John Stinger

ECEN 499

Executive Summary

Powder Scale and Dispenser

Overview

This semester of the ECEN 499 senior project course, I had the opportunity to be a part of the powder scale and dispenser team working with Creedmoor sports. The scope and intention of this project was to design a system with a fully integrated touchscreen GUI that could communicate with a digital scale to provide feedback so that accurate measurements of gunpowder could be made automatically.

The whole project was subdivided into three primary sections: hardware, frontend software, and backend software. The frontend software included the designing of the touch screen display interface. The backend software covered the operating system and intersystem communications. Hardware was designing a PCB that could facilitate everything.

I was on the hardware team.

Hardware

The first step I took in helping to design the hardware was to decide which type of processor and architecture to use for our system. After several hours research, the initial options leaned towards either using a Raspberry Pi-like system or to create a BeagleBone clone. The Raspberry Pi system wasn’t chosen due to its general unavailability and since the initial design was on a BeagleBone platform, as a team we decided to take the route of a BeagleBone clone. This would allow for a simple continuation rather than a complete redesign. Plus, the BeagleBone platform was more than robust enough for our purposes.

It was at about this point that we received a package from Creedmoor including a prototype system with a powder trickler, controller, and a scale. We then worked to get the prototype working which helped quite a bit with understanding the operation and mode flow.

The accelerometer was next. Built into the base of the trickler, there was a pre-installed 3-axis accelerometer which would sense the angle of the trickler to provide feedback as the height of one of the legs was being adjusted by a stepper motor. I analyzed this component and determined that that sensor would suffice.

As this was being determined, progress started happening on the designing of the circuit board.

One of the first questions we asked about the board itself was determining what to do with our power supply. The original design operated at 24 volts. Or at least that's what the desired voltage for the stepper motors was to be. Dealing with this kind of voltage, we thought it might be a better idea to divide our PCB in two. One board being the control and processing and the other being a power supply circuit that would distribute power. After talking with Jared, our representative and point of contact from Creedmoor, he expressed how much emphasis they wanted to put on making this a very compact product so that they can reduce the footprint on a bench as much as possible.

Now that the board design was underway, it quickly became important that we decide what motor drivers we would want to use with our design. Included with the prototype was several EasyDriver Stepper Motor Controller A3967 chips. These are what were being used up to this point, but there were some issues (that we were able to recreate) with the driver board getting a little too much voltage and subsequently blowing up.

While doing the research for a more suitable motor driver, I made two important decisions. First, I noticed that the operating voltage for the selected stepper motors that we had only required 5 volts. The original design of 24 volts was for the purpose of increasing torque. We corrected this issue, because as the resistance on the motor stays the same, and voltage increases, the current decreases proportionately. In a stepper motor it is the total current through the motor that drives overall torque. So, we decided to go with a 5 Volt system after concurring with Creedmoor. The second decision was to try out the Trinamic TM2208 Stepper motor driver. This would allow for maximum voltage versatility due to its wide range of 4.75V - 36V. The Trinamic driver also had the added benefit of having a cleaner output voltage signal which in turn produces less vibration in a motor and in theory should be quieter in operation.

We were now about halfway through the semester. An initial estimate of the total cost and raw products became prudent. The total cost of all the parts including the touch screen, PCB fabrication and population, and small component purchases came to about $120-140. With that being done, I continued to help Colby in designing the board.

We now had the primary bulk of the board laid out, and time had been spent straightening out silkscreen component labels. The next step was to design the layout of the motor drivers on the board. This was not trivial due to the uncertain documentation for the motor driver IC, but we were able to determine the correct resistor values and pin outs associated with the driver.

With the completion of the motor driver layouts, and some last-minute rerouting of a few differential pairs, we were done with the PCB design. A 3D model was rendered of the work we had so far so that it may be ported into SolidWorks, and we then shifted to acquisition mode. This is where we started to run into some hardcore issues.

The first major problem that we encountered was when we tried to create a bill of materials and open it on the JLCPCB website. We had about 70 or so components that were not selected. We Simply put our heads down, and started powering through the list to select the components manually, but we very quickly realized that many of the components that we needed were still being affected by the ongoing chip shortage.

This was not an issue we were going to be able to work around. It was at this point we had to sit down with our instructor and our Creedmoor representative and confront reality together. We decided that the final design of the circuit board just wasn't going to happen. That at this point we were going to draft up what we had, and get the project ready for the next group to take it over.

I personally started helping Austin with the set up for some demo hardware. The GUI guys had made great progress on their work, so Austin and myself set up a BeagleBone board to be able to have a working demo of the software.

Team Effectiveness

This section was a little hard for me. My individual role was rather complex. As a hardware person, I was much less experienced then Colby. The only PCB experience that I had was from the very basic projects we did in 250 and being currently enrolled in 430, still I hadn't received much extra experience. So, my role became mainly research and support. By trying to apply the principles learned in ECEN 430, I gave technical advice to Colby throughout the entire board layout process while he taught me how to design. Researching components for others became a regular job as well as picking up the odd jobs such as designing the poster.

Future Work

Again, I cannot stress hard enough the necessity of looking up availabilities on parts. In the future, spend a great deal of time at the very beginning researching exactly what it is you need to do, what hardware are you going to need to accomplish it, what components do we have available to order, and then to finalize the direction of the project with that information. As I'm sure Colby will also recommend, going the route of a four-layer board would help immensely with decluttering the board. So, review the viability of the BeagleBone platform and also look into other options such as the Raspberry Pi platform to compare and contrast.

Conclusion

Things that I personally learned myself would include the importance of getting crystal clear expectations at the beginning of a project. This would have saved us at least a week of work. I also learned a great deal about stepper motors, and how the drivers work. This is a specific subject that I would love to delve into much deeper and even design my own stepper motor drivers someday. Obviously, the biggest problem we had was due to logistical issues. If we had started by looking for components that were available, and design our system from the ground up with these available components, we probably would have been able to have a completely mocked up prototype.

I would also like to express my gratitude for working with this project. I had a great team. Everybody contributed a great deal to the project, everybody was positive, and everybody was willing to help others when they needed it.

|  |  |  |
| --- | --- | --- |
| Week | Hours | Major Tasks |
| 1 | 12 | Began outlining the entire project  Defining the mission statement  Dividing responsibilities |
| 2 | 10 | Researching processors and architectures |
| 3 | 6 | Checked the viability of the current accelerometer  Started drafting the circuit board |
| 4 | 11 | Board design  Component selection |
| 5 | 12 | Work the board schematic and layout |
| 6 | 11 | Motor driver research  Board development |
| 7 | 10 | Motor driver research  Acquiring a power adapter  Board development |
| 8 | 11 | Board development  Pricing |
| 9 | 10 | Board development  Driver layout |
| 10 | 10 | Board development  3D modeling of board  Ordering the board |
| 11 | 12 | Documentation  Component definition on JLCPCB |
| 12 | 13 | Documentation  Setting up the demo hardware |
| 13 |  | Poster  Documentation  Setting up the demo hardware  Presenting |