Execute summary

Colby Hatton

# Technical Contributions

1. I found, sourced, and improved the schematic for the hardware on our project. This involved finding the original, open-hardware, project, converting it to a Ki-Cad project, and altering the connections and components to fit our needs. These changes required removing unnecessary components of the board such as permanent memory, header pins, and wireless components. It also involved adding the motor’s power system, and all associated components.
2. I laid out the full board and all its associated components. This included sizing the board to our exact dimensions, adding the GND and CGND planes, dictating trace widths and via radii, and connecting every component. Careful consideration was made to follow good routing practices, based on the 7 Habits of Successful Board Designs by Eric Bogatin. Decoupling Capacitors were placed near power sources, several links were made to provide wide paths for the ground plane, and others were followed. I ensured the data lines and power lines were well separated, so noise from the motors would be separated from the calculations and other processes running. Everything was routed on a two-layer board.

# Team Contributions

1. My primary contribution was leading the hardware group in our project. I worked with John and Austin to ensure that our added hardware would preform to specification, specifically, we worked on the motor drivers. I always had a positive attitude, and promptly responded to any questions, requests, or issues that occurred.

# Final Report Contributions

1. I created the Power Point presentation that we used to present our findings and conclusions. I also led a major section of the presentation itself.
2. I verified the information and accuracy of each document relating to the hardware, ensure all the necessary information was included.
3. I wrote out major sections of the documentation, especially for the hardware. In particular, I wrote out the various steps I took to create the schematic and layout. I also wrote out our issues section, Bill of Materials, and future recommendations.

# Future work on project

1. The processor is totally out of stock and won’t be back in until around October or November. This, as well as the totally missing graphics driver might be grounds to change the base of the board from BeagleBone to a different platform entirely, such as Raspberry Pi, or perhaps an extremely powerful Arduino, though the latter would require a fundamental rewrite of the code.
2. The board, while effective, would certainly be expensive. Various components could probably be replaced with minimal functional differences and be much less expensive. In particular, the processor and video controller are quite spendy.
3. Once a physical board is actually created, there would need to be a lot of testing to ensure that the board works properly before beginning a manufacturing run.

# Week Breakdown

|  |  |  |
| --- | --- | --- |
| Week | Hours | Major Tasks |
| 1 | 8 | Met Team, initial preparations, big picture roadmap |
| 2 | 10 | Decided on Specific Development board, defined goals for project |
| 3 | 13 | Determined CAD software, learned the basics of KiCad, began sourcing components and symbols |
| 4 | 15 | Built sections of Schematic, researched parts and supplies for project |
| 5 | 15 | Continued Schematic, set plans for layout, further parts research |
| 6 | 13 | Finished Schematic, Began layout |
| 7 | 16 | Continued Layout |
| 8 | 16 | Continued Layout |
| 9 | 14 | Finished Layout |
| 10 | 13 | Testing for Manufacturing |
| 11 | 13 | Created Part List for JLCPCB, Began Documentation |
| 12 | 13 | Continued Documentation, Found Alternate possible components for future semesters |
| 13 | 14 | Finished Documentation, Presented Progress, Passed on Project |
| Total | 173 |  |