THE UNIVERSITY OF QUEENSLAND

Team 39

Final Report

Composed by: Vigneshwar Parikumar
Student #: 43952991
Team Members: Samual Kelly, Jonathan Chang, Duong Anh Khoa, Vigneshwar Parikumar

1 Team Design Overview

At the beginning of the project, the team decided that project responsibilities should be split up according to individuals' strengths. This worked well, as the main software portion of the project was completed by week 6. This aspect of the project was undertaken by Khoa. The MCU responsibilities were allocated to both Sam and Jonathan. This involved the programming of the microcontroller and designing the MCU board, which included the power management for standby mode. Vignesh was allocated the sensor design responsibilities. This involved the design of sensor circuits and design and testing of the sensor PCB.

Milestones were set during the semester in the form of a sprint. This sprint dictated the timeline of the project. If a milestone dictated by the sprint was not complete by the due date, all effort would be placed on completing that one task. This ensured that all systems needed to complete the project was completed at a timely manner. This sprint methodology did work for most of the semester, however due to mistakes made within the hardware design during the semester, certain aspects of the sprint were modified.

Trello was used for project management throughout the project. It was used to identify the project backlog, systems that need to be completed, systems that have been completed but not checked and systems that have been checked and verified by the team. Systems were split into subsystems and entered into Trello. Once these subsystems were completed, they would move onto the verification backlog. Once the team has verified the work, the subsystem would move to the verified list. Figure 1 shows the scrum board from week 8.

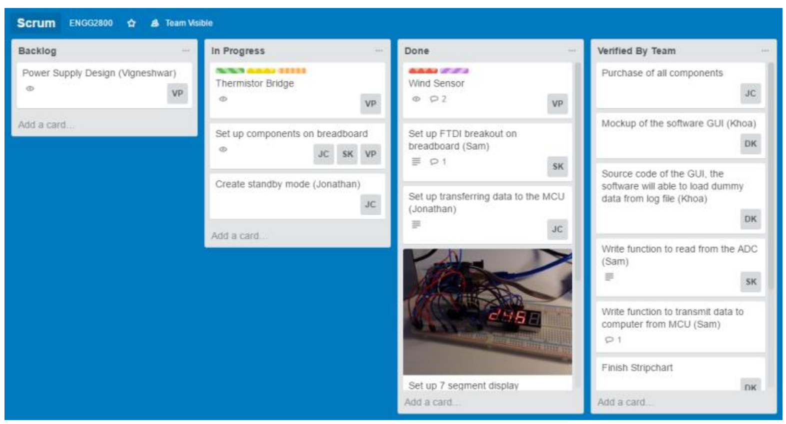


Figure 1: Scrum board from week 8

Commenting on each subsystem within Trello allowed for the team to communicate progress throughout the semester. A Facebook chat was also set up to enable communication. This was used to arrange team meetings and communicate concerns about the project.

2 Individual Contribution

My contribution towards the project is the design and testing of the sensor module. This included the design of each individual sensor module, the design of the PCB layout, the soldering of components on the sensor PCB and testing the PCB, ensuring that it conforms to tolerances.

The design process began by researching projects that other people have successfully completed. The light sensor was first considered, and a design that employed a comparator op amp configuration was built. This involved the LDR placed in a voltage divider and feeding the divided voltage to the positive input of the op amp. A fixed voltage divider was coupled to the negative input of the op amp. This method worked as a light sensor, however voltage changes in the circuit were discrete.

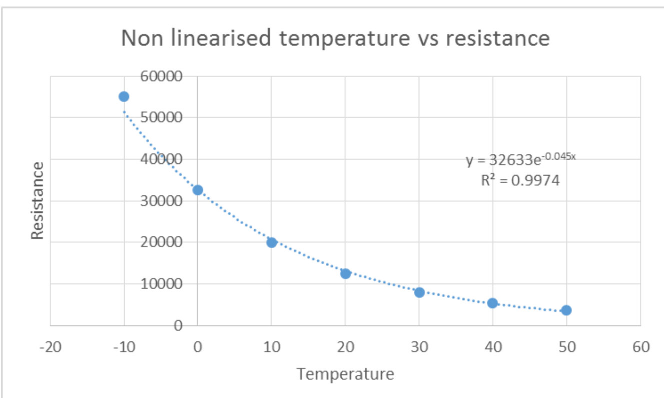
This idea was soon sidelined. An incorrectly designed differential op amp configuration was then considered. This consisted of a fixed divider passing through the positive op amp input and variable divider passing through the negative op amps inputs, and then given negative feedback. This did allow the voltage to be changed at a linear rate, however the circuit could not be easily analysed. It was therefore sidelined.

The design that was chosen was a LDR placed in a voltage divider, with the input voltage divided to 1 volt, then passed through a buffer before passing through the LDR divider. This buffering ensured that the impedance did not mix and alter the output of the LDR circuit. The 1 volt reference voltage in the divider ensured that the voltage output was contained between 0-1 volts. Semi-linearity was achieved in the hardware by using the LDR as half of a voltage divider (Muff Wiggler Forum). A low pass filter was added to filter out any excessive noise.

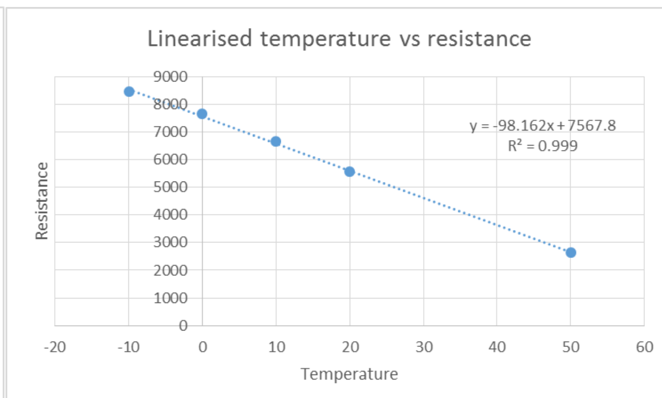
The design of the wind sensor was that of the initial PCB task. This design was suitable for the project, as we discovered that the sensitivity of the sensor was appropriate for wind sensing use. The voltage ranged from 0 to 4 volts. Noise filtering was decided to be non-essential in this circuit, as the voltage changes were discrete.

The temperature sensor was designed employing a differential amplifier. A unity gain differential amplifier was chosen for ease of calculation. The first design consisted of a variable voltage divider that ranges from 0 to 1.5 volts, which was subtracted off a fixed 3 volt reference. This however created a voltage output range of 0 to 1.5 volts. To fix this problem, a voltage divider was added that divided the voltage by 1.5.

The thermistor was linearized within the hardware. This was achieved by placing the thermistor parallel with a fixed resistor (Create-and-Make, 2012). The graphs below show the effect of adding a parallel resistor on the linearity of the thermistor.



Graph 1: Nonlinear nature of thermistor



Graph 2: Linearised thermistor

As graph 1 shows, the resistance nature of thermistors is inherently nonlinear. Graph 2 shows the effect of adding a parallel resistor on the resistance nature of the thermistor. The nature of the thermistor is altered, and is far more linear.

Figure 2 below shows the final schematic of the circuits described above.

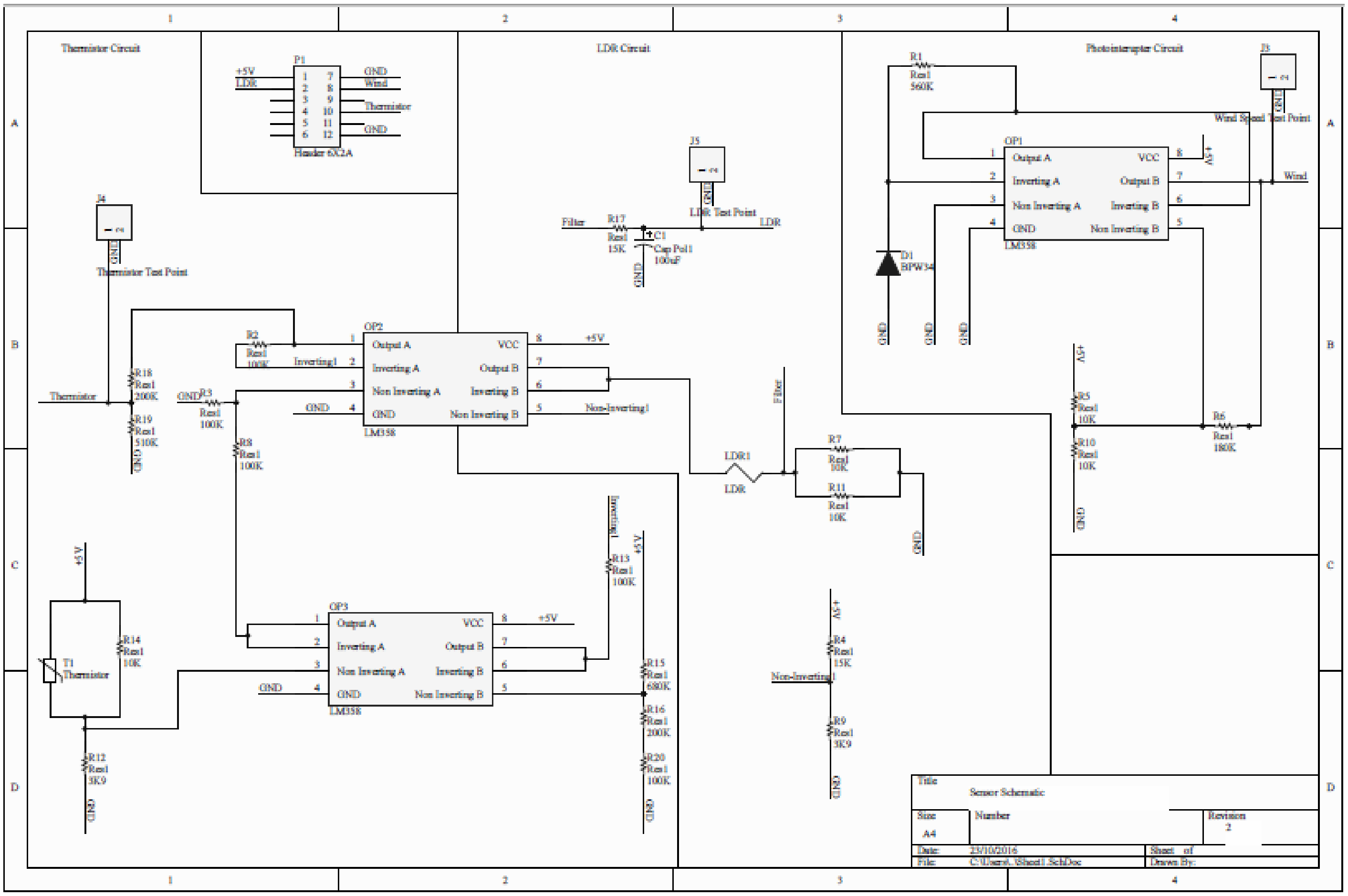


Figure 2: Final circuit schematic

Components were placed on the PCB as to minimise the physical size of the PCB. Component size factors were also taken into account. Since the electrolytic capacitor used would interfere with the turbine blade, provisions were made to place this greater than 2cm (radius of the turbine) away from the turbine. The PCB layout is presented in figure 3.

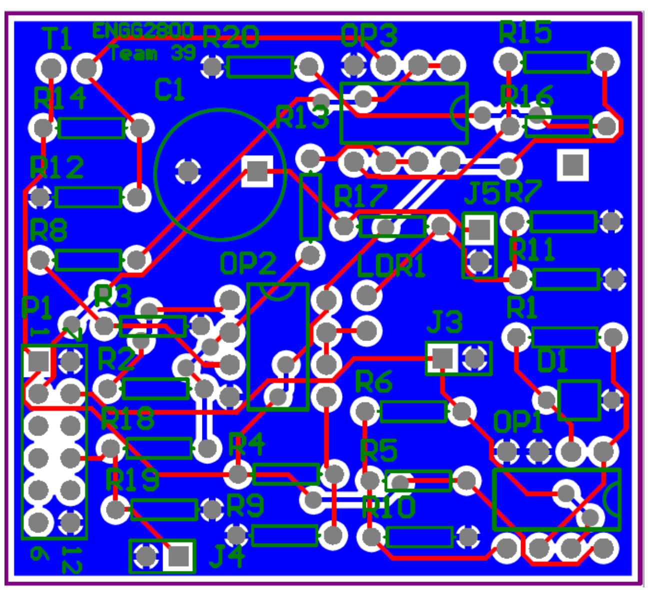


Figure 3: PCB layout

When the final PCB arrived, mistakes were found. The thermistor schematic that was submitted (not that of figure 1) consisted of R18 connected to the other end of R2. This changed the output reading dramatically. To fix this, the trace connecting R2 to R18 was cut, and flywire was place from R18 to the other side of R2. This fix is shown in figure 4.

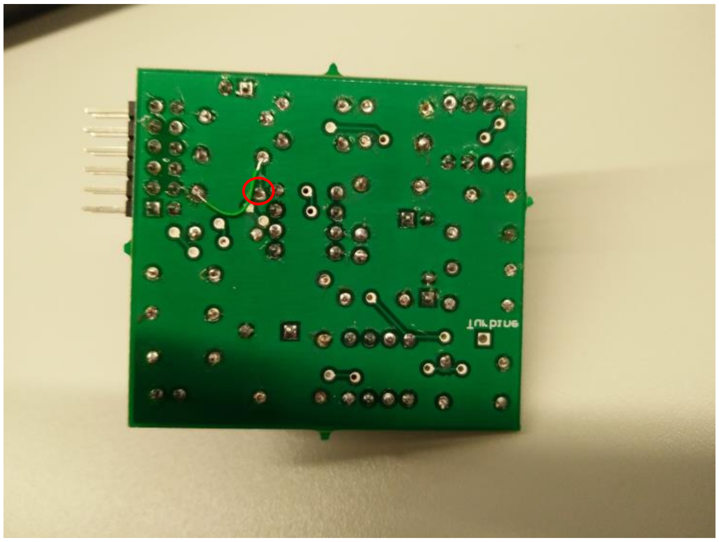


Figure 4: Flywire fix (showing cut trace in red circle)

Test point placements for J3 and J5 were also incorrect, with them lying directly under the turbine. The LDR was also partially under the turbine, which affected the final prototype's use. These problems could not be fixed. A provision for an LED was also missed on the PCB. This was fixed by attaching an LED to the power and ground of other resistors. Figure 5 shows the mistakes made and the attempted fix.

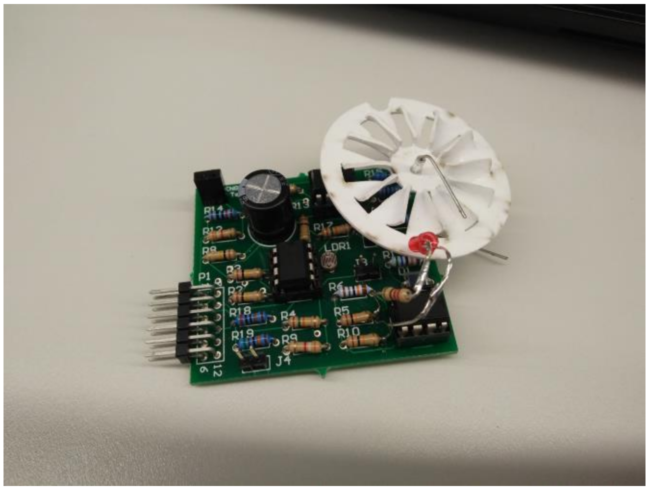


Figure 5: Mistakes and attempted fix on sensor PCB upper layer

The design did flow smoothly, although there were complications with the PCB. However these complications could be overcome and a successful result was produced.

3 Sustainability Considerations

During the design of the sensor module, sustainability issues relating to hardware and software were at the forefront of the decision making process. On the prototype, hardware sustainability considerations included the use of restricted substances on the physical hardware and energy efficiency of the hardware. Software sustainability for the prototype dictated that the code is maintainable.

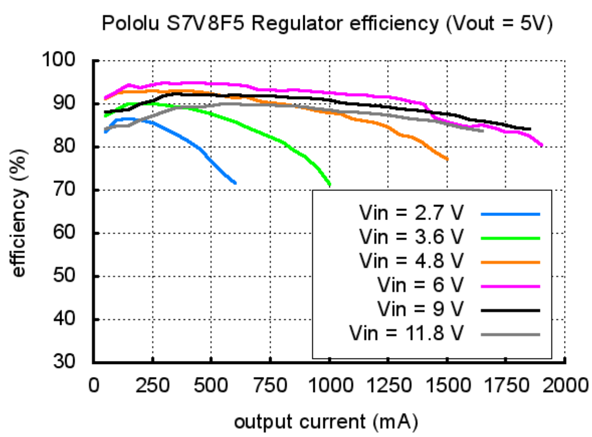
Hardware wise, the ROHS directive gave indication as to what components were used on the prototype. This directive restricted the use of substances like lead or mercury in the prototype. This product was constructed with materials that complied with ROHS. For ease of soldering however, leaded solder was used. This approach was appropriate for a prototype, however for mass production, this method of soldering will make the product non ROHS compliant.

Also for ease of soldering, through-hole components were used. This was suitable for the prototype however, the use of through-hole components increased the PCB footprint. To reduce the end of life waste from the disposal of the PCB, the footprint of the PCB should be reduced when mass producing. This can be done by using surface mount components, which have a smaller footprint compared to its through-hole counterpart. A smaller PCB footprint will reduce mass production costs relating to the PCB manufacture. This will also increase the amount of PCBs that can be manufactured at any one time, reducing the energy used during the PCB manufacture. The result of this will be a reduced embodied energy over the life of the full scale product.

Embodied energy of the total products lifespan should also be considered in full scale production. Efforts should be taken to reduce the total embodied energy of the product. This could be done by using alternative processes when manufacturing.

Energy efficiency was not dictated by a directive in this project, however guidelines were given for the operating life of the device. To address this need in the prototype, design considerations relating to power were vital during the design phase.

The prototype was designed with current draw and efficiency considered. A Pololu 5V step up/step down voltage regulator was chosen due to its measured efficiency of over 90% and its large operating voltage



range. Graph 1 shows the efficiency of the Pololu voltage regulator over a wide voltage range.

Graph 1: Efficiency of the Pololu 5V step up/step down voltage regulator (Pololu, 2016)

Over the voltage range of 2.7-6V, the efficiency of ranges from 85% to 95%. However, this efficiency did depend on other external factors, including ambient temperature and heat sinking (Pololu, 2016).

As shown on graph 1, the Pololu 5V step up/step down voltage regulator achieves optimal efficiency when operating at low currents. To reduce current draw from the sensor module, any voltage dividers used were implemented with large resistors.

WEEE and REACH directives were not explicitly addressed in the prototype. However, for mass production purposes, these directives should be considered. The WEEE directive dictates that producers of electrical and electronic equipment are responsible for the recycling of the equipment (eco3e, 2016). Considerations should be made in terms of component recyclability for a mass produced product. The REACH directive aims at restricting the use of harmful chemicals in the manufacture of electronics. For a mass produced product, considerations should be made before undertaking certain processes. If harmful chemicals are being used, alternative methods should be sought.

These process combined should improve the sustainability credentials of the project, by reducing total embodied energy and ensuring the product is recyclable.

4 Course Reflection

From the beginning, I expected this course to challenge my understanding of electronics, and force me to learn new concepts from a design standpoint. After my experience in this course, I feel as though this course has delivered this outcome. Throughout the course, I have made many mistakes however, each mistake I made posed as a learning experience.

At the beginning of the course, my electronics knowledge consisted of ENGG1300 content and other light reading I did in my spare time. Having no experience in electronics design, I begun the task by looking at projects others have completed. Later in the project however, I realised this was the wrong process to take, as the circuits they built did not satisfy the needs of the project.

After this process failed, I started playing with different op amp configurations, fixated on getting the LDR to change from 0-1V. Again, this process lead me to create a circuit that was filled with noise, and could not be analysed easily. From these mistakes in my initial design process, I have learned to first break up my task into small subsystems, then design each subsystem individually and analyse the circuit on paper before building the circuit. Once I did this, the circuit design process became more streamlined and efficient.

Since I have never designed a PCB before this course, this was also another major learning curve for me. The initial PCB task worked, however the placement of the LED could have been refined. When designing the sensor PCB, mistakes were not noticed until the PCB arrived. One of the major issues was with an incorrectly placed track. This track had to be cut and a flywire had to be placed to correct the incorrect connection. This mistake allowed me to learn the process of PCB modification. Other mistakes were made in terms of component placement, with test points placed straight underneath the turbine and the LDR placed in close proximity to the turbine. Again, these mistakes are mistakes I will probably not make in future projects, since I have made them now.

In terms of team work, I have experienced the disasters that result from a lack of effective project planning. Initially, in the first few weeks of semester, progress was slow due to inefficient use of project management tools. In the early part of the course, any tasks that was entered in Trello was vague. After the first meeting, we realised we had to make each task more concise, splitting systems up into subsystems. In this course, I have honed my project management skills, and learned how to effectively use Trello for project management.

For most of the semester, the team communicated regularly and work on the project flowed smoothly. However, there were times when communication broke down. When this happened, progress slowed. When communication issues were resolved however, progress picked up. The team worked well when communication channels were efficient. The work was also split up according to individual member's strengths, with Sam and Jonathan working on the MCU, Khoa working on the software and myself working on the hardware. This worked to the team's advantage, with all of the software completed by week 6.

Communication breakdowns occurred with the ineffective of use of Trello earlier in the semester. We did not effectively outline and communicate the early task we had to complete in order to continue with the rest of the project. Instead vague descriptions of tasks were outlined. This slowed down initial progress. Once Trello was fully utilised however, problems regarding communication issues were less frequent. Full utilisation involved outlining each subsystem involved with each task and regularly commenting on each subtask to communicate our progress with them.

Overall, I believe that I have learned many processes that professional engineers undergo in order to create a successful prototype. I have not come across most of these processes in my career thus far. Self-learning has been an integral part of this course. I believe that I have plenty of work to do to hone my self-learning skills, but am heading in the right direction with my self-learning. Changes in my overall time management is necessary to achieve optimal results.

5 References

Basic Electronics Tutorials. (2016). *Differential Amplifier - The Voltage Subtractor*. [online] Available at: http://www.electronics-tutorials.ws/opamp/opamp_5.html [Accessed 9 Nov. 2016].

Basic Electronics Tutorials. (2016). *Op-amp Comparator and the Op-amp Comparator Circuit*. [online] Available at: http://www.electronics-tutorials.ws/opamp/op-amp-comparator.html [Accessed 9 Nov. 2016].

Echa.europa.eu. (2016). *Understanding REACH - ECHA*. [online] Available at: https://echa.europa.eu/regulations/reach/understanding-reach [Accessed 9 Nov. 2016].

Eco3e.eu. (2016). *WEEE – WEEE Directive | Eco-3e*. [online] Available at: http://eco3e.eu/regulations/weee-regulation/ [Accessed 9 Nov. 2016].

Facstaff.bucknell.edu. (2016). *Unity Gain Buffer Amplifier*. [online] Available at: https://www.facstaff.bucknell.edu/mastascu/eLessonsHTML/OpAmps/OpAmp3Note1Buffer.html [Accessed 9 Nov. 2016].

Fox, G. (2012). *Principles of Linearizing the Thermistor.*. [online] Create-and-make.com. Available at: http://create-and-make.com/principles-of-linearizing-the-thermistor/ [Accessed 9 Nov. 2016].

Muffwiggler.com. (2016). *MUFF WIGGLER :: View topic - Photoresistor/LDR to CV module*. [online] Available at: https://www.muffwiggler.com/forum/viewtopic.php?p=1607461 [Accessed 9 Nov. 2016].