

Solder Time Desk Clock

Build Guide v0007



This guide will show you how to solder and assemble the SpikenzieLabs Solder Time Desk Clock Kit.

For the best outcome, follow each step in order. This is a relatively easy kit to build, however there are certain steps that need to precede other steps. Read through the entire guide, and check your parts. Questions or problems, let us know at support@spikenzielabs.com

These are to call attention to certain elements of the build that require additional care & attention.



Caution

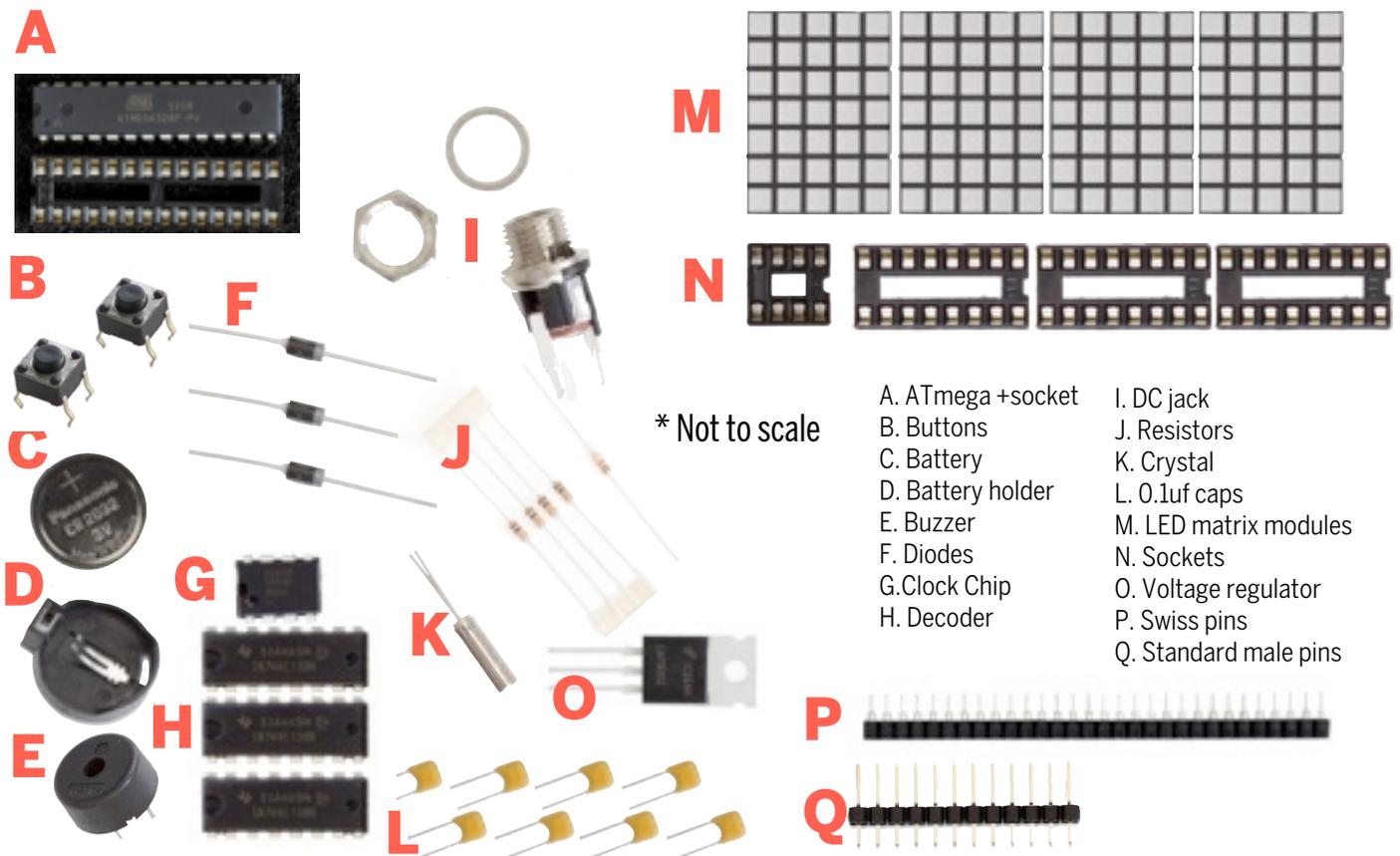


Look here...



Info

Unpack the electronics parts.



Preparing the resistors

Take resistors, and snip them from the tape. Bend the legs as close as you can to the body of the resistor for all of them, as in the picture (right).

Note the color bands. You have four resistors that are 10k, (brown - black - orange) and one 100 ohm (brown - black - brown).

They all get the same bend, but it is important to solder them in the correct locations on the PCB.

The value for the required resistor is printed on the PCB to reduce mistakes.



One : 100 ohm



Four : 10K ohm

credit: samengstrom.com

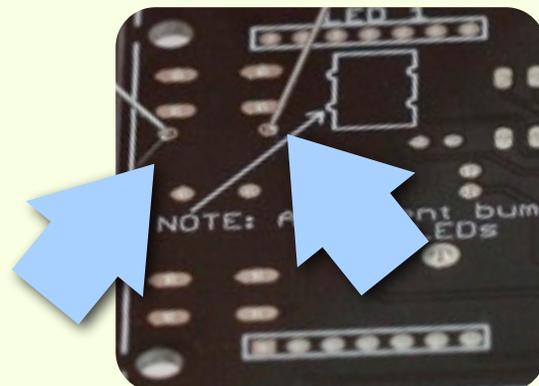


Preparing a 100 ohm resistor

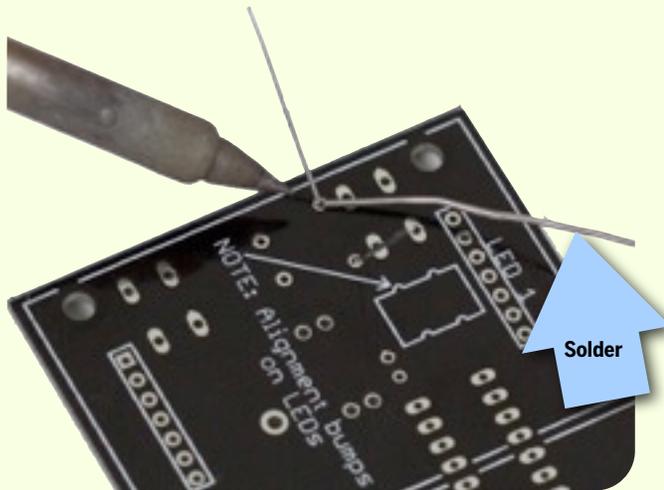
Placing and soldering the resistors

Start with the single 100 ohm resistor. It gets placed in either orientation (no polarity). If you have bent the legs properly, the resistor will slide all the way down, and the resistor body will touch the PCB.

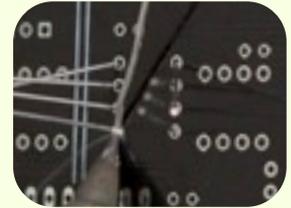
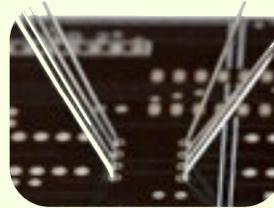
Secure the resistor in place with some low tack painter's tape, or flare the legs out to hold it in place. Double check to make sure it hasn't shifted, before soldering in place.



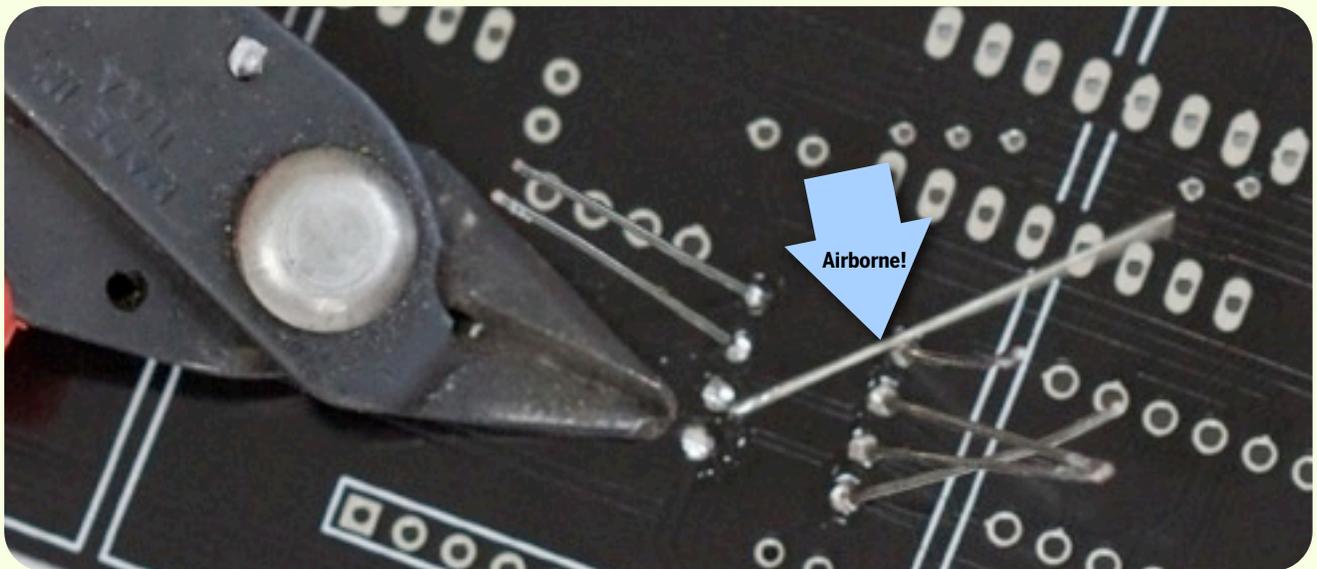
Solder the legs.



Repeat for for the 4x 10k ohm resistors



Snipping legs of components is about to start. If you haven't put on a pair of **safety glasses** yet, now is the time. When snipped, the bits get airborne, and somewhat unpredictable.
Working with someone? They need a pair too!



Carefully snip the excess legs. Use caution to not scratch the surface of the PCB.

Don't pry or pull up on the leg with the snips. **It is better to snip twice than to scratch the PCB.**

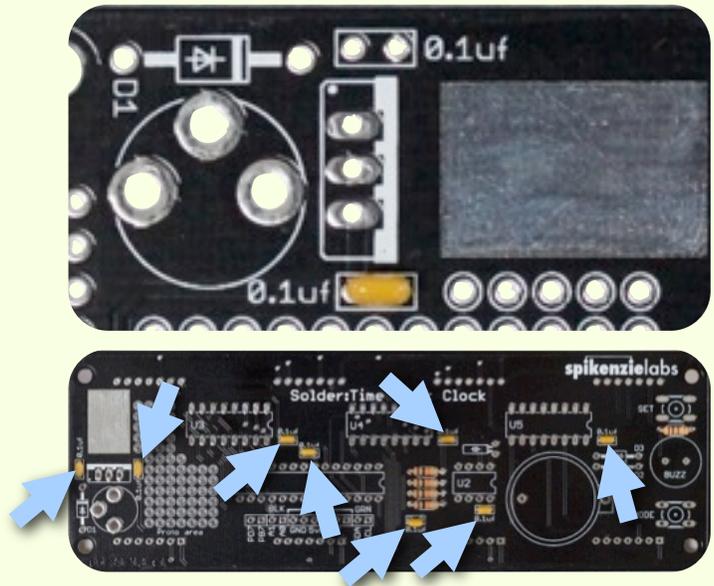
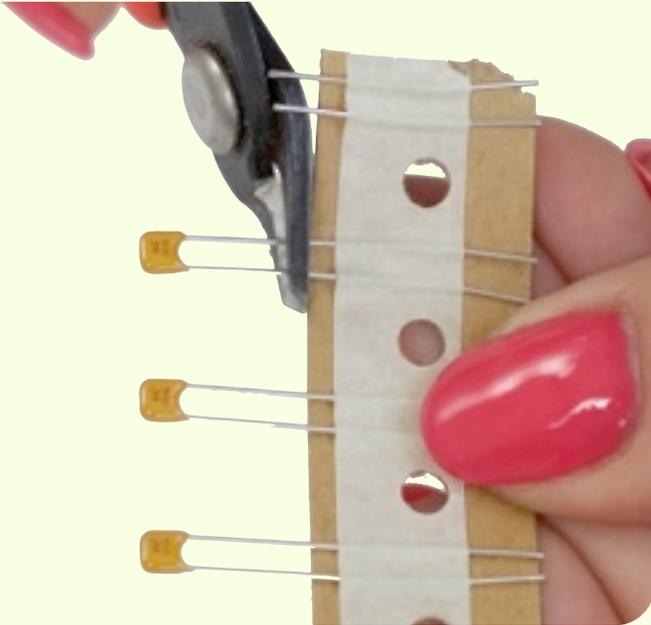


Preparing and placing the capacitors

There are eight 0.1uf capacitors included in your pack.

Remove them from the paper backing by snipping them free. (see below)

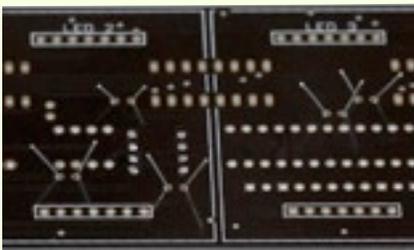
The capacitors get placed in the white boxes marked 0.1 uf on the PCB. Slip the capacitors in place, flare the legs, or tape them down in preparation for soldering. This type of capacitor does not have polarity, you can insert them either way.



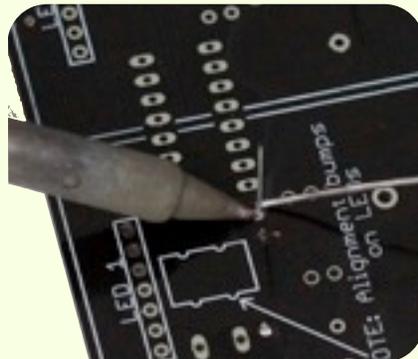
Placements of the 0.1uf capacitors



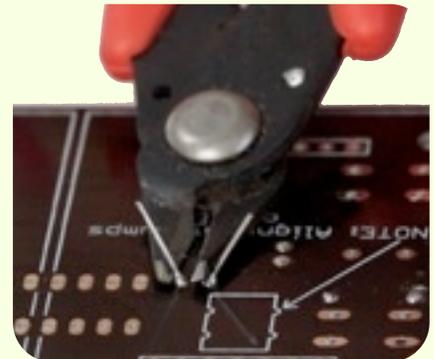
Did you remove your safety glasses? Put them back on for this step.
It is recommended to keep them on throughout.



Flare out the legs

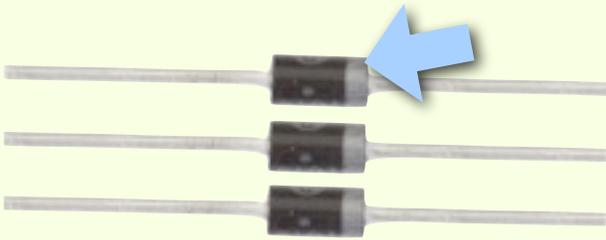


Solder in place



Snip the excess

Preparing and placing the diodes

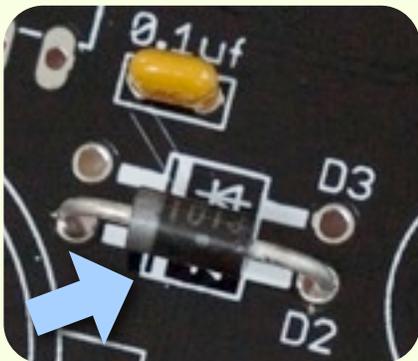


The Solder Time Desk Clock uses 3 of these diodes. Take a moment to notice that they have a grey stripe one end. **These have polarity**, and can only be installed in one way.



Grasp the diode a little bit away from the body, and bend the legs similar to the way you prepared the resistors.

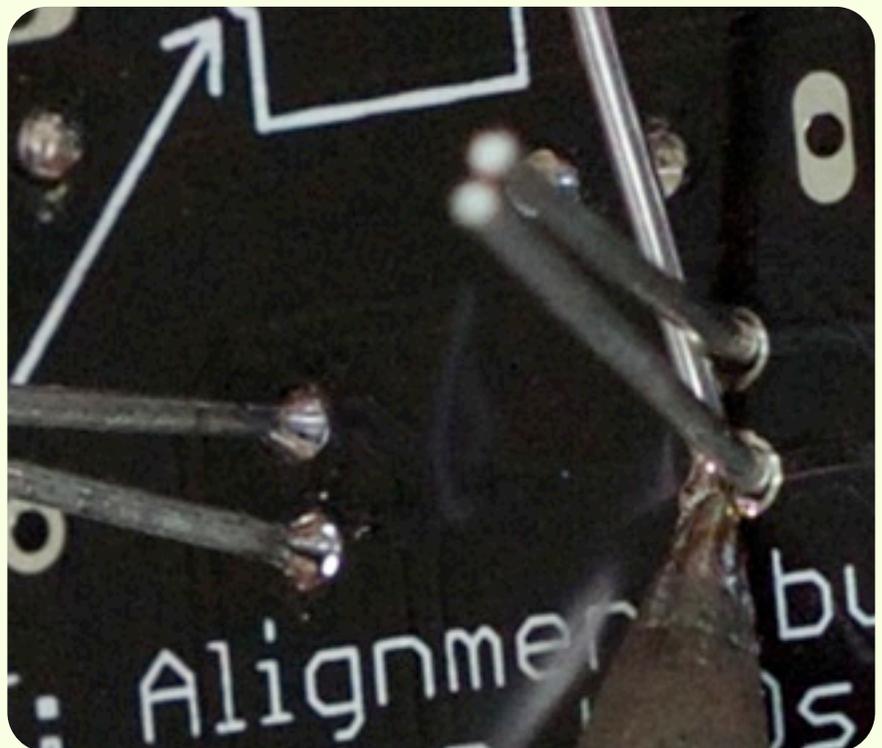
These diodes get their legs bent a little bit away from the body.



Note the white stripe on the PCB.

Match the stripe with the grey stripe on the diode

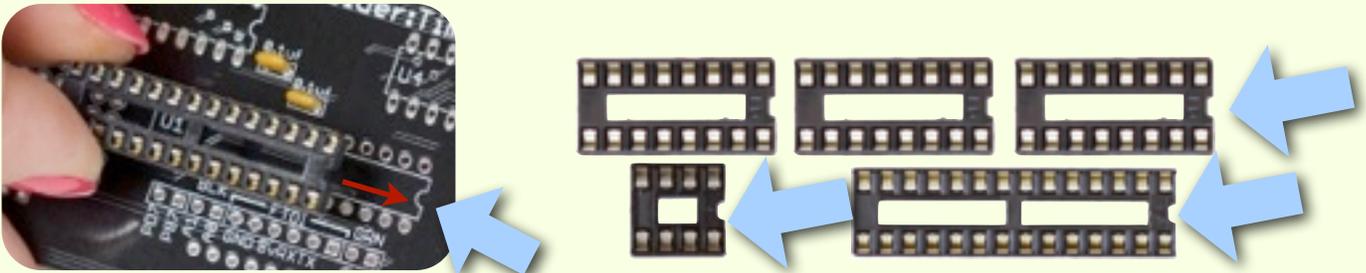
Flare the legs, solder then snip the excess.



Once you have soldered and snipped these diodes into locations D1, D2, and D3, continue to the next step.

The IC Sockets

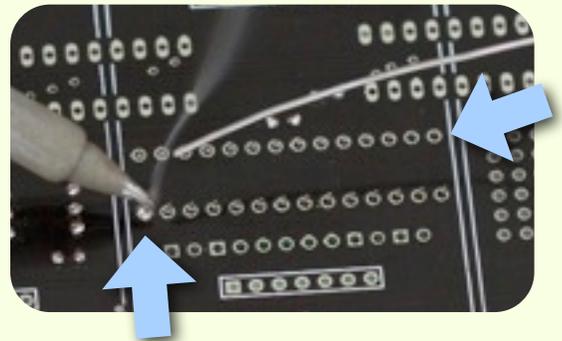
The Solder Time Desk Clock has 5 sockets. Each of them has a notch on one end. When you are soldering these in place, be sure to match the notch with the white printing on the PCB.



Hold each socket in place with tape, or carefully with your finger while soldering. Solder the first pin on one row, and then the last pin on the other row. Double check to make sure all of the legs are poking through, and the socket is laying as flat as possible on the PCB. If it's uneven, reheat the first and last legs while pressing the socket into place.

Once you're happy with how it looks, continue soldering the remaining legs. Trim, and move on to the next socket.

Be sure to solder all of the legs.



Placing and soldering the buzzer

Every great clock needs an alarm function. The Solder Time Desk Clock has a little piezo buzzer that gets soldered to the PCB. It is not polarity sensitive, so you can install it in either orientation.



This is the buzzer.



Buzzer held in place with tape.



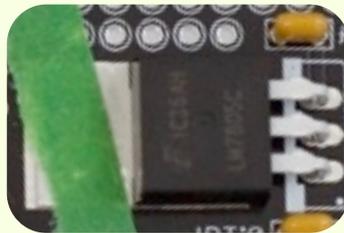
Solder and trim the excess.

Placing and soldering voltage regulator

The voltage regulator needs to have its legs bent at a 90 degree angle. The best way is as in the photo below:



Place it onto the PCB, the three legs through the holes, and the metal back plate touching the exposed metal pad on the PCB. Tape in place.



Solder the three legs, and trim the excess legs.



You do not solder the metal base of the regulator.

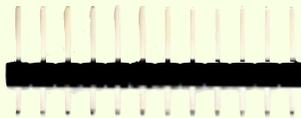
Battery holder placement and soldering

The Solder Time Desk Clock uses a standard CR2032 cell as a backup for the time. Match the outline of the battery holder placement on your PCB, secure and solder in place.

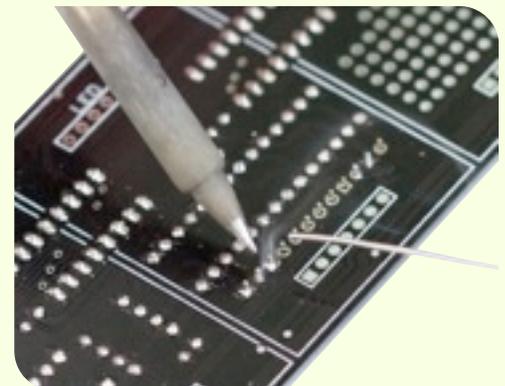


Soldering the FTDI / Hacking pin header

The 12 pin header that is included in the kit gets installed next.



Place the shorter legs against the component side of the PCB, and then solder from the LED side. Hold the pins in with a piece of tape, or carefully with your finger on one of the pins that you're not soldering.



The Swiss pins: Placement and Soldering



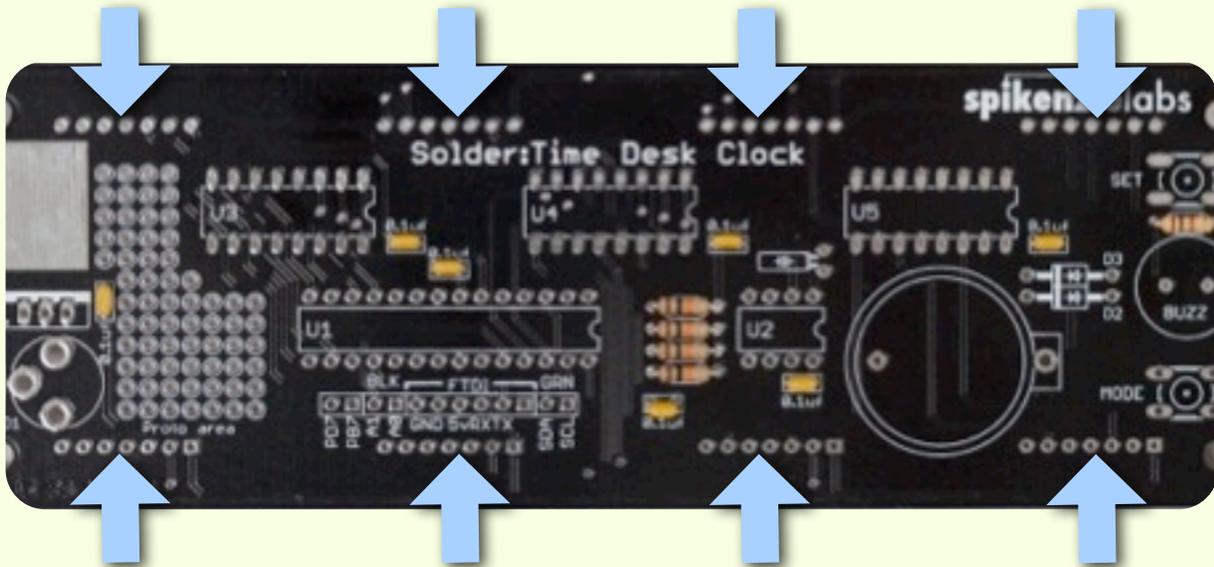
The Swiss pins get soldered to the LED matrix side of the PCB. Make sure your soldering matches these photos exactly.



This kit uses Swiss pins to hold and connect the LED matrix displays to the PCB. Snip the pins into 7 pin strips. You will need eight strips in all.

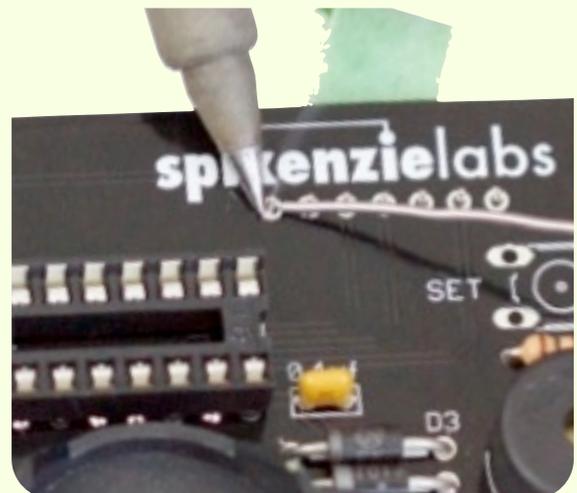
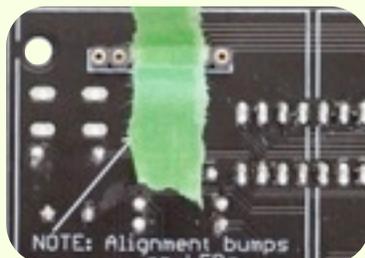
Locations where you are going to **SOLDER** the Swiss pins:
The Swiss pins will be inserted through on the other side. The arrows show where you will be soldering. **Be sure not to insert the Swiss pins on the wrong side!!**

Apply Solder
Where indicated
by blue arrows



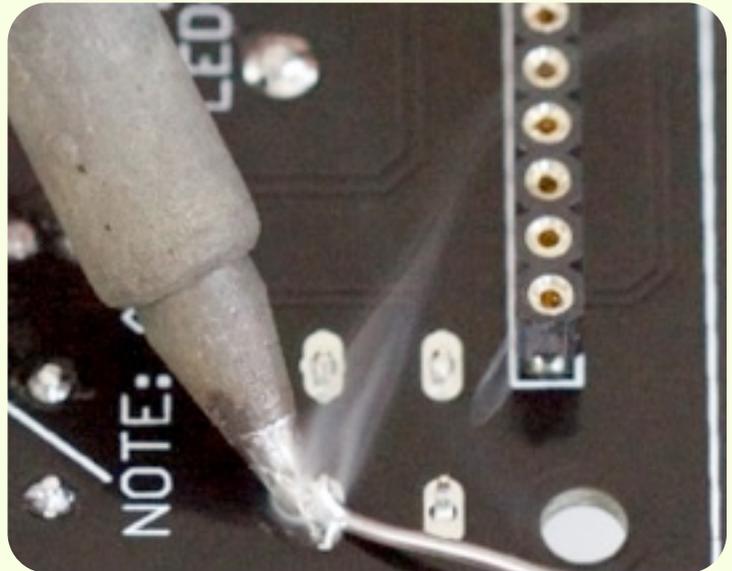
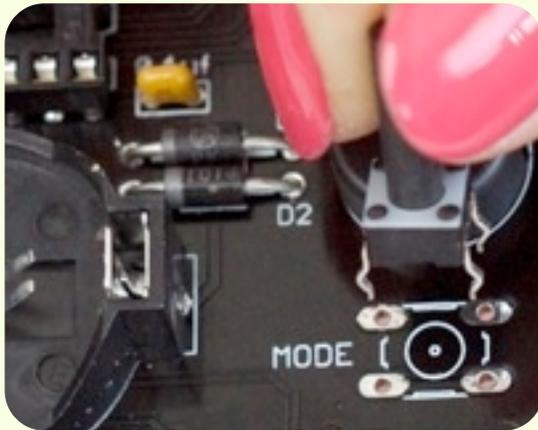
The Swiss pins get soldered to the LED matrix side of the PCB. Make sure your soldering matches these photos exactly.

Swiss pin inserted properly:



Buttons placement and soldering

There are two momentary buttons in your kit. Place them in their mounting positions beside the text '**mode**' and '**set**' on the PCB.

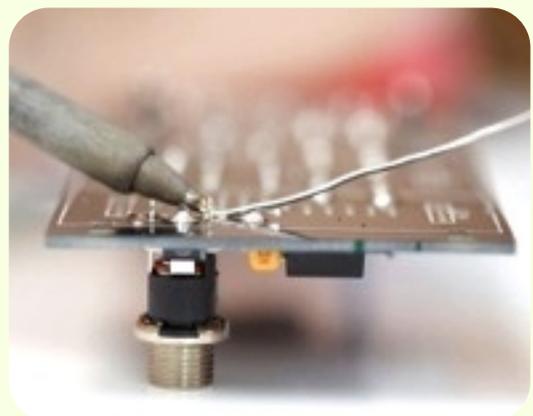
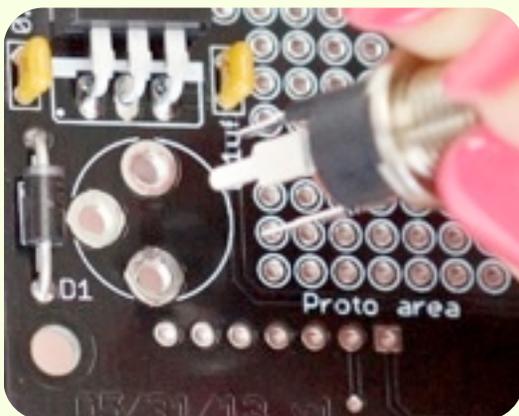


Solder the 4 legs. Repeat for the other button

Placing and soldering the DC jack

Place the DC barrel jack as in the photos below left. Hold it in place with a piece of tape, and solder one of the three legs.

Make sure it is perfectly flat against the PCB, it will bottom-out on the thicker part of the legs, leaving a small gap. Adjust it to be perfectly level, and then solder the other 2 pins.



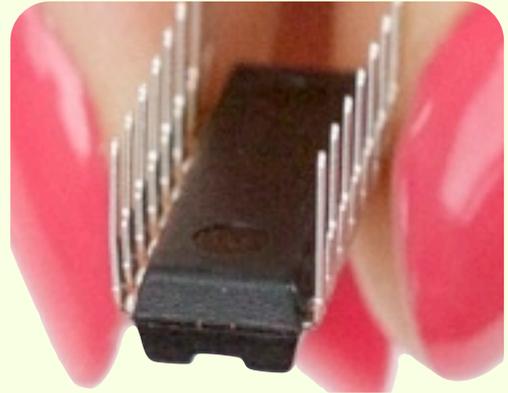
Preparing the Chips.

There are five chips that make the Solder Time Desk Clock work. One 8 pin RTC (Real time clock). Three 16 pin 75HC138 decoders and one 28 pin (ATmega microcontroller). Before we install the chips, we need to slightly bend the legs perpendicular to the body of the chip. The easiest way is like this:

Do this to each of the five chips



Hold on the ends, press evenly
against a flat surface



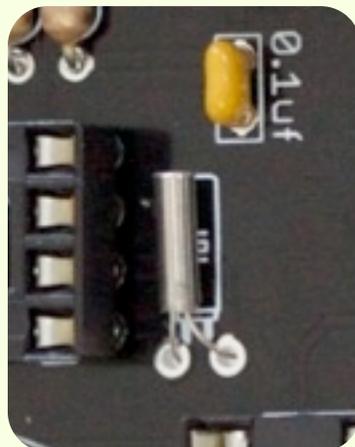
Perfectly perpendicular.
Legs are square to the body.

Place and solder the crystal

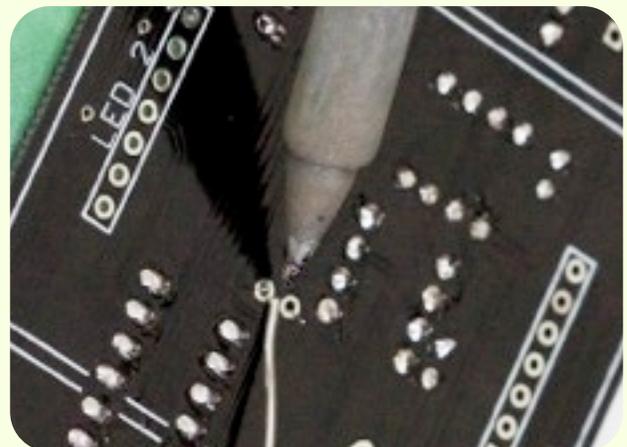
The crystal is a small thin metal barrel, the legs are very thin. Place the crystal legs through the holes as in the photo below middle. Hold it in place with a thin piece of tape, and solder.



The crystal



Spread the legs, and insert
one of the crystal legs into
each hole



Fill the hole with solder. The legs
on these are thinner. Make sure
there is good contact.

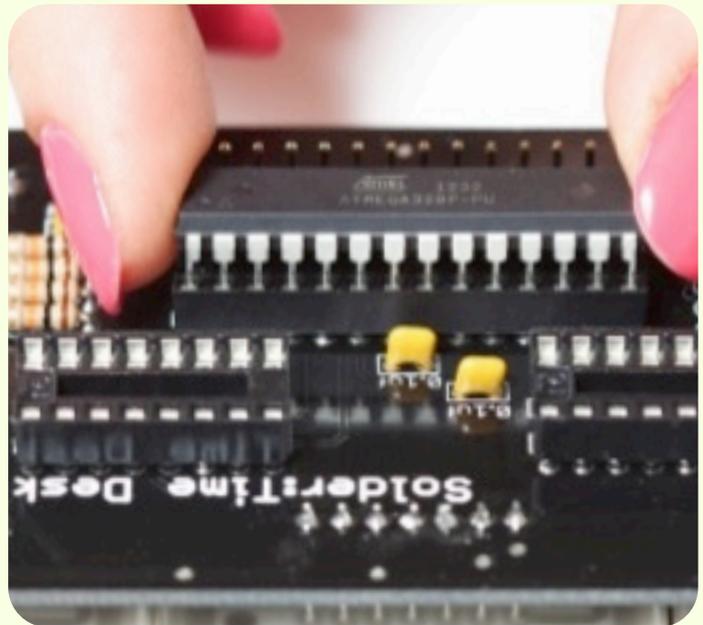
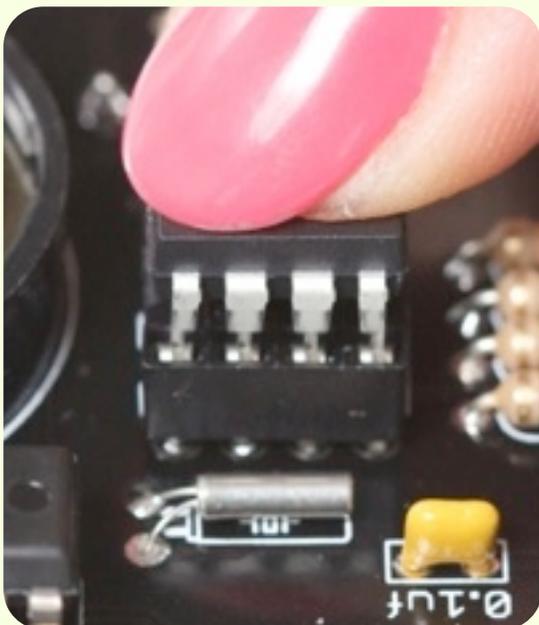
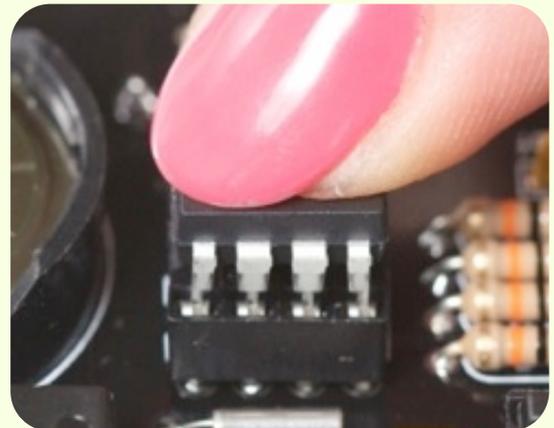
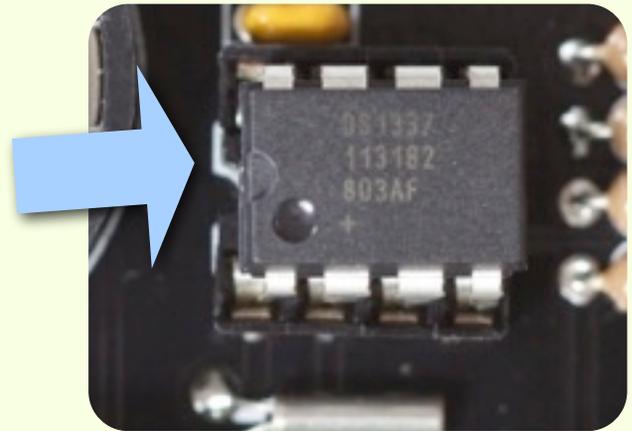
Chip insertion

With the chips now having their legs ready for installation, we are ready to mount them into the sockets on the PCB. Start with the 8 pin RTC chip. Look at the notch that is on one end of the chip, and line it up with the notch that is on the socket, and the PCB. The **notch** is there to ensure that the chip doesn't get installed in the wrong orientation.

Place the chip over the socket, and check to make sure that none of the legs are bent too far towards the middle of the chip, and also that they are not aimed over the side of the socket. Ideally they will be aiming directly into the slots of the socket.

Gently push evenly down until the chip stops, and look around the chip once again. If any of the legs have buckled under, or missed the holes in the socket, remove the chip, straighten the legs, and try again.

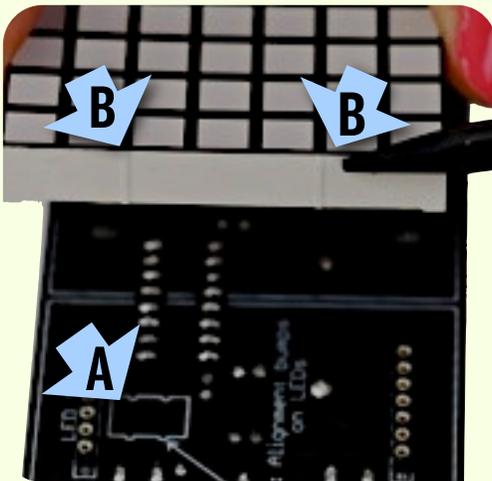
If the legs have been properly straightened in the previous step, this should go very smooth.



Display mounting

Before mounting the assembly into the housing, we are going to mount the LED display modules, and test. The ATmega ships pre-programmed, so all we need to do is install the LEDs and test power up.

The LCD modules need to be installed in the proper orientation. Much like the chips, there are notches along the side to denote which side is which.



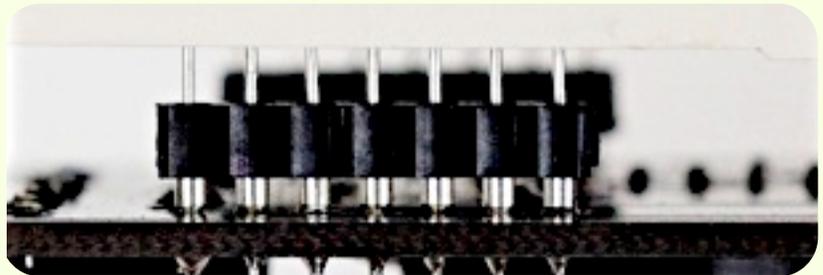
Hold the display along the short sides, pins down as in the photo (left). One side has 2 bumps, and the other side 2 recesses.

Match the bumps and recesses (arrows 'B') to the direction printed on the PCB. (arrow A)

Making sure that all of the pins under the LCD module are slipping inside the Swiss pins, press evenly until the display module stops. You should have a gap that looks like the photo below.



The displays will only work if they are installed correctly.



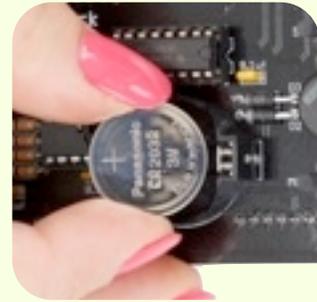
Continue placing the other 3 LED modules by matching their protrusions and recesses one after the other.

You may need to adjust the pins by hand so that they mate with the Swiss pins.

The swiss pins hold the matrix modules in tight. Take your time inserting the modules, you don't want to bend the legs.

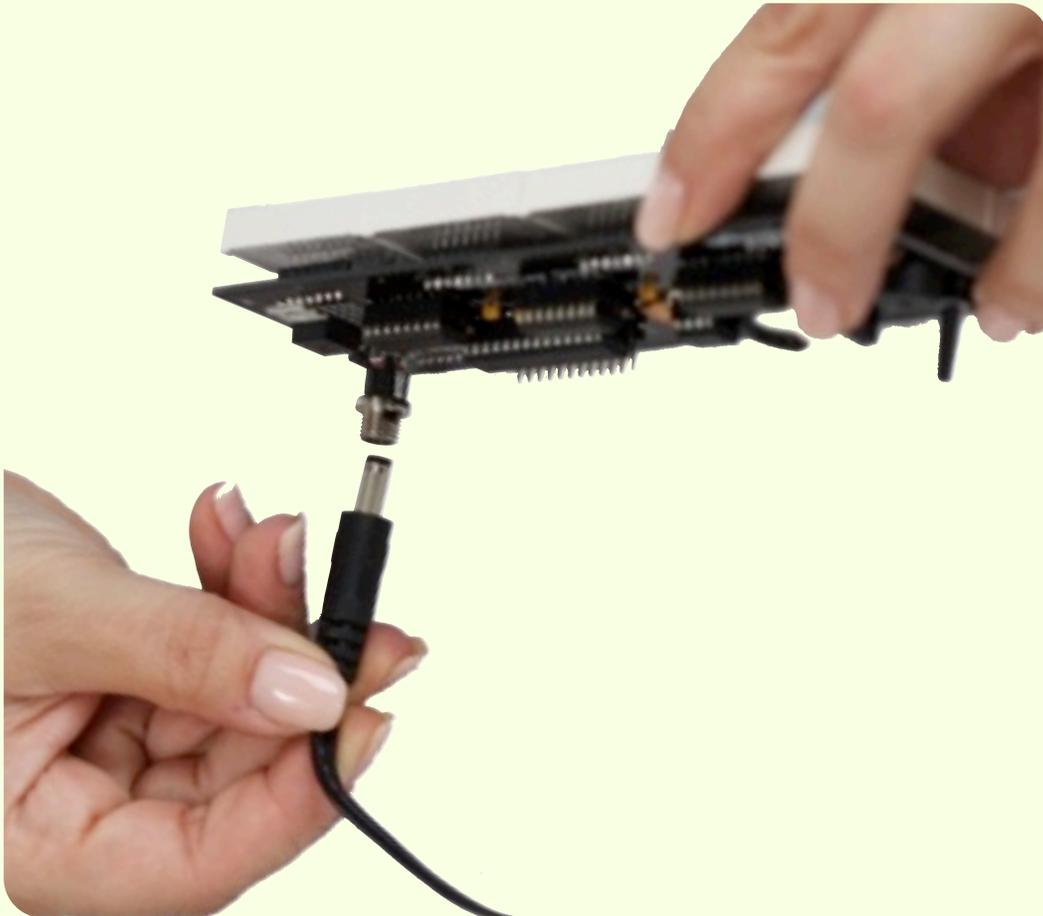
Backup battery installation.

Remove the CR2032 battery from the little bag, and insert it into the battery holder. Make sure you are installing it with the writing up. As in the photo (right):



Test Power-up

Connect a 9~12v DC power source to the DC barrel jack. You should see the LED display light up. Press mode and set, and go through the menus. Set the time, and unplug the power.



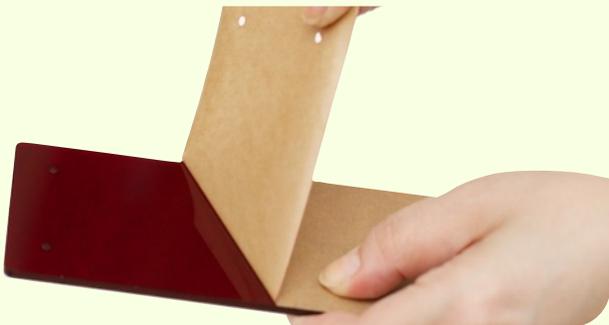
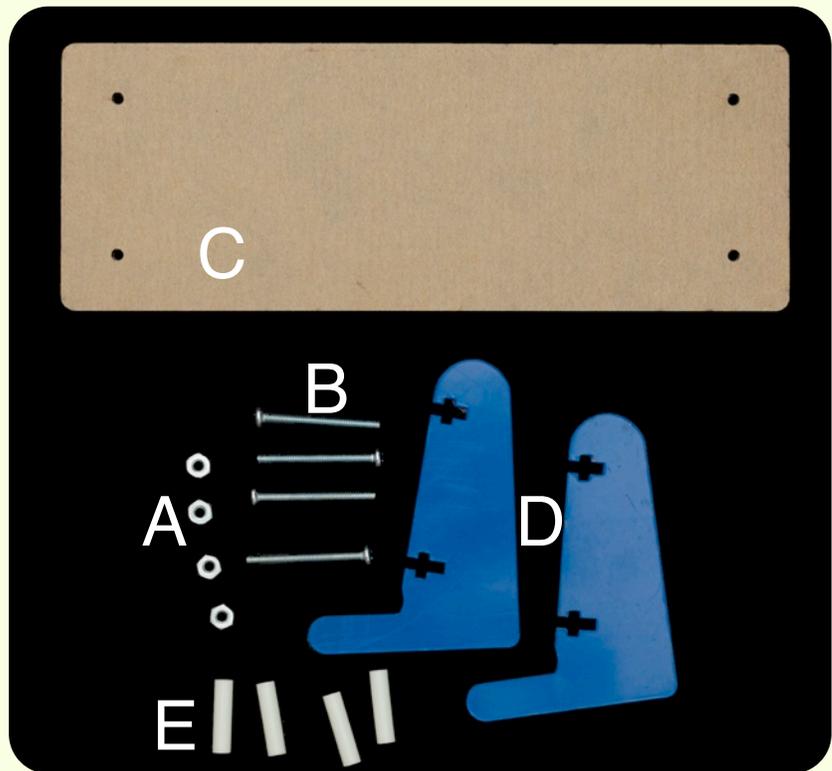
The Solder Time Desk Clock Stand Assembly:

Solder Time Desk Clock Housing Parts:

The red tinted sheet [C] is protected by peel-away paper to keep it from getting scratched during production and shipping.

The two stand pieces [D] are covered in blue plastic sheeting, which also is easily peeled away.

- [A] Four nuts
- [B] Four 1-1/4" screws
- [E] Plastic standoffs.

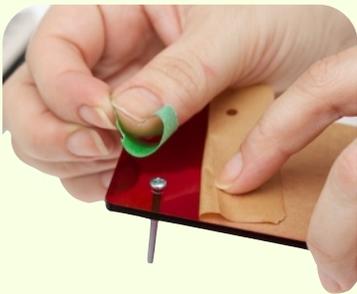


Peel either (only one) side of the plastic, and discard the paper. Leave the other side intact for the moment.

As much as possible, try not to touch the exposed surface of the red acrylic. This is to avoid dust and fingerprints.



On the other side, peel up both sides, and fold the ends down as you see in the photo here.



Place the screws through the holes on the folded up paper side of the acrylic sheet. You can hold the screws in place with a small piece of masking tape.



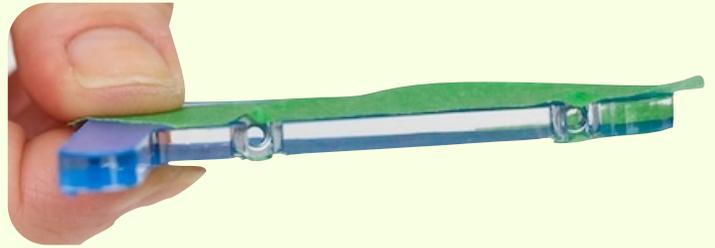
Slide the four spacers over the ends of the four screws. These hold the red acrylic panel offset in front of the display.



To make the next few steps easier, you are going to use masking tape to hold the 4-40 nuts within the clear acrylic feet. Start by placing a piece of tape over one side of each foot. Then drop a nut inside the slot on the opposite side.



Place another small piece of tape on the other side. Using some tape in these steps stops the nut from falling out while being mounted.



Place the completed Desk Clock PCB over the ends of the screws. Make sure that all of the screws go through the holes on the four corners of the PCB.



Now the Desk Clock is going to be fastened to the feet.

Hold the side foot in the orientation shown here, and check to make sure that the PCB is top side up.

(White printing on the PCB will help you figure this out.)



Line up the screw so that it is aiming towards the center of the captive nut.

Peel off the small piece of tape holding the screw down into the red acrylic sheet.

Tighten until you feel the nut catch, then go to the other screw with the same process. Once both screws are through the nuts, continue to tighten both.

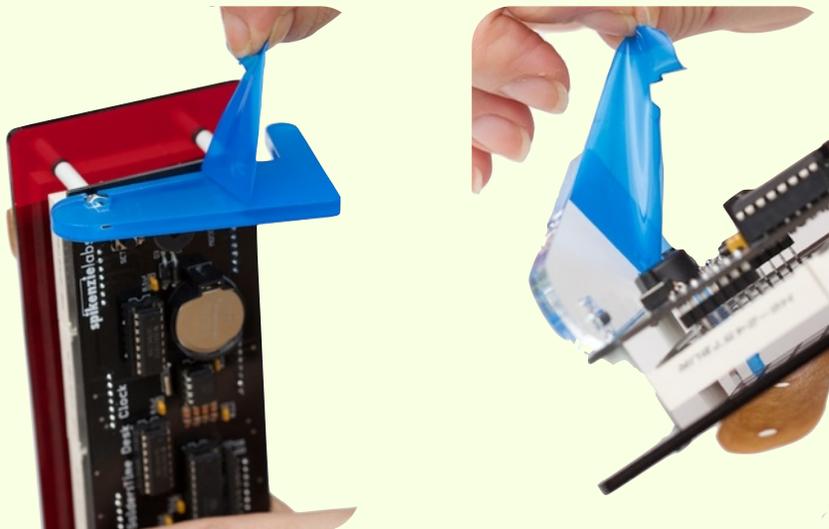


Follow the same steps for the other foot.

Helpful hint: Be careful with the way you hold the screwdriver. You don't want to have it slip off the end of the screw, and scratch the red acrylic sheet.



Peel and discard the acrylic protective coverings. Remove the masking tape, as well as the plastic protective sheeting from the feet.



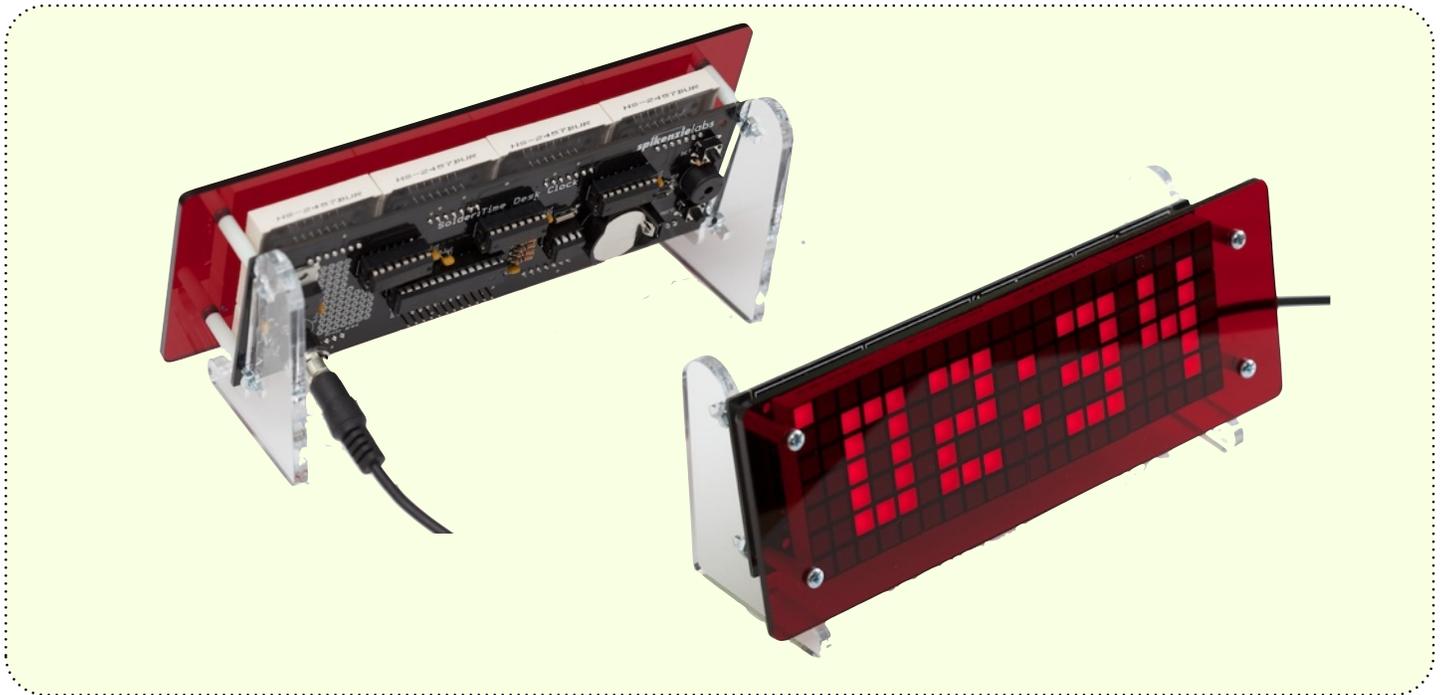
Peel away the paper protective covering from the front of the now complete assembly.

Helpful hint: Try not to touch the surface of the red sheet, and do not lay it down on this surface.

A clean, smudge and dust-free front is ideal.



Plug in and enjoy!



If you've run into a problem during the build, and you need some assistance, support@spikenzielabs.com is the place to get help.

Click here to learn about how to use the clock: [Clock instructions.](#)

For comments, other than support related inquiries, please email feedback@spikenzielabs.com, we would love to hear from you.

Follow us on twitter : @spikenzielabs

Thank you