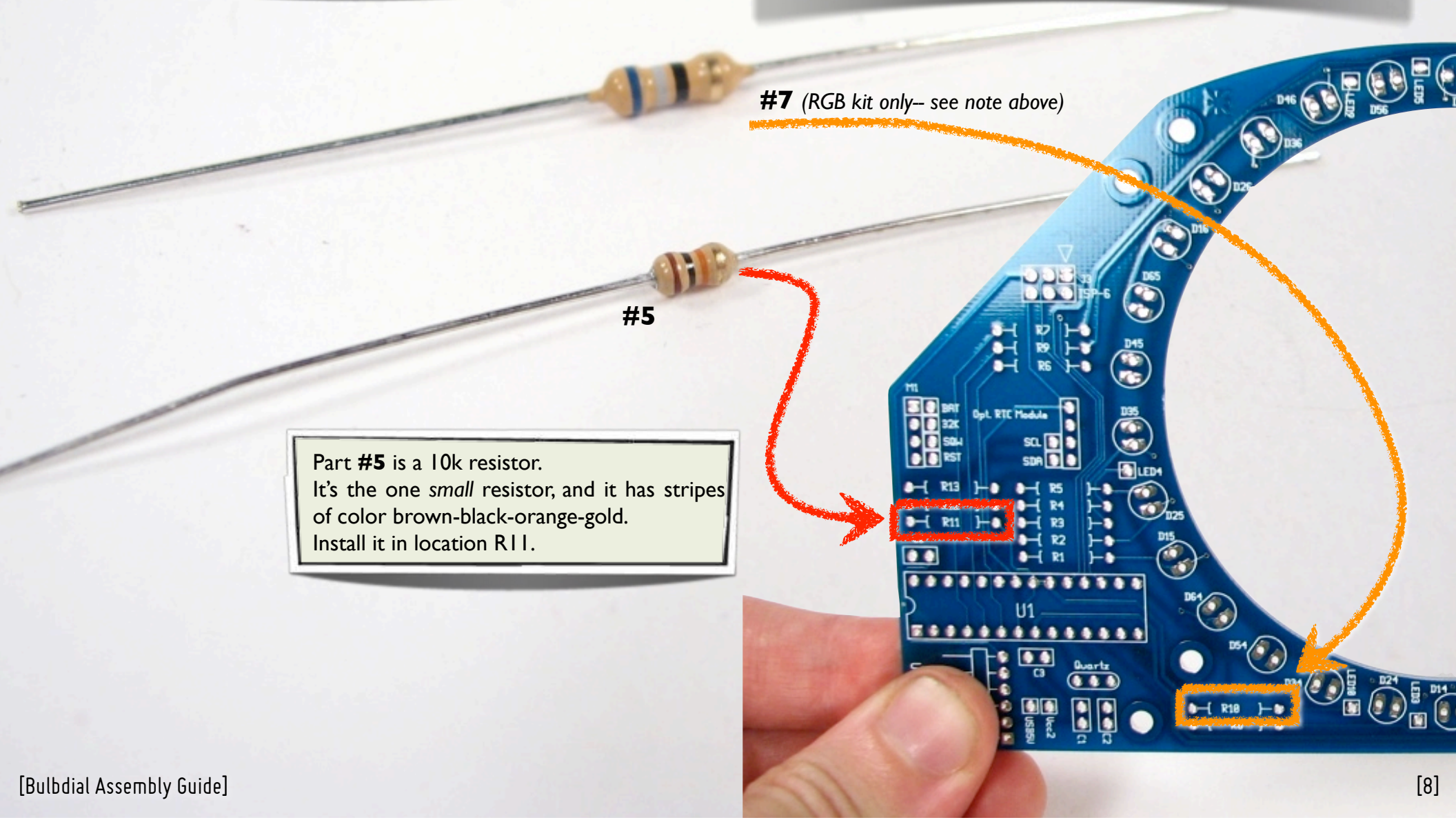
Resistors are added to the board in the same way as the zerohm jumper, and are not sensitive to orientation.

If you have the RGB kit:

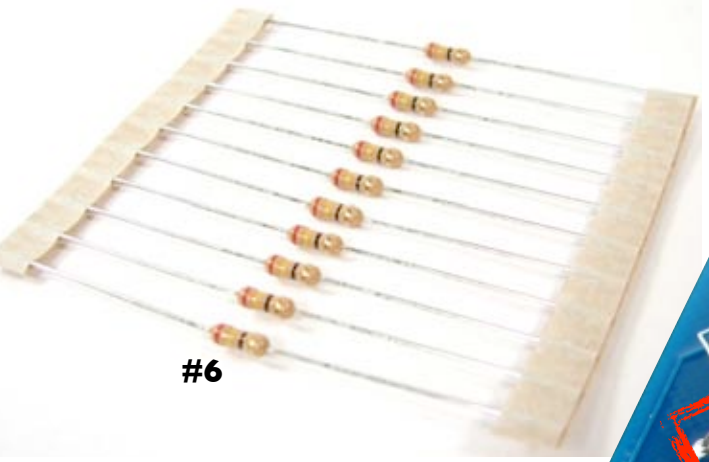
Part **#7** is a 68 ohm resistor (color code blue-gray-black-gold). It's the one "normal size" resistor that looks different from all the others. Install it in location R10.

If you have the monochrome (all white) kit:

Go on to Step 6, leaving R10 empty for the moment.



STEP 6: The rest of the resistors



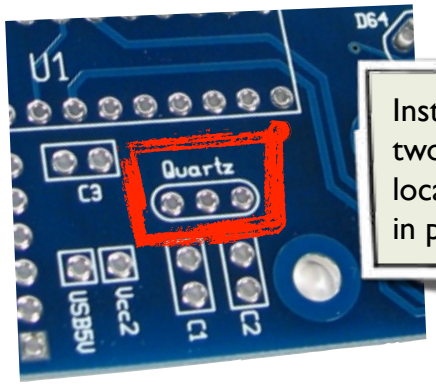
The last resistor type is part **#6**, a 24 ohm resistor (color code red-yellow-black-gold). Install nine of these in locations R1 - R9.

Those resistors are located in the three highlighted locations. *Don't forget R8!*

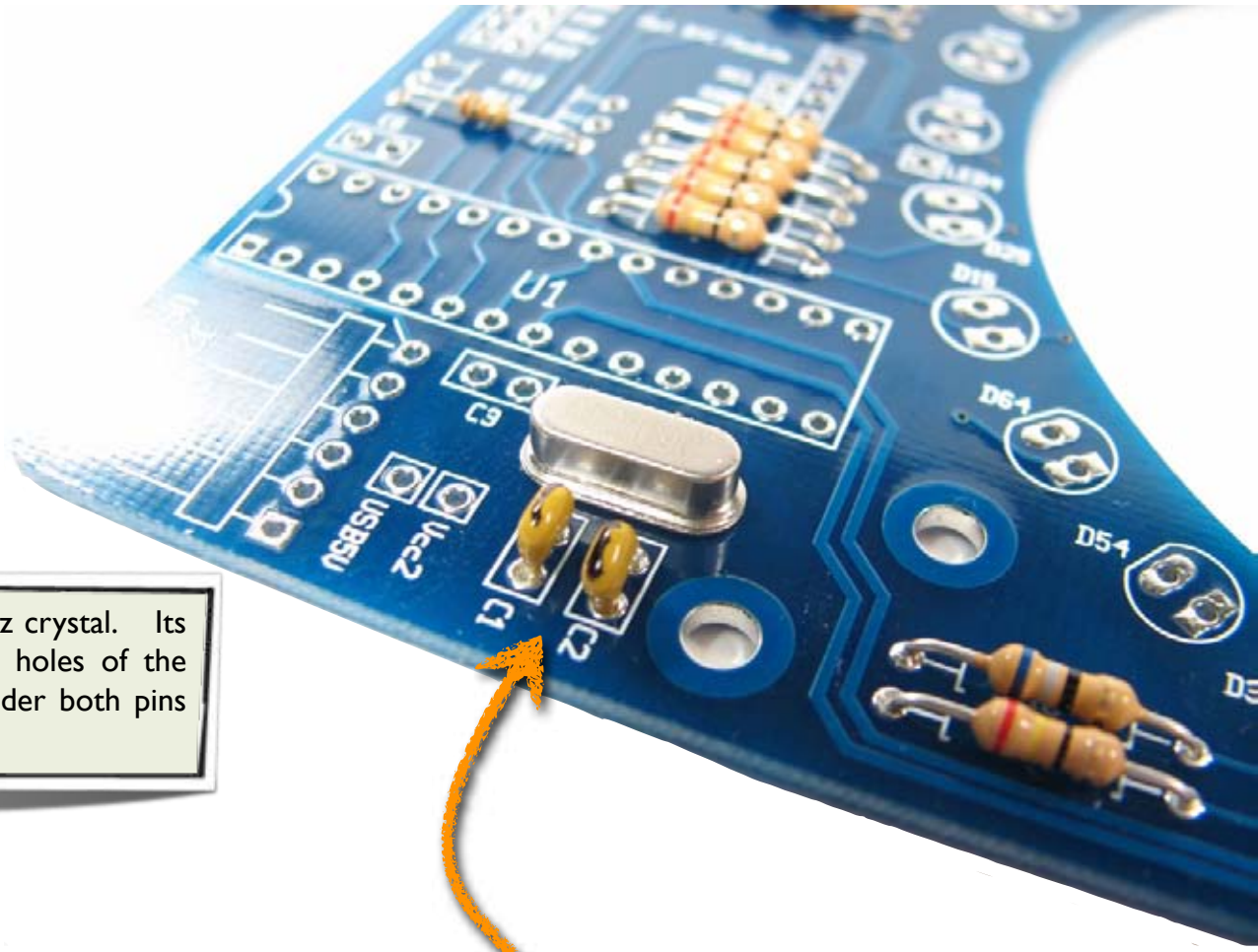
If you have the mono kit: Install one more 24-ohm resistor (**#6**) at location R10.

If you have the RGB kit: There should already be a 68-ohm resistor installed at R10, as shown here.

STEP 7: Crystal & Caps



Install part **#8**, a 16 MHz quartz crystal. Its two pins go in the outer two holes of the location marked "Quartz." Solder both pins in place, much like a resistor.

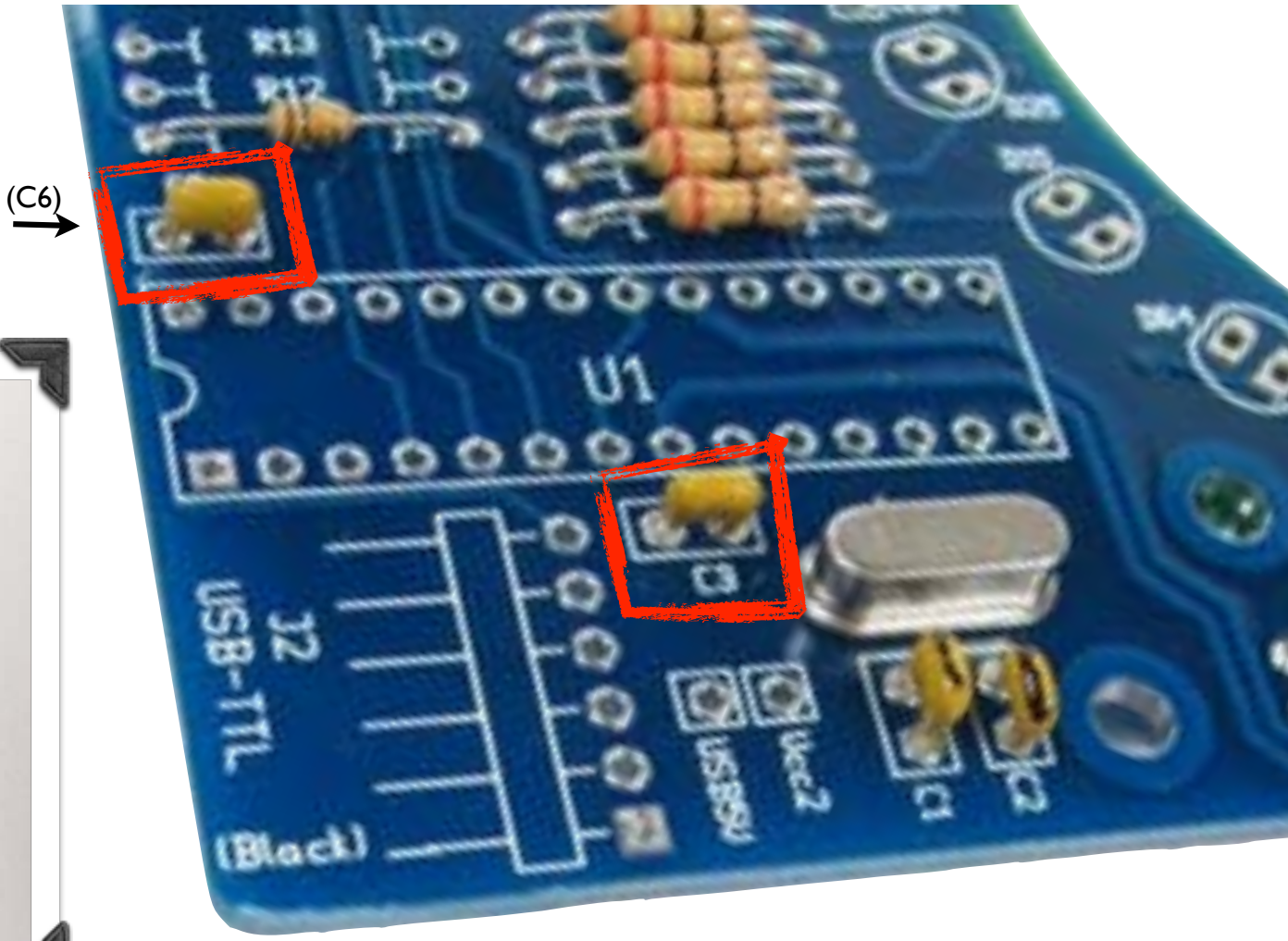


Parts **#9** are 18 pF ceramic capacitors-- little yellow beads with two pins. There are two similar types of ceramic capacitors in the kit, so these ones are marked with a *black stripe*.

Solder these two capacitors on the board at locations C1 and C2.

If they are taped together, pull them straight out of the tape.

STEP 8: Two more caps



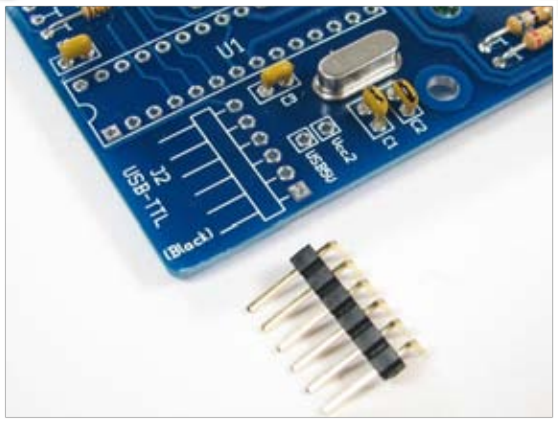
Parts **#13** are 0.1 µF ceramic capacitors that look almost the same as the 18 pF caps. (These ones don't have the black stripe.)

Install two of them, at locations C3 and C6.

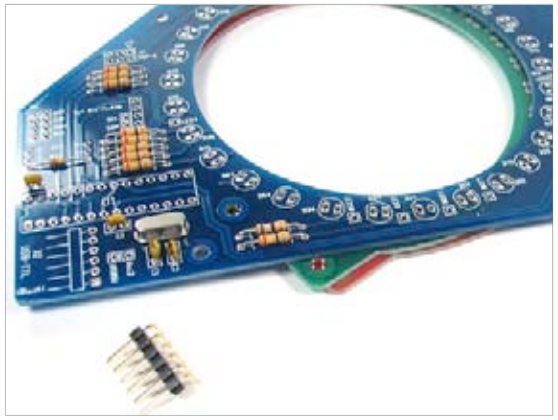
STEP 9: The USB-TTL connector

In this step, we add a header that allows the Bulbdial clock to be connected to a computer through an FTDI USB-TTL cable.

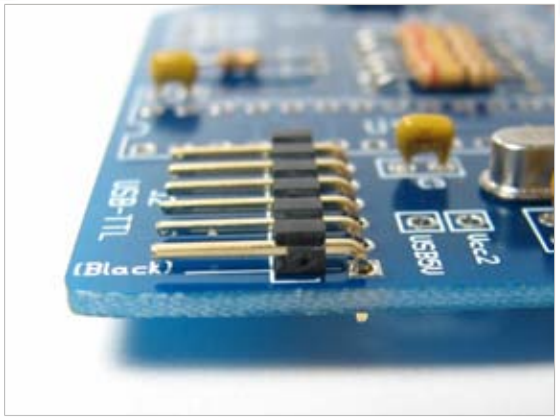
If you do not plan to connect to a computer, you can skip this step.



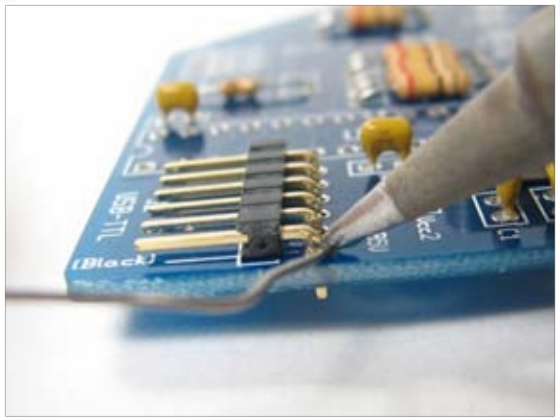
Part **#11** is a 6-pin header that goes at location J2.



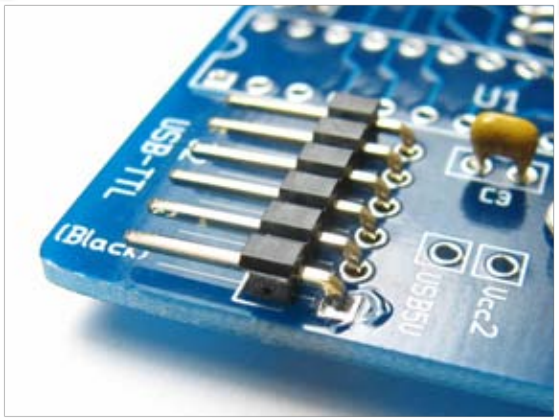
Set the circuit board atop the other two, to raise it a bit above your work surface.



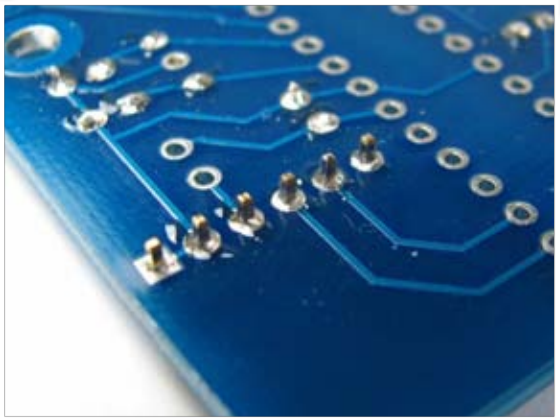
Place the header in its location; it should sit flat as shown.



From the top, solder one pin of the header to tack it in place.



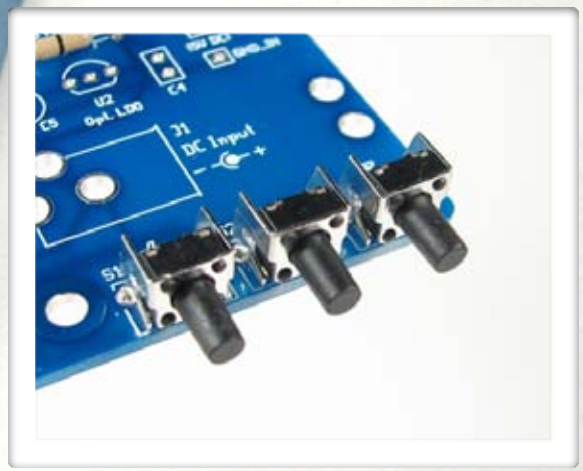
Here's how it looks with one pin tacked in place.



Solder the other five pins on the bottom, and then finish up the first one. The leads do not need trimming.

STEP 10: Buttons!

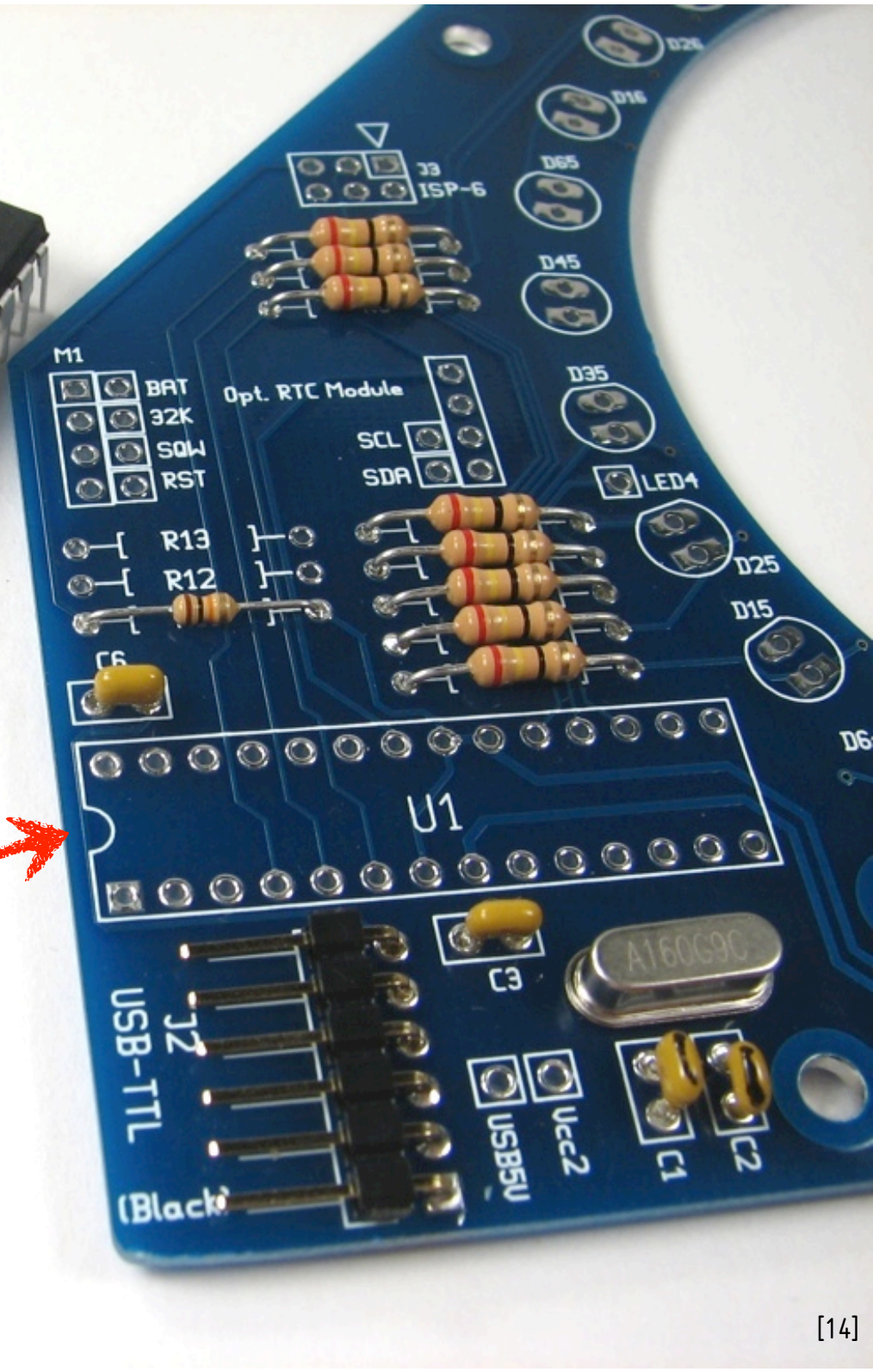
Parts **#10** are right-angle tactile button switches. "Snap" them into locations S1, S2 and S3, and make sure that they sit flat on the board. Solder all four pins of each switch.



STEP 11a: AVR Microcontroller

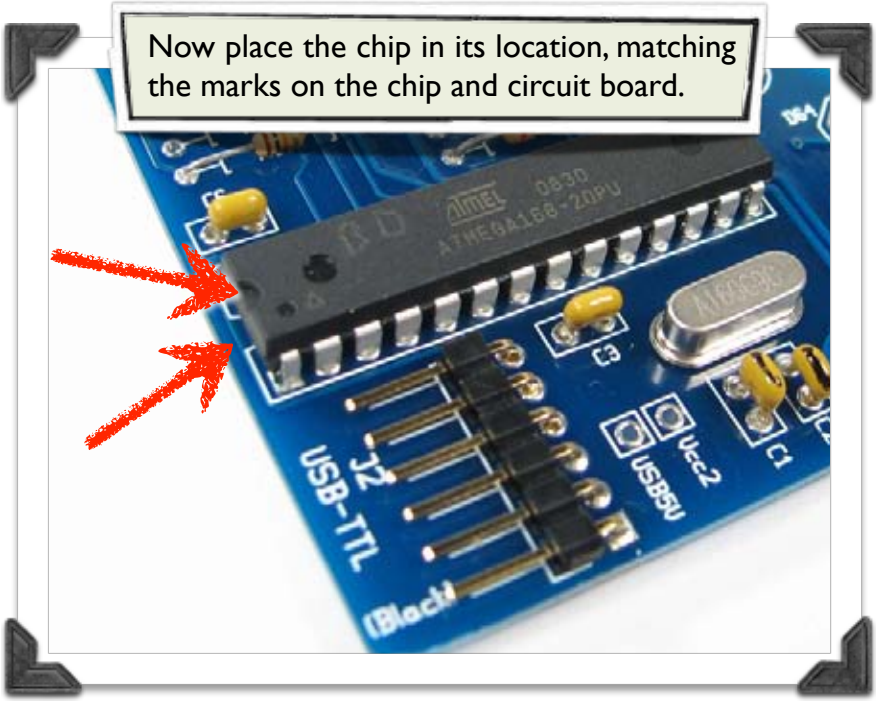
Part **#14** is an ATmega168, a type of AVR microcontroller. It goes in location U1, and must be oriented correctly.

First, identify the end of both the chip and of location U1 that have the half-moon shape.



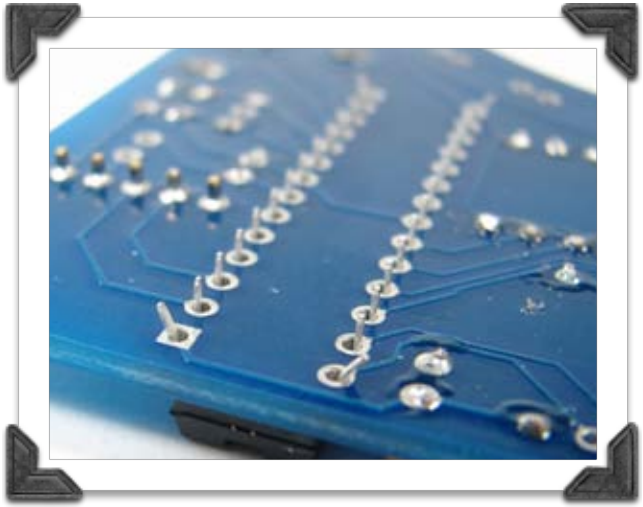
TO BE CONTINUED...

STEP 11b: AVR Microcontroller, continued

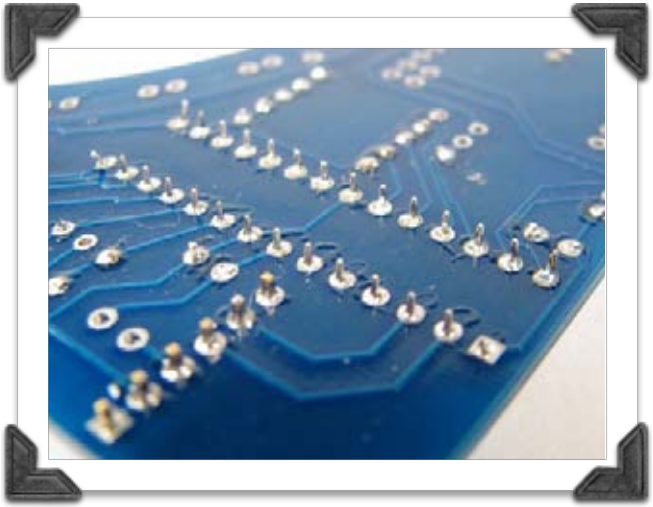


The chip should easily slip into the board. If necessary, bend the pins of the chip to straight up and down before inserting the chip. Do not bend them by hand; bend all pins on one side at a time by pushing them against a hard flat surface.

From end of chip:

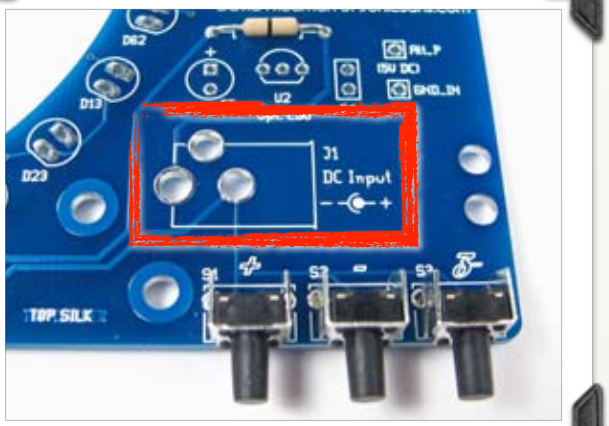


To hold the chip in place while you solder it, you can bend out the corner pins, again by up to 45°.

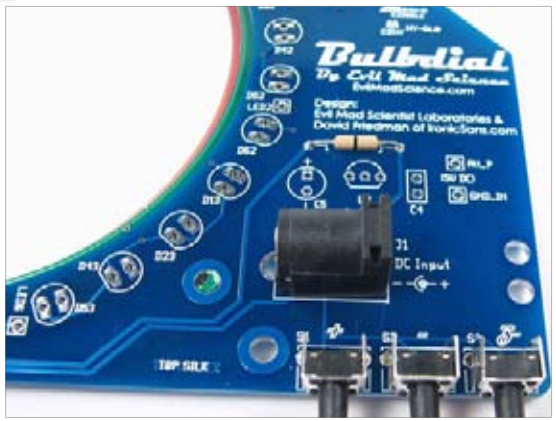


Solder every pin of the chip. Remember to keep the soldering time per pin brief. The pins do not need to be clipped.

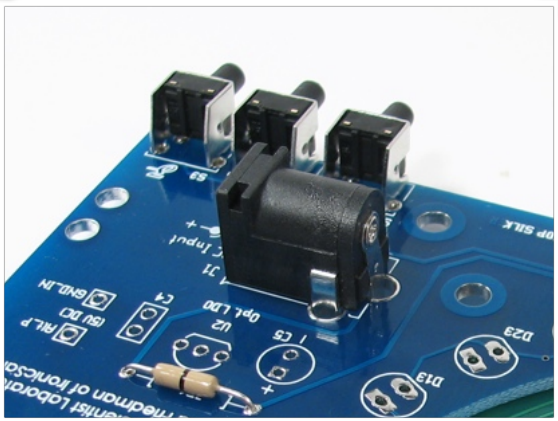
STEP 12: Power jack



The DC power jack, **#15**, goes in location J1, matching the outline on the board.



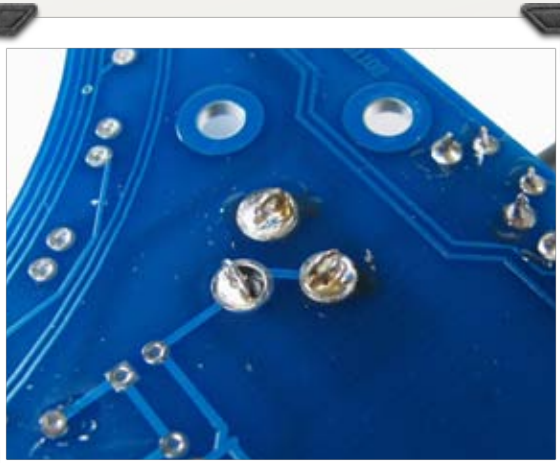
Set the circuit board atop the other two, to raise it a bit above your work surface, as in step 9. Place the jack in its location.



From around back, it is possible to solder one pin of the jack to tack it in place.

Solder one pin of the connector to tack it in place. The larger metal here takes a while to heat, up to about 8 s.

Here's the view from the bottom side after tacking the jack in place. Solder the other two holes as well.



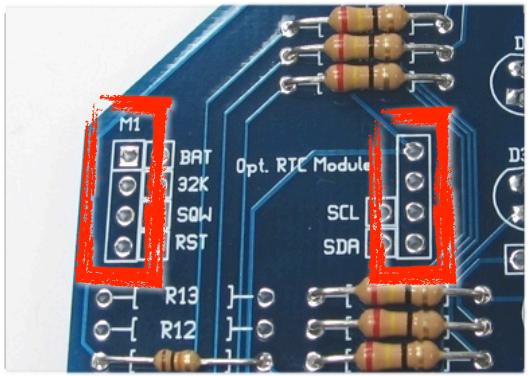
All three holes are now soldered. It is not necessary to completely fill the holes with solder.

STEP 13: RTC Sockets

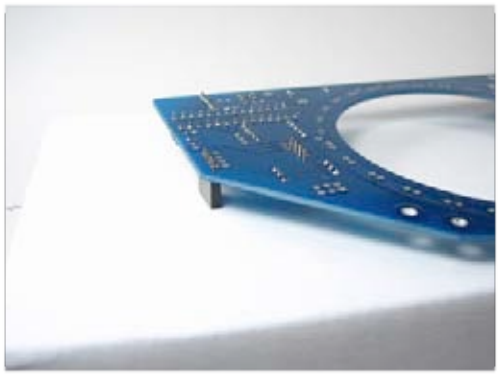
In this step, we add the sockets that allows the Bulbdial clock to be connected to a real-time clock module (e.g., Chronodot)

If you do not plan to use an RTC module, you can skip this step.

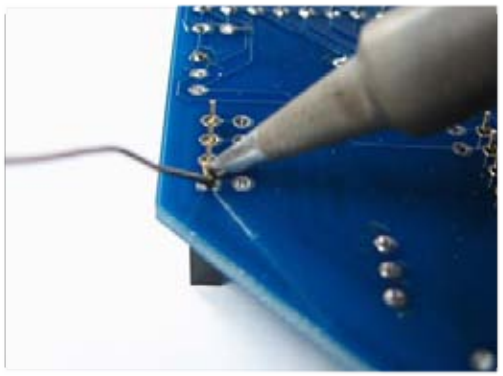
Parts **#12** are 4-pin female headers that go at location M1.



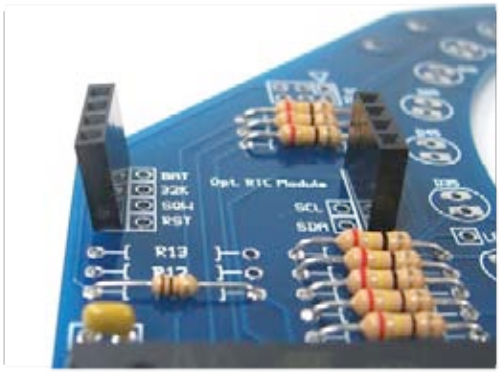
There are actually two locations at M1, the headers go in the outer positions.



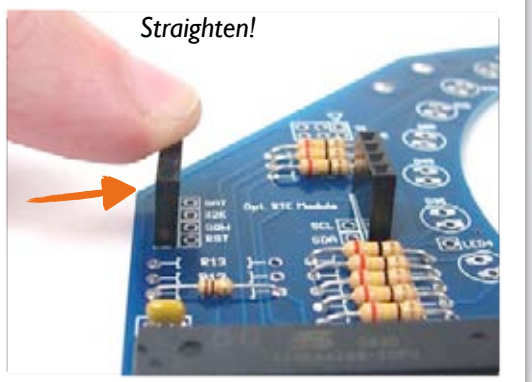
Put the headers in their places. Turn the board upside down to rest flat on the headers. To verify that they are sitting flat, it may be helpful to rest it on top of a pedestal, such as the kit box.



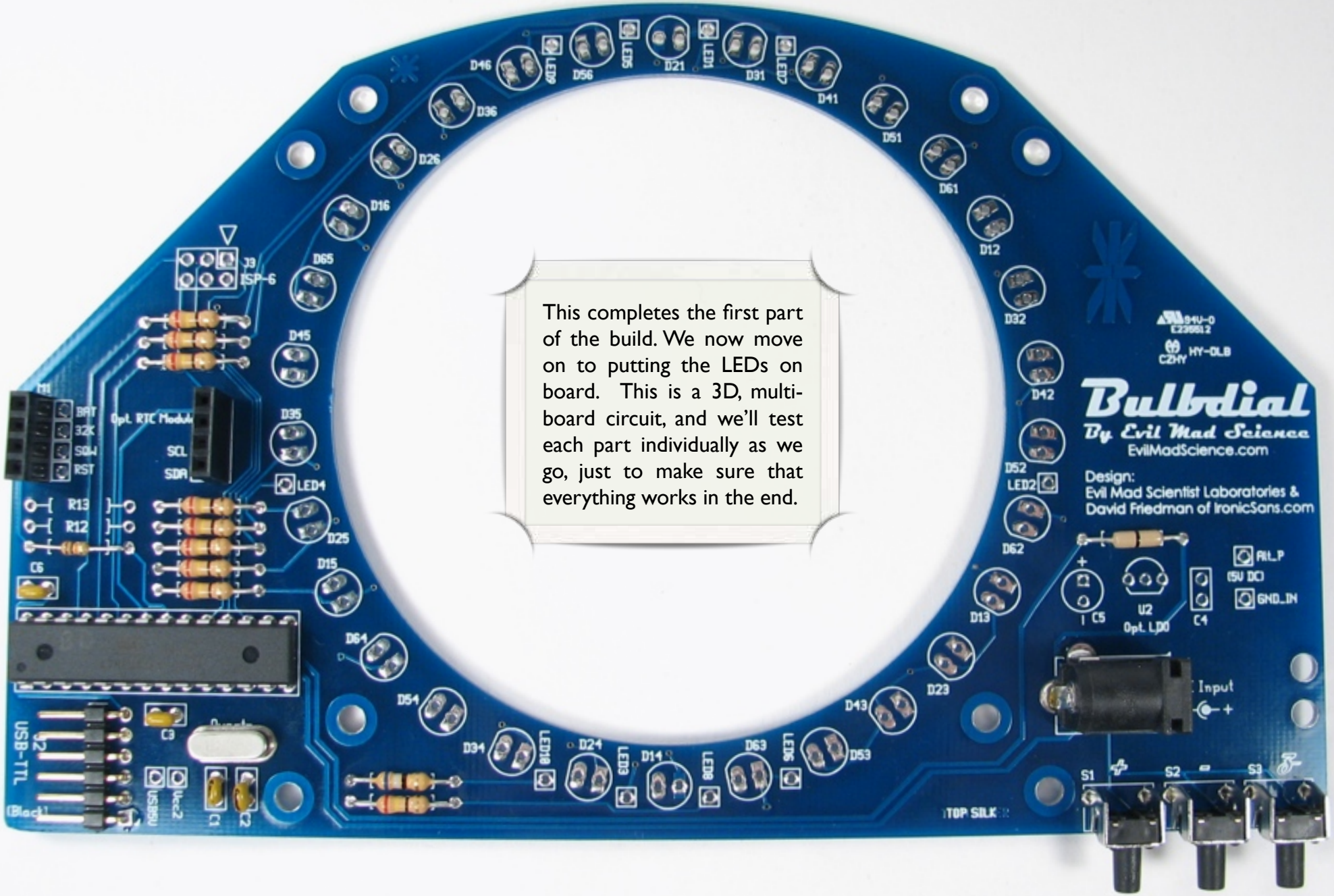
Solder one pin of each header to tack them in place.



With only one pin soldered, the headers are somewhat "hinged"-- do your best to straighten both of them.



Finally, solder the remaining three pins of each header.



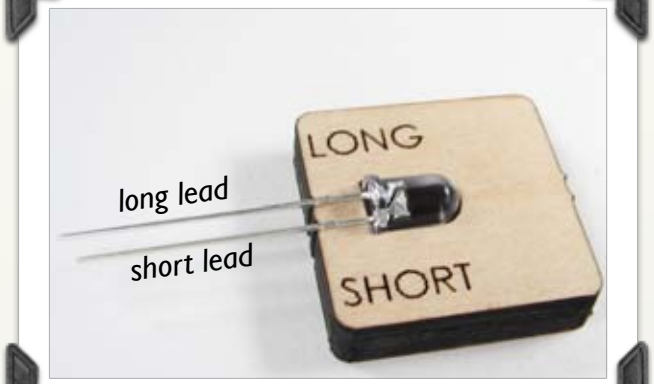
This completes the first part of the build. We now move on to putting the LEDs on board. This is a 3D, multi-board circuit, and we'll test each part individually as we go, just to make sure that everything works in the end.

STEP 14: The LED bending guide

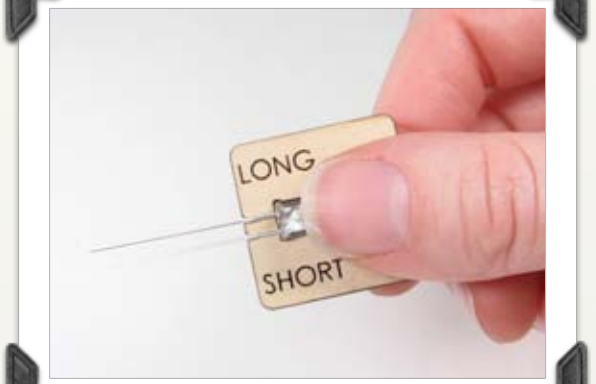
Properly aligning the LEDs to make good shadows is an important part of the assembly process. We use several tools, including this LED guide, to ease this process along.

[If you do not have an LED guide, bend the LEDs 5.4 mm from their base, in the direction shown.]

The LED bending jig is part **#17**. To follow along, grab an LED from your bag of blue LEDs, **#16A**. *

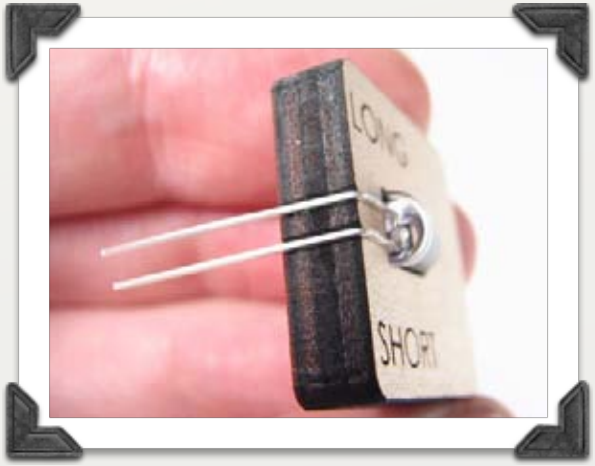


Each LED has a long and short lead. Place an LED in the jig, being careful to put the short lead on the side labeled "SHORT."



Use your thumb to hold the LED body in place.

Use your other thumb, or a firm surface, to bend the leads firmly around the corner.



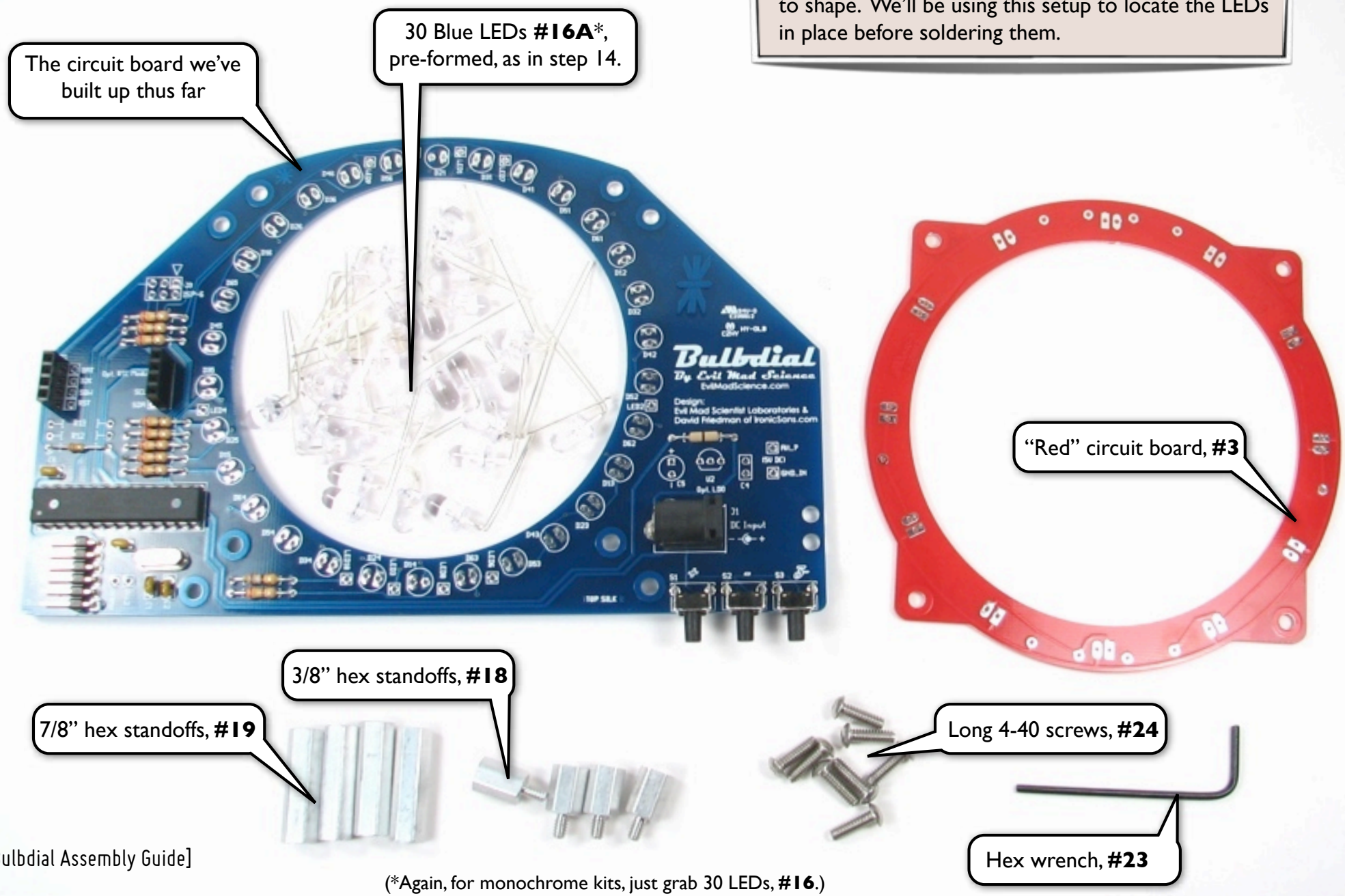
Executed properly, the LED should have a sharp, right-angle bend and not spring out.

The LED, bent with the guide, with its 90° turn intact. Your LEDs should look like this.

(*For monochrome kits, just grab any one of your LEDs, **#16**.)

STEP 15: Get ready to build the "blue" ring.

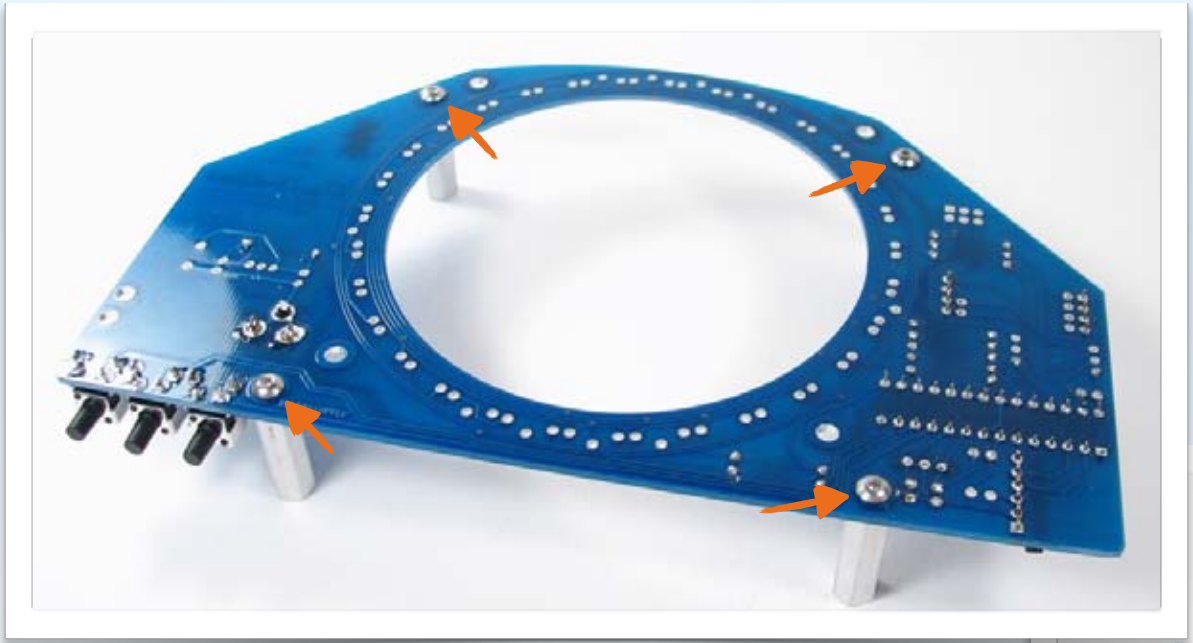
Before starting to put the first ring of LEDs together, gather up the necessary materials and have them at the ready. You'll also need to bend the first 30 LEDs to shape. We'll be using this setup to locate the LEDs in place before soldering them.



(*Again, for monochrome kits, just grab 30 LEDs, **#16**.)

STEP 16: The temporary bridge

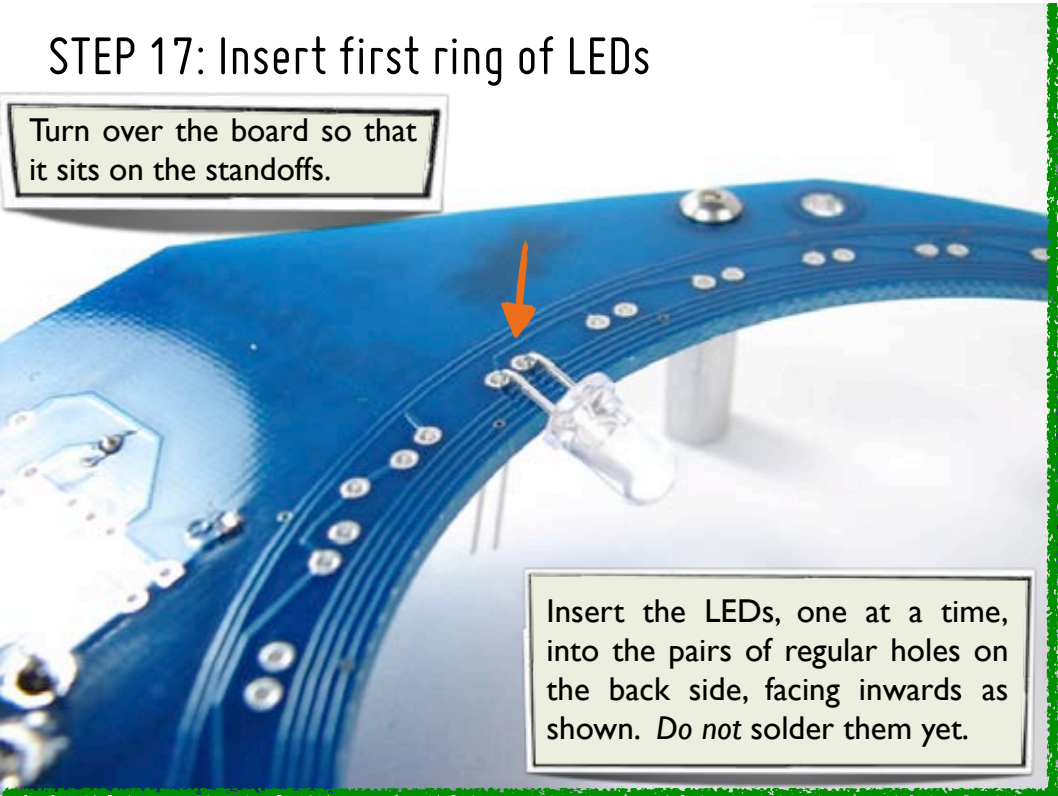
Use the screws to mount the four long hex standoffs on the top side of the circuit board, in the *outer* four holes. (Do not tighten the standoffs; this is a temporary fixture.)



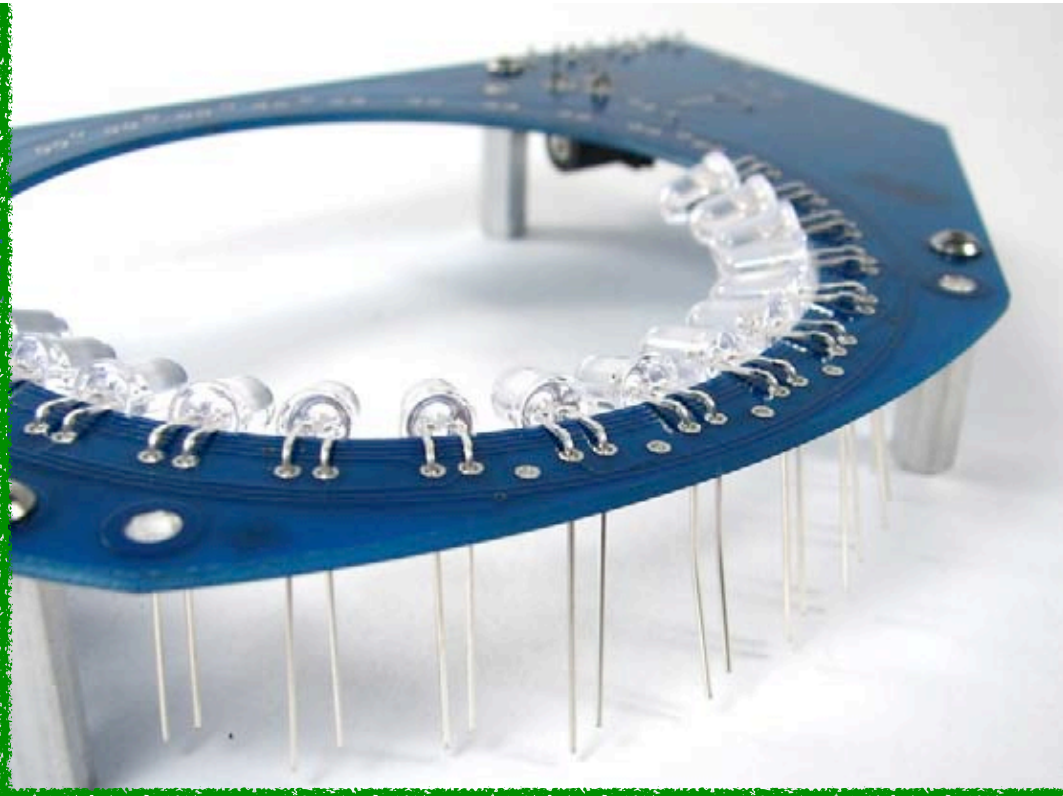
Tip: If you press your finger your finger over the screw head, you can usually screw on the standoffs without using the hex wrench.

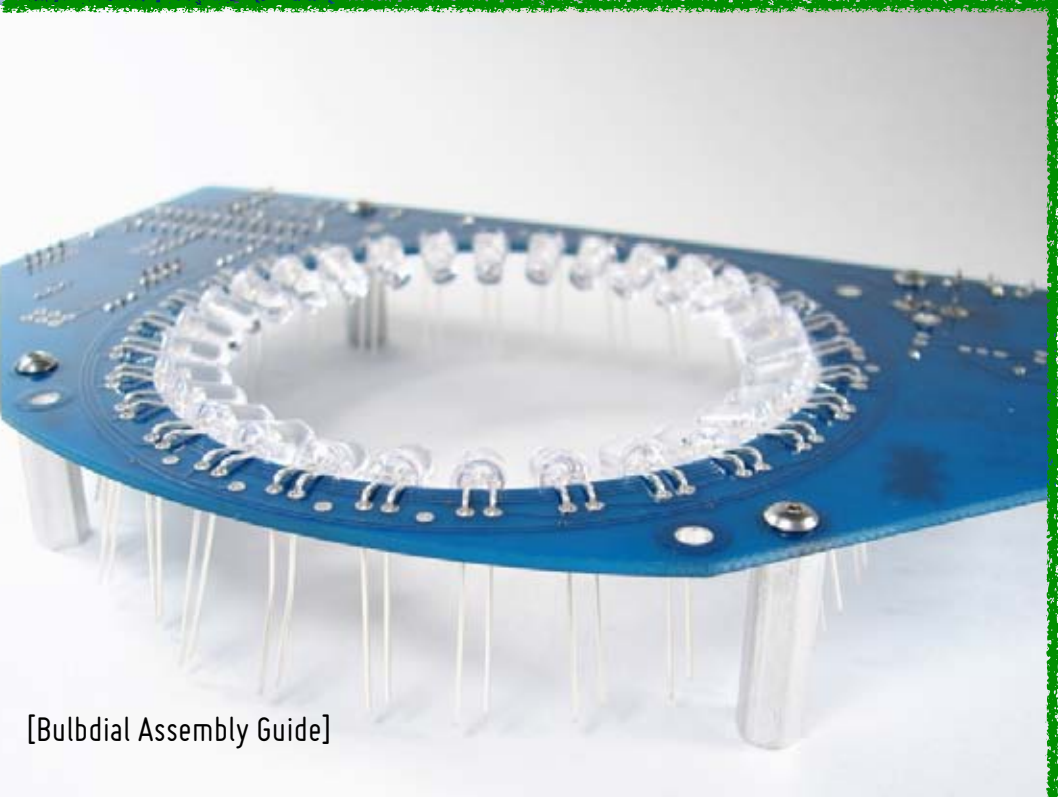
STEP 17: Insert first ring of LEDs

Turn over the board so that it sits on the standoffs.

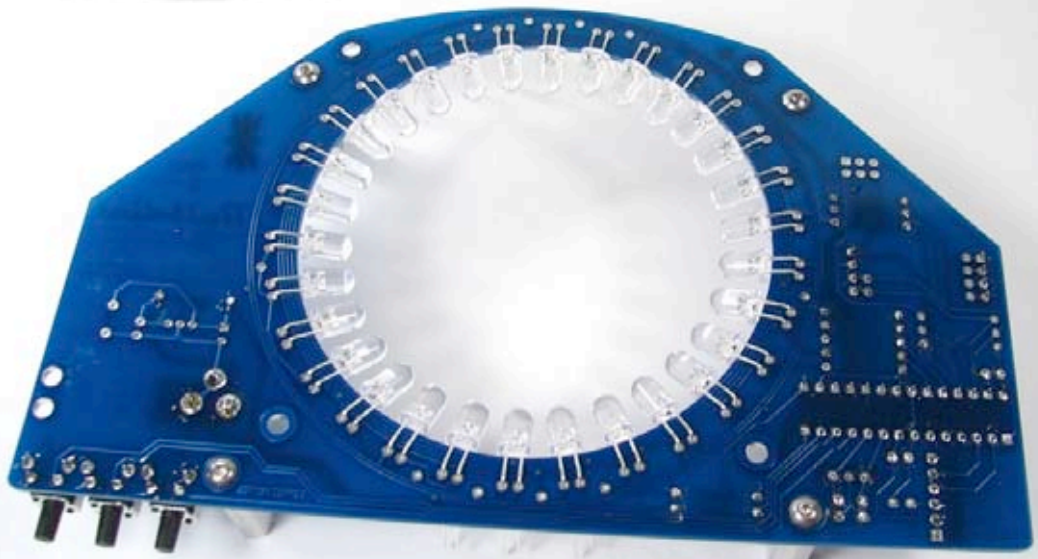


Insert the LEDs, one at a time, into the pairs of regular holes on the back side, facing inwards as shown. *Do not* solder them yet.



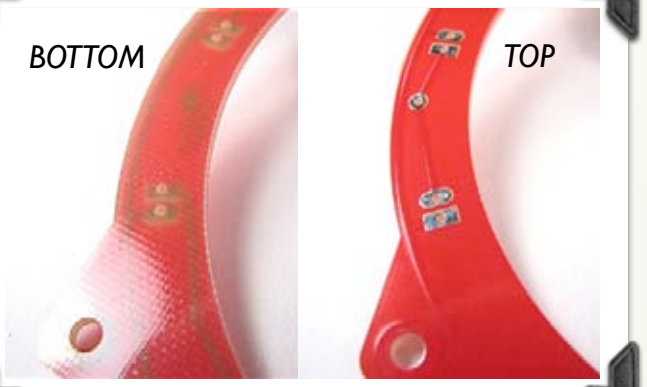


Add all 30 blue* LEDs.

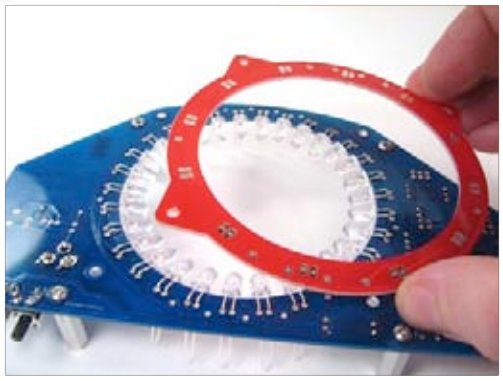


(*For monochrome kit, just "30 LEDs," of course.)

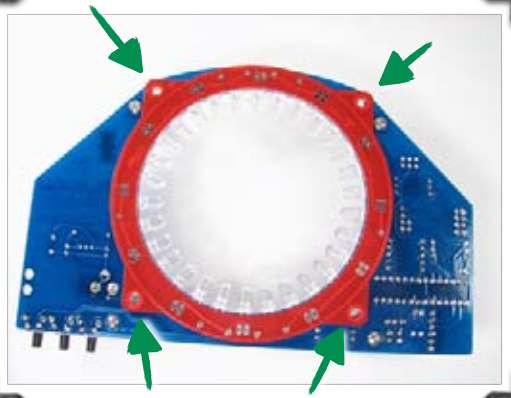
STEP 18: Add the red retaining ring



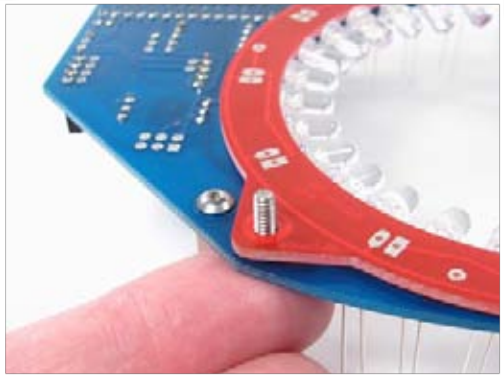
Identify the top and bottom of the red circuit board. The top side has solder features and red ink; the bottom is blank and translucent.



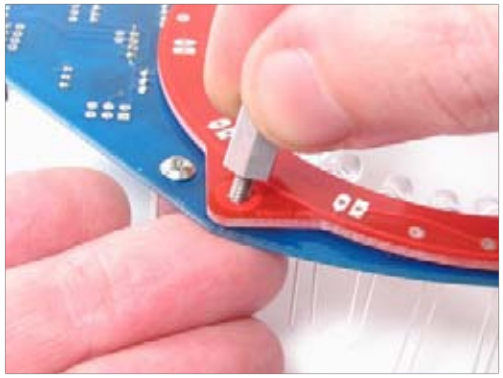
Place the red circuit board over the LEDs. Its bottom (blank) side should rest against the leads of the LEDs.



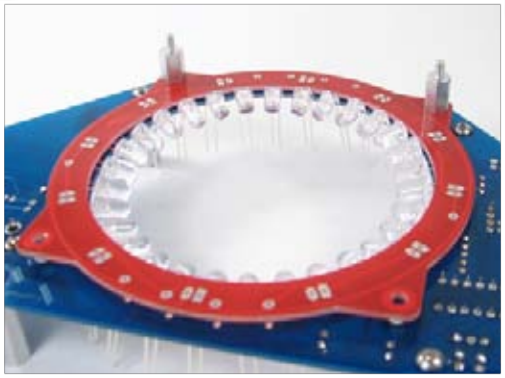
Line up the four corner holes of the red board to the matching holes on the main board.



Without flipping the assembly over, feed a screw up through one of the corner holes.

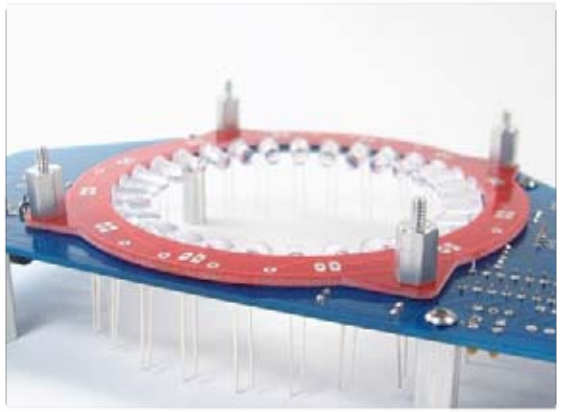


Hold the screw in place while you thread on one of the short M/F hex standoffs. Do not tighten it yet.

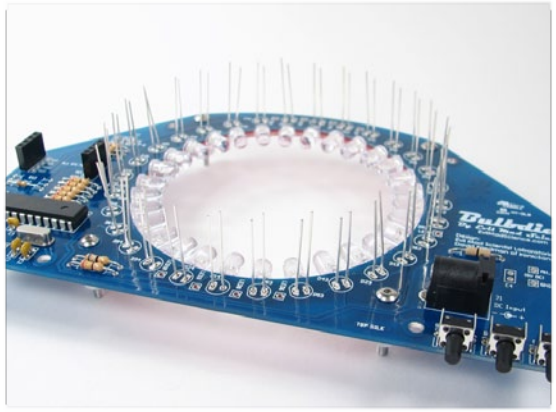


Add the other hex standoffs the same way. Once you have two on board, the LEDs should be protected against falling out.

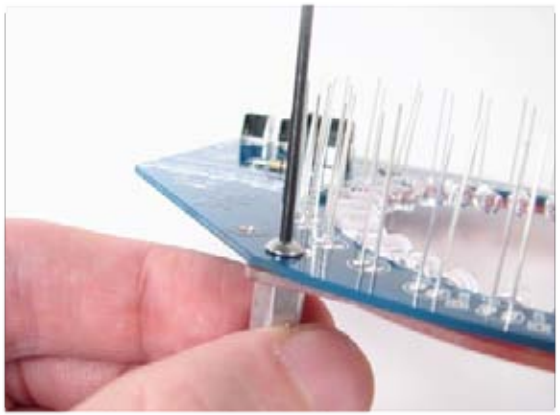
STEP 19: Clamp & inspect the ring of LEDs



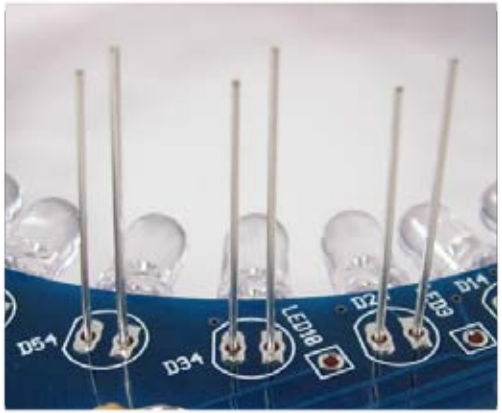
Finish adding the four short standoffs. This loosely "clamps" the LEDs in place.



The four long standoffs can now be removed. (You may prefer to wait until after the upcoming inspection steps.)

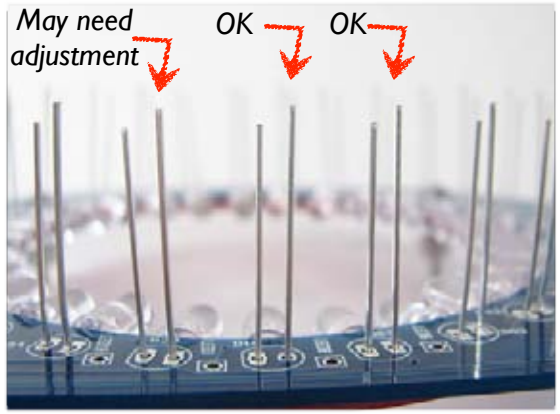


Use the hex wrench to gently tighten the four short standoffs in place. Only go to "finger tight" Do not overtighten.



Inspection I: The LED locations have a squared-off pad and a rounded pad. Make sure– for each LED –that the *short lead* is at the *rounded pad*.

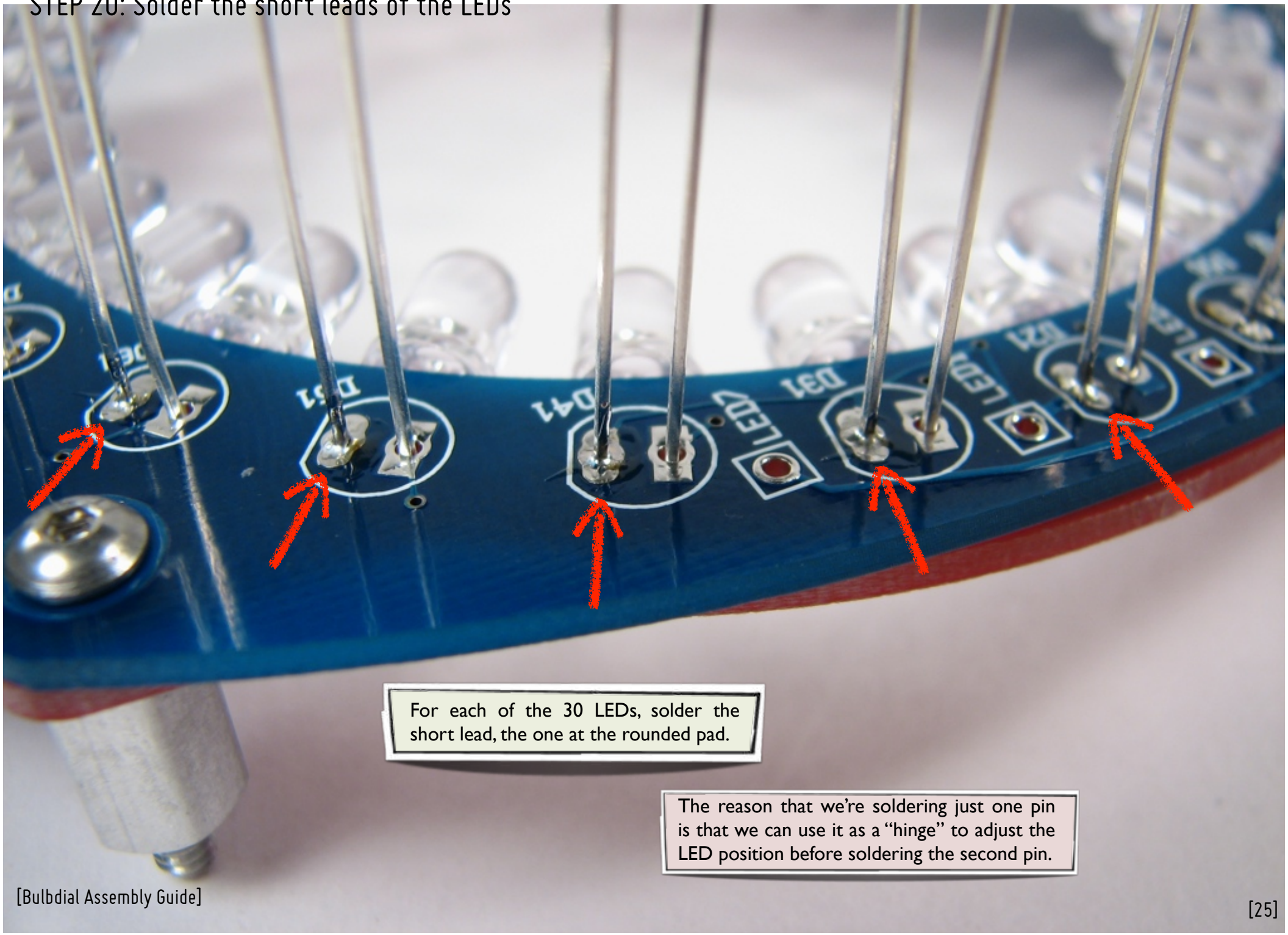
Inspection II: Check that all of the LEDs are roughly pointing towards the center and in the same plane– not wildly pointing up or down.



Inspection III: The leads of each LED should point straight up, not at an angle. If not, nudge the LED to correct it.

After inspection, remove the four long standoffs (if you have not already done so).

STEP 20: Solder the short leads of the LEDs



For each of the 30 LEDs, solder the short lead, the one at the rounded pad.

The reason that we're soldering just one pin is that we can use it as a "hinge" to adjust the LED position before soldering the second pin.