

Notebook Entry 2

Date: February 15th

Objective: Begin schematic design for PCB1, select core components for control and power subsystems.

Record of Work:

Today we began drafting the schematic for PCB1, the board that would house both the control and power systems. We spent a significant amount of time evaluating different microcontrollers before settling on the ESP32-WROOM-32D. This chip checked all the boxes—it has integrated Bluetooth and RTC, plenty of GPIOs, and saves valuable board space by eliminating the need for additional modules like the HC-05 and DS3231.

Using KiCad, we laid out the initial schematic, focusing on three areas: UART communication (via CP2102N), voltage regulation (from 12V to 5V and then 3.3V), and visual pairing indicators using LEDs. We paid close attention to the datasheets for each component, particularly the ESP32, to ensure our decoupling capacitors (0.1uF and 22uF) were placed as close as possible to power pins to stabilize voltage.

We also began validating our design against the ESP32 reference layout to ensure we didn't miss any important strapping pins or pull-ups. Figure 2 shows the current schematic section for the ESP32, and we plan to iterate on this before sending it out for fabrication.

By the end of the day, we felt cautiously optimistic. There's still a long road ahead, but seeing the schematic come together was motivating. This wasn't just a concept anymore—it was becoming real.