ECE 388 Final Report for the Catch the Light Project

Team Life-Support

Members: Alec Pimentel, Nickolas LaForce, Jay Votta, Dylan Souza

Project Background

The project that our team received for our ECE 388 junior design project was called “Catch the Light”. The project’s inspiration was drawn from the “Stop the Light” game created by Justin Meninno, a former ECE student at UMASS Dartmouth. The project consisted of two different Phases. For Phase 1, a PCB was to be designed using only 7400 series discrete logic, just like that of Justin’s design. Phase 2 required us to design a more advanced handheld version of the game using the Atmega328PB microcontroller. After completing the project, it can be said that the team gained a substantial amount of knowledge and valuable experience throughout the semester. In addition, this class taught students about PCB designing by offering hands on experience with real world applications.

New Knowledge Gained

**Data Sheets**

Perhaps one of the most critical resources that had to be used for this project was datasheets. Prior to this project, the only class where students had to use a datasheet quite often was ECE 263. However, to successfully complete this project, data sheets for many components had to be reviewed to understand how the components operate, in addition to how they were to be interfaced into the project. One datasheet that was reviewed frequently was for the **Atmega328PB**. Although all members in the team had experience working with the microcontroller, certain aspects of the project such as the capacitive touch button and sleep mode required extensive research in the datasheet. The datasheet for the Atmega328PB offered a lot of helpful information when it came to interfacing the Atmega with the Q-Touch button on the project. It was learned that Atmel Start could be used to access the Q-Touch library that was specifically written for coding capacitive touch buttons.

Besides the datasheet for the Atmega, many different datasheets had to be consulted for Phase 1 and Phase 2 of the project. For Phase 1, the datasheets that were used were for the 555 timer, 7805 voltage regulator, 3-8 decoder, synchronous counters, and inverters. One commonality between all of these components is that their respective datasheets were useful for information regarding pinouts, various operating conditions, and physical characteristics such as size. When making PCBs, it is critical that all factors are taken into consideration to prevent any design flaws from negatively impacting the performance of the PCB. As for Phase 2, the datasheets that were frequently referenced were that of the Atmega328PB, battery holder, crystal, tag connect, battery, LEDs, and all other components on the PCB. By looking at the data sheets, each component could properly be interfaced into the Phase 2 PCB design.

**Software**

To successfully complete the project, different types of software had to be utilized. Although software such as Multisim has been used frequently in classes such as circuit theory, this project exposed the team to Fritzing, EAGLE, and \_\_\_. It was learned that Fritzing is an advanced software tool used by designers and engineers alike to transition projects and ideas into a functional prototype. The features that were specifically utilized were the schematic and digital breadboard features. The purpose of using this software was to create a breadboarded schematic of the original “Stop the Light” game in the SENG hallway. When trying to search for these components in Fritzing’s parts library, it became apparent that Fritzing’s library wasn’t as advanced as the libraries of other schematic software. To compensate for this issue, each of the 7400 series chips in the Phase 1 design had to be custom made using Fritzing’s “Parts editor” tool. This made the process of recreating the circuit in Fritzing much more involved than expected seeing as we had to learn how to go about this process.

To make a custom IC, the first step that had to be done was clicking on the Core tab in the parts window and scrolling down until the ICs section came into view. Following this, The IC icon was clicked and dragged onto the breadboard window to import the part into the project. After clicking the Inspector window and clicking on the IC properties tab, the name and number of pins on the IC chip could be changed. This process was done for each chip on the board. As for the other components in the project, they were all included in Fritzing’s parts and components library, so they were added into the project normally. With all the components imported into the project, the schematic was able to be made and the breadboard schematic was wired after.

The easiest and most important schematic of this phase of the project was made using the EAGLE software. EAGLE stands for Easily Applicable Graphical Layout Editor, which allows users to make printed circuit boards (PCBs) based on wiring schematics designed by the user. EAGLE also allows users to manufacture their PCB designs using computer-aided manufacturing (CAM) features. Before this class, two out of four members in the team had experience using the program. This was helpful as it reduced the learning curve experience by the group as a whole seeing as help from teammates was always available. The hardest part of learning the program was in making the PCB. At times, all of the parts on the PCB made routing quite overwhelming, especially since we were novices in using the program. However, as time went on and more experience was gained, this process became substantially easier and we felt as if the layout of our board was pristine. Overall, we believe that EAGLE is a well-rounded piece of software and it was vital for success in the class.

The last program that had to be learned was using Fusion 360. Fusion 360 was used to make the design of the battery holder tray that was used to hold the battery holders on the pick and place machine. After watching tutorials on YouTube, it became apparent that the process of making the design for the battery holder was quite easy. However, when we uploaded the design to the laser cutter and did a test cut on a piece of paper, it was found that the holes made for the tray were to close to the edges of the holder. This test helped us fix our mistake which allowed us to successfully make the wood tray in the laser cutter.

**GitHub**

For the documentation of design part of the project, the team decided that GitHub was the perfect resource to use. A link was provided on the back of the Phase 2 PCBs so that people who are interested in our design could read about all the steps that were taken from start to finish to create the PCB. In addition to the steps that were taken, the documentation talks about what was learned and challenges that arose while working on the project. Links to datasheets and other resources are also present in the documentation of design challenge page since we felt that it was important to give readers insight to decisions and design tradeoffs that were made in the project. All forms of documentation that were made for the project such as the Bill of Material, Requirements, Acceptance Test Plans, and Engineering Test plans are all included in the documentation of design challenge as well. Lastly, any methods of testing that were done to the Phase 2 PCB are also included in the documentation of GitHub.