Learn to Solder and Program Temperature Logger
PART NO. 2169459

The goal of this kit is to teach both soldering and programming in BASIC. It can be flown in weather balloons to record the temperature of the air as a function of altitude, left out over night in the garden to discover how its temperature varies over a 24 hour period, or placed inside the freezer to discover its behavior.

This solder kit is a PICAXE-08M2 based project. The PICAXE uses its READADC command to measure the temperature of its LM335 temperature sensor. Each reading is stored in data memory for later readout. A momentary push button switch starts the data downloading and temperature logging processes. An LED indicates the operating status of the project.

Time Required: Two hours depending on experience
Experience Level: Beginner

Required tools and parts:
- PICAXE-08M2 microcontroller (Available at RobotShop.com): You will NEED to program the IC and the source code shall be available on the janecho product page
- PIC Programmer
- Soldering iron and solder
- Digital Multimeter (optional)
- Wire cutters
- Small screw driver

Bill of Materials:

<table>
<thead>
<tr>
<th>Qty</th>
<th>Jameco SKU</th>
<th>Component Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>266757</td>
<td>IC,LP2950CZ-5.0, TO-92</td>
</tr>
<tr>
<td>1</td>
<td>526299</td>
<td>SOCKET,IC,8 PIN,390261-2</td>
</tr>
<tr>
<td>1</td>
<td>15781</td>
<td>Connector</td>
</tr>
<tr>
<td>1</td>
<td>1586074</td>
<td>SWITCH,PB,TACT,SPST,OFF-(ON)</td>
</tr>
<tr>
<td>1</td>
<td>253631</td>
<td>LED, GREEN, 565nm, T-1 (3mm)</td>
</tr>
<tr>
<td>1</td>
<td>158327</td>
<td>Capacitor, Radial, 22μF, 35V</td>
</tr>
<tr>
<td>1</td>
<td>120820</td>
<td>IC, LM335AZ</td>
</tr>
<tr>
<td>1</td>
<td>216452</td>
<td>Battery Snap</td>
</tr>
<tr>
<td>1</td>
<td>2158864</td>
<td>Resistor, CF, 4.7K OHM, 1/4 WATT, 5%</td>
</tr>
<tr>
<td>1</td>
<td>691180</td>
<td>Resistor, CF, 22K OHM, 1/4 WATT, 5%</td>
</tr>
<tr>
<td>1</td>
<td>2157167</td>
<td>Resistor, CF, 10K OHM, 1/4 WATT, 5%</td>
</tr>
<tr>
<td>1</td>
<td>2157159</td>
<td>Resistor, CF, 1K OHM, 1/4 WATT, 5%</td>
</tr>
<tr>
<td>1</td>
<td>2158872</td>
<td>Resistor, CF, 680 OHM, 1/4 WATT, 5%</td>
</tr>
<tr>
<td>2</td>
<td>2094320</td>
<td>SCREW, PAN HEAD, SPN2-56X3/8</td>
</tr>
<tr>
<td>2</td>
<td>38165</td>
<td>NUT, HEX, 2-56, ROHS</td>
</tr>
<tr>
<td>2</td>
<td>393272</td>
<td>Spacer, 1/4, RND, #6, AL, 1/4&quot; LMG</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>PCB</td>
</tr>
</tbody>
</table>

Step 1 - Watch the Electronic Component Power Point

The Electronic Component Power Point introduces the kit's electronic components to the new project builder. The Power Point includes speaking notes.
**Step 2 - Watch the Soldering Introduction Power Point**

This Power Point introduces the basics of soldering to the new project builder. The Power Point includes speaker notes for the teacher.

**Step 3 - Introduction to the Temperature Logger circuit**

For the Temperature Logger to operate, it must combine several electronic components into a coherent whole. The schematic shown here illustrates how the components are connected to one another. Let's take a look at this circuit, starting at the right and working our way to the left.

The schematic begins with the 9V battery snap. The positive lead of the battery snap connects to component U2, the LP2950 voltage regulator. The negative terminal of the battery forms the ground of our circuit. As the ground, it will connect to all the components but one.

Next is the voltage regulator. The regulator converts the voltage from the battery into a constant 5V. So even as the 9V battery discharges and its voltage decreases, the LP2950 maintains a constant, dependable 5V. The regulator is a low drop-out version. This means it can continue operating until the 9V battery drops down to 5.5 volts (possibly even lower). A regular voltage regulator would stop functioning when the battery voltage dropped down to 7V.

The next component is C1, the 22 uF capacitor. It acts like a temporary battery and helps the voltage regulator do its job. If the Temperature Logger suddenly needed an increase in current, the capacitor would discharge and give the circuit a squirt of current. When the demand was over, the voltage regulator would recharge the capacitor. Because of the voltage regulator and capacitor combination, the Temperature Logger maintains a more consistent 5V.

Next is R5 and the push button switch. Notice that they connect to pin #4 of the PICAXE. When the push button is not pressed, resistor R5 and PICAXE pin #4 are only connected to the 5V supply. As a result, pin #4 detects 5V, or a high signal (this is also a logic 1 in the programming). The high signal to pin #4 indicates the push button is not pressed down. The resistor ensures the current flowing into pin #4 is limited to about 1 mA (from 5V divided by 4,700 ohms). If that resistor was replaced with a wire instead, the increased amount of current flowing from pin #4 to 5V would damage the PICAXE.

When the push button is pressed, there is a dead short between PICAXE pin #4 and the circuits ground. As a result, the PICAXE detects 0V or a low signal (this is a logic 0 in the programming). The low signal to pin #4 indicates to the PICAXE that the button has been pressed.

Next is U3, the LM335 temperature sensor. It is connected to 5V by resistor R4. The circuit of R4 and U3 acts like a voltage divider. As the temperature of U3 goes up, so does its voltage (it behaves as if its resistance increases). Notice that R4 and U3 are also connected to the PICAXE pin #5. The PICAXE is monitoring the voltage created by R4 and U3 with the READADC command.

**Step 4 - Introduction**

Begin by first looking over the printed circuit board (PCB). The first thing to notice is that the PCB is a sheet of fiber glass. Fiber glass is dimensionally stable, meaning that it resists warping and changing its shape. It is also resistant to moisture, so it won’t swell in humid conditions or dry out and crack in dry conditions. These factors are important because they protect the electronic circuit that you will build on the PCB from breaking or cracking. The result is a reliable electronic circuit. Next notice that the PCB is coated with a
green plastic covering. This is called solder mask and it repels molten solder. As a result, solder will only stick to the silver rings. This reduces the chances of creating a short circuit while assembling your temperature sensor. A short circuit is an electrical circuit where it doesn't belong and when they occur, circuits fail.

Step 5 - PCB Pads

Finally, notice that each hole drilled in the PCB is surrounded by a silver ring (called a pad).

Step 6 - PCB Traces

Between pads are thin lines of light green, called traces. They electrically connect connect two or more pads together.
Step 7 - PCB Pads #2

The silver is tin plating; the ring and traces are actually made from copper. The pads are not covered in a layer of solder mask, so they remain silver whereas the traces are covered in solder mask, turning them light green. Each pad on one side of the PCB is matched with another pad on the other side of the PCB. Between the two pads is a tube of tin plating.

Step 8 - Top Silk

The top side of the PCB is marked with shapes and white lettering, called top silk. The top silk boxes and lettering indicate each component's position on the PCB. So for example, you will find a rectangle labeled with R1 to identify the location for the first resistor. Therefore, before soldering a component, you will use top silk to identify the component's proper location on the PCB.
Step 9 - Component Leads

Electronic components have wires, which are called leads. Before inserting a component lead into the PCB, you will first bend the components leads into shape. This is called forming.

Step 10 - Components Mounted Against the PCB

There are five components in this kit that are sensitive to their orientation. Therefore, when required in the directions, verify the component is properly oriented before inserting it into the PCB. Most components lay flush against the PCB.
Step 11 - Components Standing Above the PCB

Some components are designed to stand above the PCB. These components are indicated in the directions.

Step 12 - Keeping Components in Place While Soldering

Since electronic components are inserted on the top of the PCB but soldered on the bottom of the PCB, it is necessary to flip the PCB over. This will create a risk that the component will fall out. Bending the leads slightly on the bottom of the PCB will guard against this happening.
Step 13 - Taping Components While Soldering

In case of the IC socket with its short leads, it's better to apply a little masking tape to it in order to hold it in place while soldering. Now you can solder the component leads to the pads.

Step 14 - Let's start Soldering

The following assembly directions indicate which position on the PCB each component is installed. So, R1: 22k means insert the 22k resistor into the pads marked by R1. Use this image to help you properly place components.
Step 15 - Resistors

Install the five resistors. Since resistors are not polarized or sensitive to orientation, they can be inserted in any direction. Resistors are marked in four (or five in some cases) colored bands. The bands indicate the resistor’s resistance to the flow of current.

R1: 22k (red, red, orange, gold)
R2: 10k (brown, black, orange, gold)
R3: 680 (blue, gray, brown, gold)
R4: 1k (brown, black, red, gold)
R5: 4k7 (yellow, violet, red, gold)

Step 16 - Tactile Button

Solder the push button switch. This switch can be inserted right-side up or upside down. Since this is a momentary switch, it is only ON while you are holding down the button.

Button: Tactile button
Step 17 - Solder the DIP socket

Rather than solder the programmable microcontroller to the PCB, you will solder a socket instead. The socket is less likely to be damaged by soldering, so by soldering it to the PCB, you are protecting the microcontroller.

Notice that there is a notch drawn into the IC top silk. You will also notice a similar notch on the IC socket. While it does not matter electrically, it is important that the socket is installed right-side up.

U1: DIP socket
Align the notch in the IC DIP socket to the notch in the top silk of the PCB.

Step 18 - LED anode and cathode

D1: LED
The LED is polarized, so if it is soldered in backwards, it won't work. Now this is one of the few errors that will still let the temperature work. However, the circuit loses functionality. In most cases, the length of one lead of the LED is longer than the length of the other lead of the LED. This lead marks the anode of the LED. Also, look carefully at the body of the LED. You will notice one side has a side shaved flat. The flat side marks the cathode of the LED.
Step 19 - Solder LED

Now solder the LED.

Step 20 - Solder the LM335

U3: LM335
The top silk indicates the proper orientation of the LM33 temperature sensor.
Step 21 - Solder the Capacitor

C1: 22 uF capacitor
Electrolytic capacitors are polarized, one lead is positive and the other is negative. In many instances, the long lead is positive and the short lead is negative. However, the sure way to tell the capacitor's positive for negative lead is to look at the side of the capacitor. One side has a band containing one or more negative signs (-). Place the positive lead of the electrolytic capacitor into the pad marked with the positive (plus) sign, +.

Step 22 - Solder the voltage regulator

U2: LP2950
The top silk indicates the proper orientation of the LP2950 voltage regulator

Step 23 - Solder DB-9

DB-9: Female DB-9 connector
First, bolt the connector to the PCB, then solder the leads of the DB-9 to the PCB.
Step 24 - Bolted DB-9

Bolt the DB-9 connector before soldering it. Place a nylon spacer between the PCB and the DB-9 wings.

Step 25 - Battery Snap

9V battery snap
In order to prevent the 9V battery snap from breaking off the PCB, it contains two strain relief holes. The leads of the batter snap are first threaded through these holes from the underside of the PCB. The leads are then inserted into pads.

The red lead goes to the pad marked +V and the black lead goes to the pad marked G.
Step 26 - Solder Check

Check solders on bottom of PCB
Verify the soldered connections do not overflow their pads. When one solder touches a second that its not suppose to be connected to, a short circuit is created. Short circuits are potentially damaging to an electronic circuit. The only time that a short circuit is safe is when there is a trace connecting the two pads together.

Step 27 - DMM Check

Optional, use a multimeter to check your work
First, set the multimeter to continuity. If your meter doesn't have a continuity setting, then set it to measure resistance. If you now tap the meter's test leads together, the meter will either ring (in continuity setting) or try to display zero resistance (if in resistance setting). This indicates there is a short circuit between the test leads, as we expect.

Now touch the meter's two test leads to the contacts on the battery snap. We do not want a short here. So make sure the meter is either not ringing or that its trying to indicate a very high resistance. If a short is indicated, then look over your soldered connections for solder blobs touching each other.

Now that it is safe to plug the 9V battery in, we want to check the operation of the circuit. Particularly in its ability to produce 5V from a 9V battery.

Set a multimeter to the 20V scale (since the circuit creates 5V from the 9V battery, there are no voltages above 20V or below 2V). Touch the meter’s ground lead (usually black) to pin #8 on the IC socket and the meter’s positive lead (usually red) to pin #1. The multimeter should read between 4.75V and 5.25V.

Step 28 - Insert IC

You may need to squeeze the IC’s leads together slightly. Place the IC in its socket, with each IC lead over its metal connector in the IC socket. Now press down firmly and evenly on the IC. It should snap into the socket.