**How to Assemble the Circuit:**

1. **Assemble the Circuit on a Breadboard:**
   * Follow the schematic to assemble the circuit on a breadboard, ensuring all components are connected correctly. This will allow you to test the circuit and ensure everything works before moving to the final PCB assembly.
   * Double-check all connections and verify that the components are properly placed on the breadboard.
2. **Flash the D1 Mini:**
   * Flash the D1 Mini using the Arduino IDE, following the instructions from the video. <https://youtu.be/qqoym5jryEc>
   * There are two variants for the firmware: one with debug messages and one without. **Recommendation:** I suggest using the version **with debug messages**. This will help you troubleshoot and fix any potential issues by reviewing the logs if bugs are discovered.
3. **Wiring the 3-Position Slider Switch:**

The34-position slider switch has four pins, and the wiring is as follows:

1. **Pin 1** is connected to **ground (GND)**. (This is the **ground pin**.)
2. **Pin 2** is connected to a **digital input pin** on the D1 Mini (or Arduino).
3. **Pin 3** is connected to a **second digital input pin** on the D1 Mini (or Arduino).
4. **Pin 4** remains **unconnected** and corresponds to the **"safe mode"** when the switch is in the third position.
5. **Wiring the Trigger Switch:**
   * The **momentary trigger switch** should have one pin connected to **ground** (GND).
   * The other pin of the switch should be connected to a digital **input pin** on the D1 Mini. This will send a signal to the D1 Mini when the trigger is pressed.
6. **Pin Setup:**
   * Use the following pin assignments for the D1 Mini:
   * **Trigger Pin (D1):** Connect the momentary trigger switch to this pin. It will read input when the switch is pressed.
   * **Selector Switch Pins (D6 & D2):** The **3-position slider switch** is wired to these pins. Position 1 goes to **D6** and position 2 goes to **D2**. Both share the common ground.
   * **Buzzer Pin (D7):** The buzzer is controlled by this pin.
   * **Solenoid Pin (D8):** This pin controls the solenoid (used for triggering or controlling mechanisms).
   * **Tracer Pin (D4):** This pin controls the tracer (LED or light). If you use two LEDs, wire them in parallel with **47Ω resistors** in series.
7. **Move to PCB and Solder the Components:**
   * Once everything works on the breadboard, transfer the components to the **PCB**.
   * Solder the components to the PCB according to the schematic. Be sure to check the component values and placements before soldering.
8. **Check LiPo Battery Polarity:**
   * Double-check the **LiPo battery input** connections to ensure proper polarity. The **positive (+)** terminal should go to the input of the **LM7805**, and the **negative (-)** terminal should go to ground (GND).
9. **Tracer LED Setup:**
   * For the tracer, use **2 LEDs in parallel** to ensure proper brightness and current distribution.
   * Each LED should have a **47Ω resistor** in series to limit the current. This helps protect the LEDs from excessive current.

### ****LM7805 Capacitors (Required for Stability):****

* Adding capacitors improves the stability of the LM7805 voltage regulator.
  + **10µF Electrolytic Capacitor** on the input: Reduces voltage fluctuations and smooths input power.
  + **0.1µF Ceramic Capacitor** on the output: Filters high-frequency noise for a cleaner 5V output.

### ****Polarity Considerations:****

* **10µF Electrolytic Capacitor**:
  + Positive leg to **input pin** (Pin 1) of LM7805.
  + Negative leg to **ground (GND)**.
* **0.1µF Ceramic Capacitor**: Non-polarized, connect either way between **output pin** (Pin 3) and **ground (GND)**.

Proper capacitor placement ensures better voltage regulation and noise reduction.

How the tracer is working: <https://www.youtube.com/shorts/O8T2DLfdGJM>