

Wabash Capstone Project Notes 09

By: John Danison

What Did I Say I Would Do This Week?



Find electrical schematic software & get working locally.



Begin writing firmware outline & find proper libraries.

What I Have Done Over The Past Several Weeks



Wrote test firmware modules for each sensor we are using. Created a simple working prototype. See slide 4.



Found electrical schematic software in the forms of Fritzing and Altium Designer. Both options are installed and working. See slide 5.

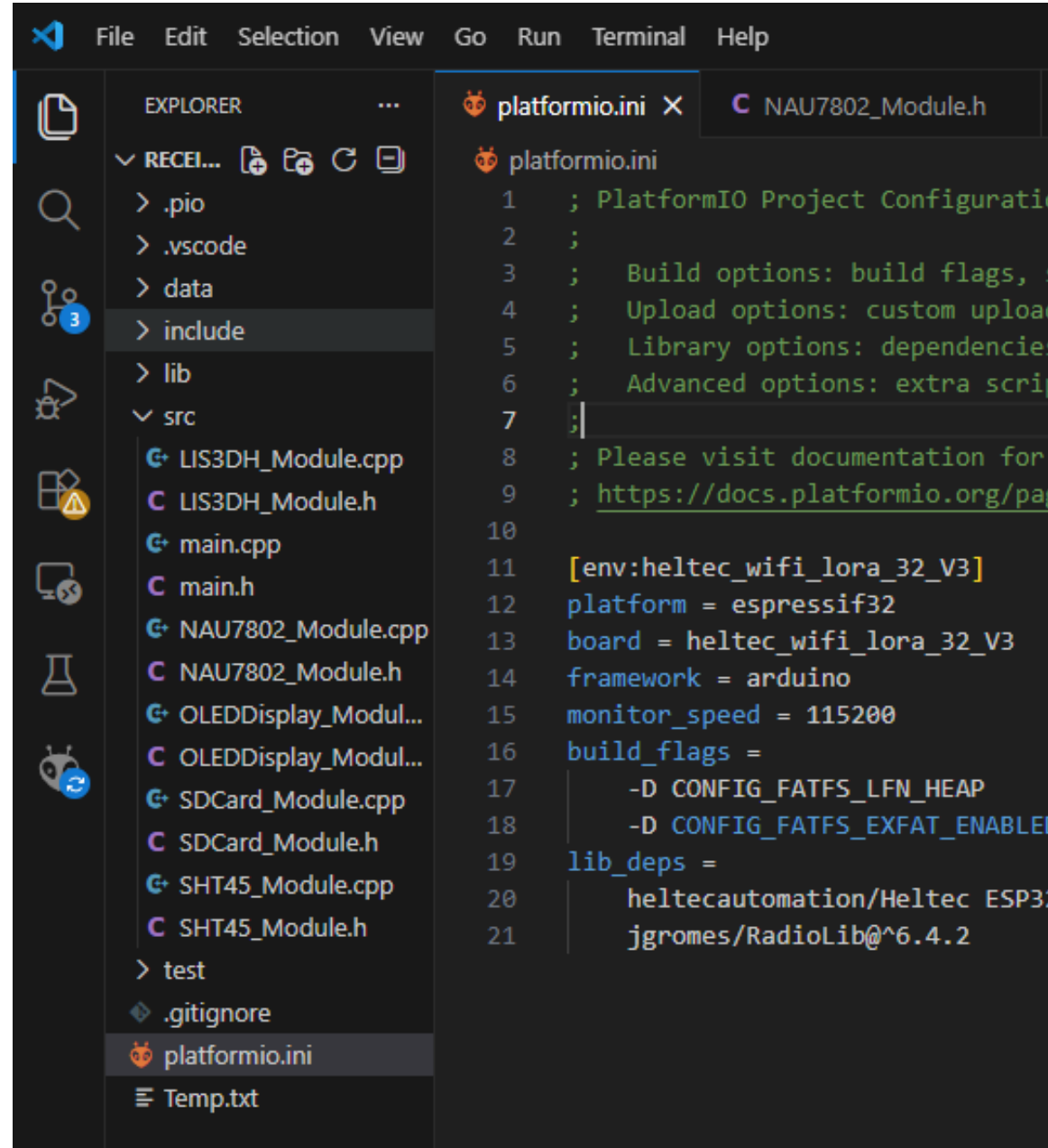


Created more electrical documentation for new circuits. Also did calculations for battery selection and circuit power. See slide 7.



Max found strain gauges, so I created the circuitry and code needed to test these for the first time. See slide 11.

Breakdown of Code Modules



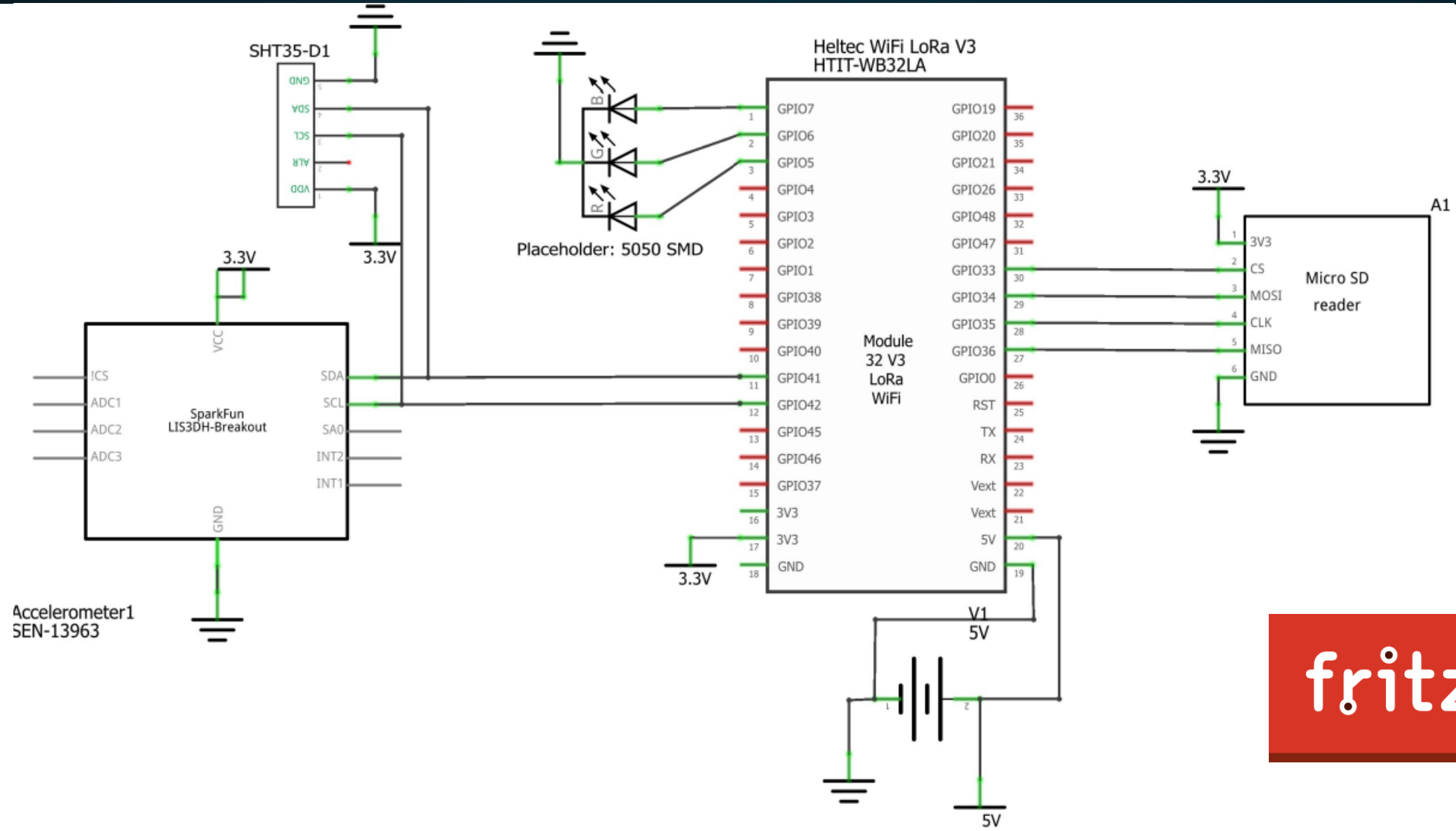
The screenshot displays the Visual Studio Code interface for a PlatformIO project. The Explorer sidebar on the left shows the project structure, including folders for .pio, .vscode, data, include, lib, and src. The src folder is expanded, revealing source files for LIS3DH, main, NAU7802, OLED Display, SD Card, and SHT45 modules, each with a .cpp and .h file. The platformio.ini file is selected and open in the editor, showing project configuration details such as the board (heltec_wifi_lora_32_V3), framework (arduino), and various build flags and dependencies.

```
File Edit Selection View Go Run Terminal Help

EXPLORER
  RECEI...
    > .pio
    > .vscode
    > data
    > include
    > lib
    > src
      LIS3DH_Module.cpp
      LIS3DH_Module.h
      main.cpp
      main.h
      NAU7802_Module.cpp
      NAU7802_Module.h
      OLEDDisplay_Modul...
      OLEDDisplay_Modul...
      SDCard_Module.cpp
      SDCard_Module.h
      SHT45_Module.cpp
      SHT45_Module.h
    > test
    .gitignore
    platformio.ini
    Temp.txt

platformio.ini
  1 ; PlatformIO Project Configuration
  2 ;
  3 ; Build options: build flags,
  4 ; Upload options: custom upload
  5 ; Library options: dependencies
  6 ; Advanced options: extra scrip
  7 ;
  8 ; Please visit documentation for
  9 ; https://docs.platformio.org/pa
  10
  11 [env:heltec_wifi_lora_32_V3]
  12 platform = espressif32
  13 board = heltec_wifi_lora_32_V3
  14 framework = arduino
  15 monitor_speed = 115200
  16 build_flags =
  17     -D CONFIG_FATFS_LFN_HEAP
  18     -D CONFIG_FATFS_EXFAT_ENABLE
  19 lib_deps =
  20     heltecautomation/Heltec ESP32
  21     jgromes/RadioLib@^6.4.2
```

Electrical Software 1 – Fritzing



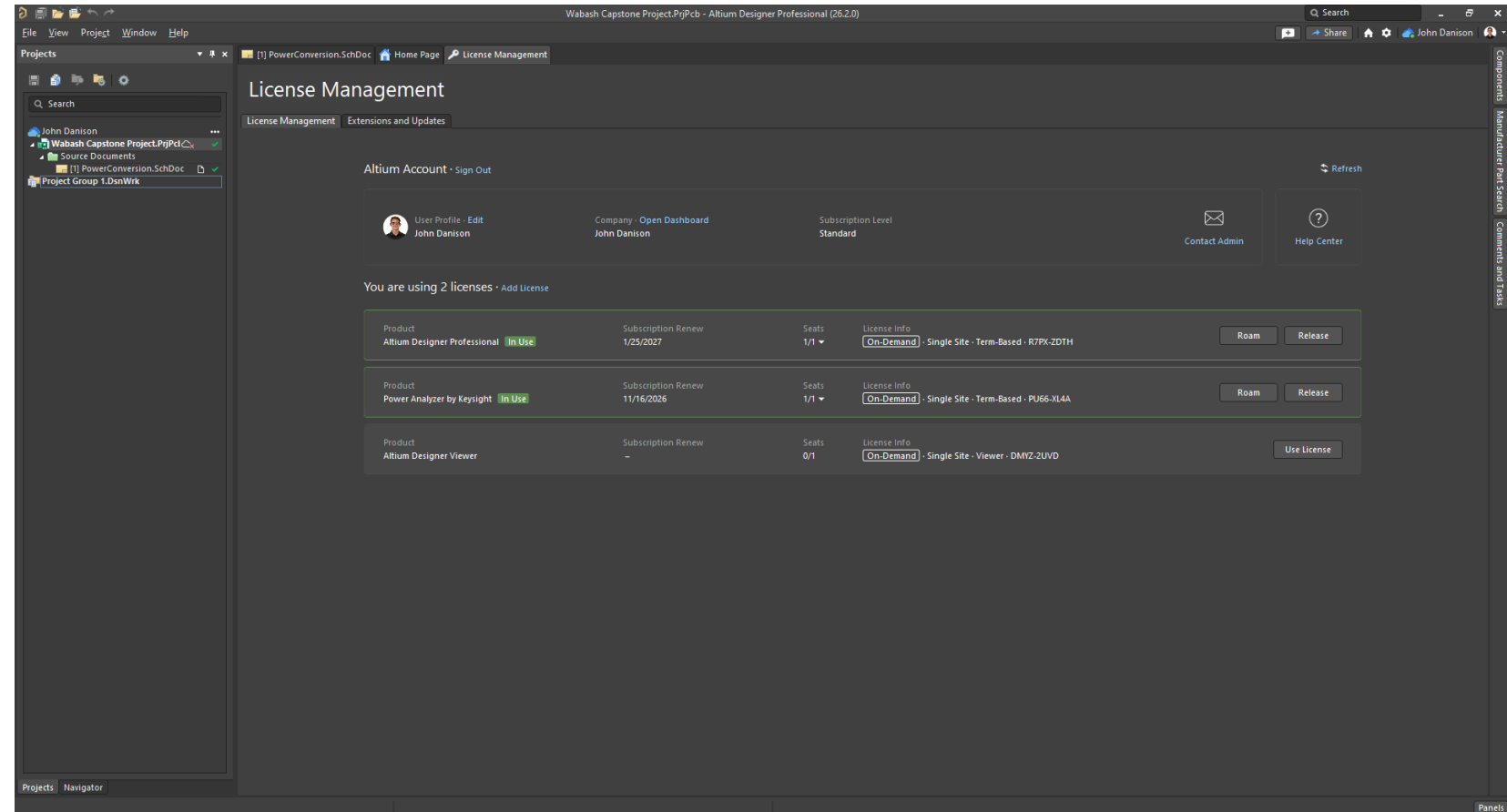
Electrical Software 2 – Altium Designer



Going forward, Altium Designer will be used.

Fritzing is good for quick simple drawings, Altium is PHENOMINAL at schematic making, PCB layouts, and more.

Nothing has been done in Altium so far, I just got it working today.



Summary of Current Breakdown & Battery Calculations

Current Breakdown

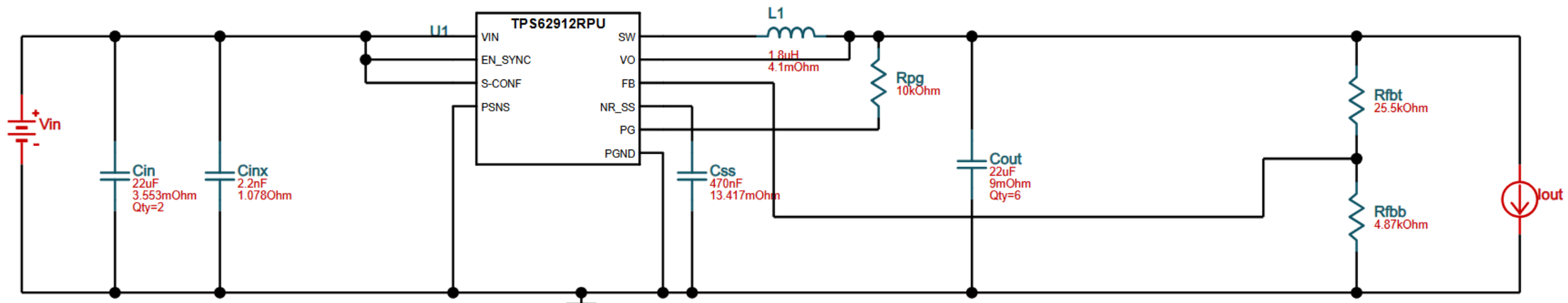
Component	Current (mA)
ESP32	20
LoRa Radio (idle)	46
LIS3DH	0.011
SHT35	0.1
microSD	2
Total	68.211
+10% Overhead	75.0321

Battery Calculations

Calculation Step	Formula	Result
mAh @ 5V	$75.0321 \text{ mA} \times 6 \text{ months} \times 30 \text{ days} \times 24 \text{ hr}$	324.139 Ah
Wh @ 5V	$324.139 \text{ Ah} \times 5 \text{ V}$	1620.699 Wh
Wh w/ battery loss	$1620.699 \text{ Wh} \div 0.90$	1800.770 Wh
Ah from 12V battery	$1800.770 \text{ Wh} \div 12.8 \text{ V}$	140.685 Ah

Sample Designed Power Converter

Sample 12V : 5V DC Buck Converter

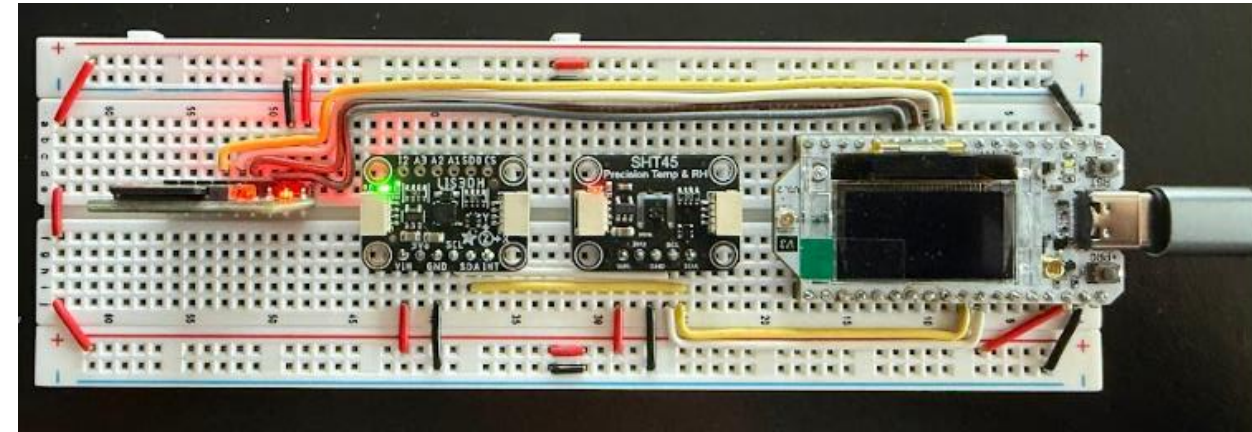
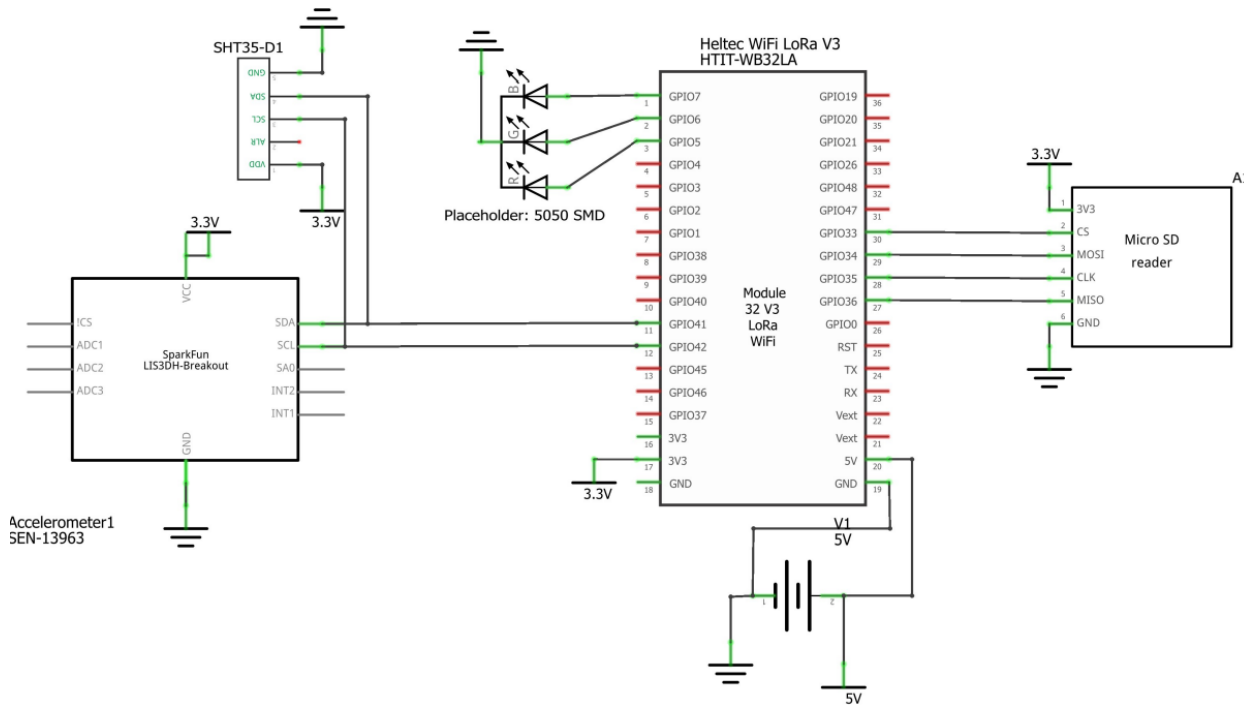


Efficiency: 96.6% BOM Cost: \$3.07 Footprint: 169 mm² BOM Count: 15 Topology: Buck

Frequency: 2.2 MHz IC Cost: \$0.84 | 1ku



Comparison of Circuit Software & Hardware Layouts



Comparison of Software Data Acquisition

Data Sample Expectations

Parameter	Value / Calculation	Result
Sensors	3-axis accel + 1 strain (high-rate) + Temp & RH (1 sample)	—
Sampling Rate (Accel/Strain)	2,000 samples/sec	—
Data Format	16-bit (2 bytes/sample)	—
Data Rate	$2,000 \times 4 \times 2$	16 kB/s
Burst Duration	—	2 s
Burst Size (Accel + Strain)	$16 \text{ kB/s} \times 2 \text{ s}$	≈ 32 kB
Temp + RH Samples	2 sensors × 2 bytes each	+ 4 B
Total per Burst	32 kB + 4 B	≈ 32 kB
128 GB microSD Capacity	≈ 137,000 MB	—
Bursts Storable	$137,000 \text{ MB} \div 0.031 \text{ MB}$	≈ 4.3 million bursts

Unoptimized Data Sample Results

The screenshot displays a Windows File Explorer window showing the contents of the 'events' folder on the 128 GB (E:) drive. It lists three files: event 1.txt, event 2.txt, and event 3.txt, all of which are 1 KB in size. Overlaid on this is a Notepad++ window editing 'event 3.txt'. The file content is as follows:

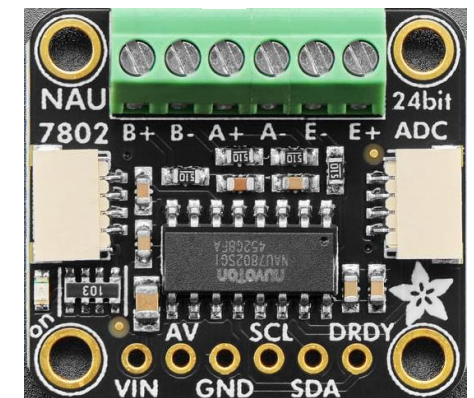
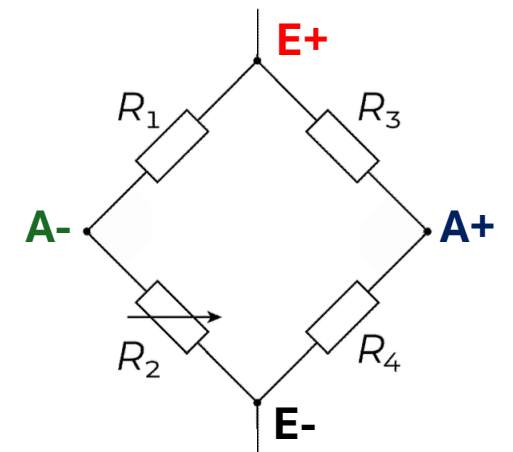
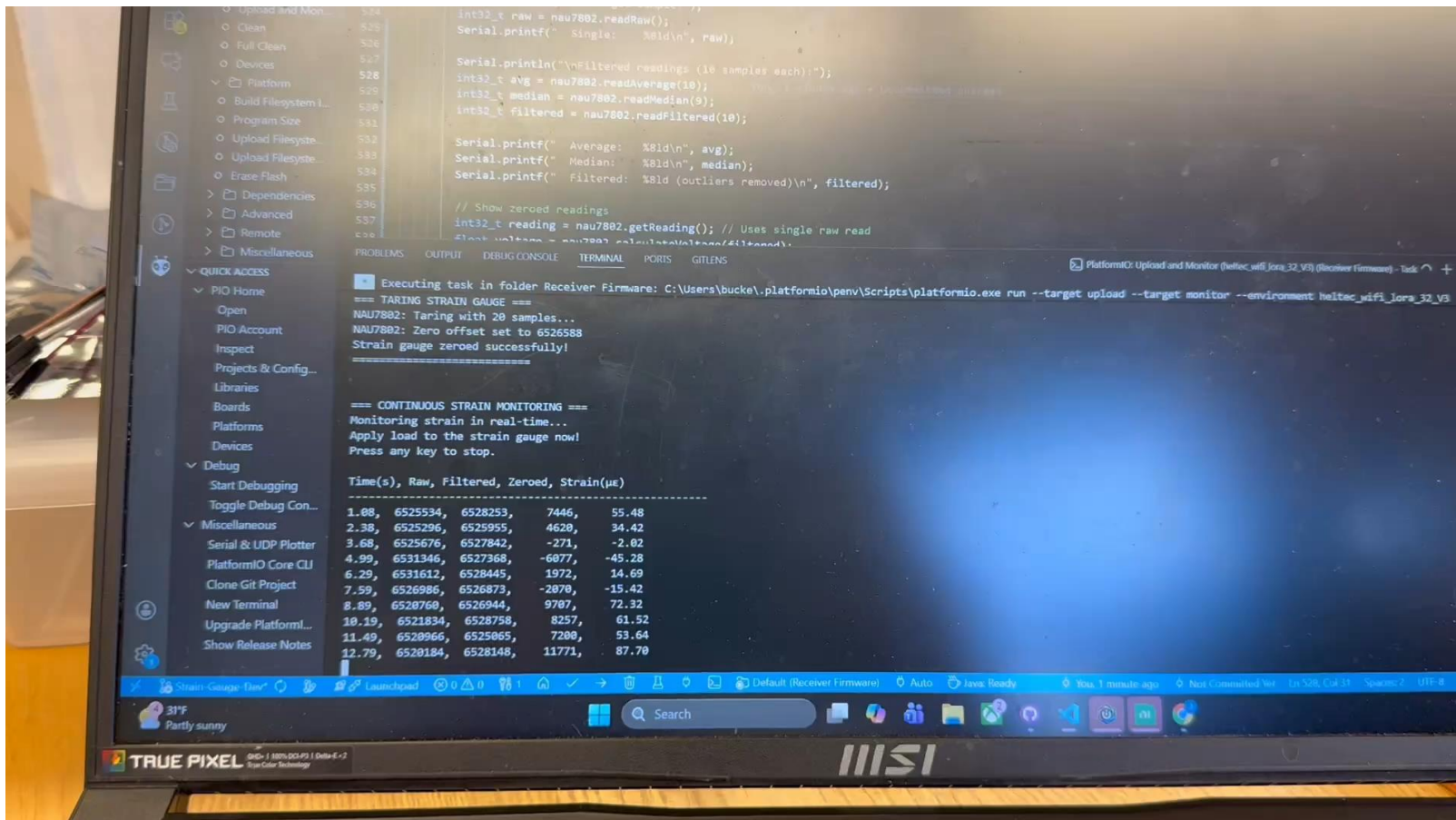
```

1  === EVENT 3 ===
2  Timestamp: Time not set
3  Temperature: 24.64 C
4  Humidity: 23.93 %
5
6  Accelerometer Samples (20):
7  Sample, X(g), Y(g), Z(g)
8  1, 1.864, -2.034, 1.334
9  2, 1.952, -0.415, 1.257
10 3, 1.996, 0.813, 1.649
11 4, 2.018, 1.427, 1.845
12 5, 2.029, 1.734, 1.898
13 6, 2.035, 1.887, 0.912
14 7, 2.038, 1.964, 0.656
15 8, 2.039, 2.002, 0.229
16 9, 2.040, 2.021, 0.477
17 10, 2.040, 2.031, 1.259
18 11, 2.040, 0.088, 1.650
19 12, 2.040, 0.088, 1.650
20 13, 2.033, -0.976, 1.845
21 14, 1.223, -1.508, 1.943
22 15, -0.070, -1.774, 1.667
23 16, -0.733, -1.907, 0.834
24 17, -0.213, -1.973, -0.100
25 18, 0.914, -2.006, -1.070
26 19, 1.477, -1.620, -1.555
27 20, 1.759, 0.210, -1.797
28

```

The Notepad++ status bar at the bottom indicates the file is 'Normal text' with a length of 622 characters, 29 lines, and is using Unix (LF) line endings and UTF-8 encoding.

New Strain Gauge Circuitry & Demo



Did I Do What I Say I Would Do?

I believe that I covered everything that I said I would do. I created the firmware and circuitry required to get all sensors working.

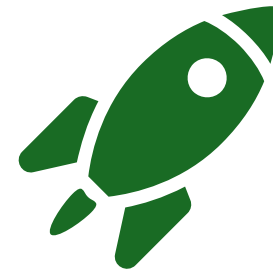
I showed up to every lecture, lab, and meeting and contributed my knowledge with everyone.

Plans For Upcoming Weeks



High Level

Start to refine system sensors & configuration.
Learn how to use Altium Designer. Will follow tutorial videos.
Continue to show up to class.



By EOW:

Continue to refine the noise readings in strain gauge circuit. Figure out a reliable solution.
Begin creating schematics in Altium Designer.
Need to start PCB design ASAP.