University of Illinois at
Urbana-Champaign

ECE 445 Team #14

Child Development Sensor
Individual Progress Report

Xiang Wen (xwen10)

November 7, 2017

Contents

1 Introduction

The Child Development Sensor project is a multidisciplinary, collaborative effort among various research teams at the University of Illinois. The aim of this project is to assist and improve current research methods used in the study of behavioral and psychological development of young children. Combining voice and ECG data, researchers are able to draw conclusions regarding a child's physiological and mental growth. Current methods to obtain this data are not optimal. Studies are conducted in a laboratory - a controlled environment foreign to the child. The instruments used to capture the data are also intrusive and uncomfortable to the child. Therefore it is necessary to develop a device that is capable of capturing data in a familiar, natural environment (i.e. at home) and unobtrusively.

My team and I will be developing the electronics needed to capture the necessary voice and ECG data. Sensor nodes will be placed on a child and data will be wireless sent through Bluetooth to a main processing hub. The hub will then send the processed data to a database using WiFi which will be accessible to researchers. I will be in charge of the power electronics for both the sensor node and the main hub.

My main responsibility will be delivering the necessary power for our circuits to run for all functions. This means that the circuits must be sufficiently powered while idle as well as collecting and transmitting data. For the sensor node, I will be developing a DC-DC converter. This must be able to step down the voltage from a 4.5 V three AAA battery source to the 3.3 V required for the components to run. For the main hub, I will be developing a AC-DC converter. This will be converting the wall outlet voltage, 120 VAC, 60 Hz, to 3.3 VDC required for the components on the main hub to run. It is clear that the power electronics is integral to the project. The components I will be developing will be key to the operation of our modules. If there is insufficient power, the components will not run. Likewise if there is too much power, the components will be destroyed. Therefore careful consideration must be used to design the power electronics. Currently both converters have been designed and are now in the physical prototyping phase. The next step is to validate the physical converters, which will be discussed later, and to create the PCBs for each of the converters.

2 Design

2.1 Main Hub

For the main hub, a full-wave rectifier will be designed. The full-wave rectifier topology was chosen to provide a larger average voltage, compared to the half-wave topology, as well as improved filtering with an output capacitor. The source of power will be the wall outlet, which, in the United States, delivers 120 VAC at 60 Hz. The wall voltage will be fed into a transformer which will isolate the circuit components from the power supply. The transformer also steps down the AC voltage through the turns ratio. For my design, I have chosen a 12:1 transformer (BC2DA)[1]. The output of the secondary side of the transformer will thus be about 10 VAC. Two diode drops, each approximately 1 V, must be accounted for due to two diodes(1N4004)[2] being in series with the output at all times. This will approximately leave 8 VAC across the output. Attaching a filtering capacitor at the output, a DC signal can be generated. This signal can then be fed into a buck converter which steps down the output of the rectifier to the necessary 3.3 V.

The filtering capacitor at the output can be calculated as follows:

$$C = \frac{I_{out}}{2f\Delta V} \tag{1}$$

Where f is 60 Hz and ΔV is 0.6 V to give a voltage ripple of 3.0 V to 3.6 V. I_{out} can be calculated simply by summing up the maximum load requirements of the components, P_{OUTmax}, on the main hub and dividing by the output voltage, 3.3 V. P_{max} is obtained from maximum operating values of voltage and current given in the datasheets of the two components: the microcontroller[7] and bluetooth module[8].

$$\begin{aligned} P_{OUTmax} &= P_{COMmax} + P_{MICROmax} \\ &= .864W + .37W \\ &= 1.234W \end{aligned} \tag{2}$$

Using Equation 2, I_{out} is calculated to be $I_{out} = \frac{P_{OUTmax}}{3.3V} = 0.374$ A. Using this value and Equation 1, the filtering capacitance can then be calculated to be approximately 5.1 mF. Knowing the output voltage and the output current, the equivalent load resistance can be calculated by Ohm's Law. This is obtained to be R $= \frac{3.3}{0.374} = 9\ \Omega$.

Figure 1 shows the proposed schematic with labeled parts with values that will be implemented in the test circuit.

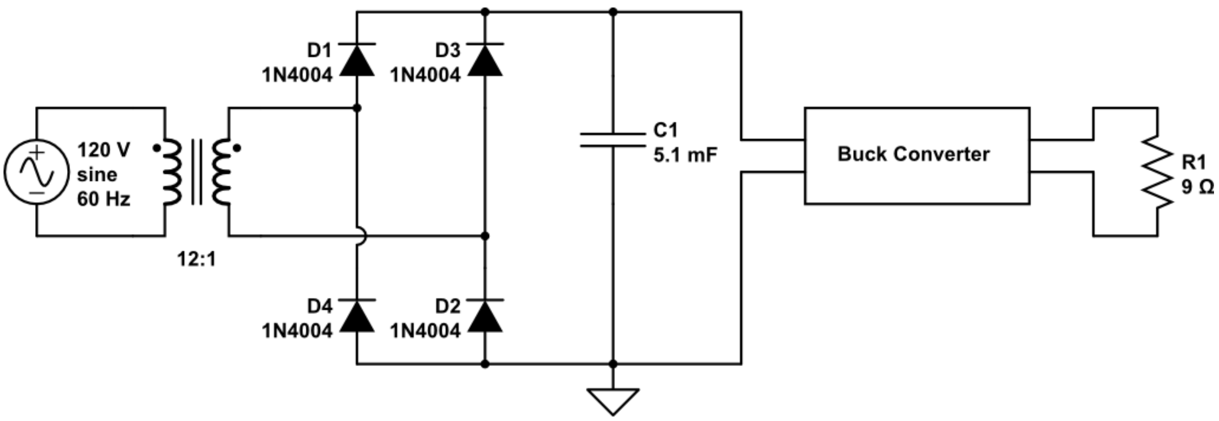

Figure 1: Rectifier Implementation

The output of the rectifier will be cascaded with a buck converter. The buck converter will be used to convert the approximately 8 VDC output of the rectifier to the necessary 3.3 V. The buck converter will be similar to the one that will be presented in the Sensor Node section and therefore will be discussed in-depth later. This is a change since the design review because physical transformers come in standard winding ratios; a unique turns ratio would require personally designing and winding a transformer. Consequently, the input to the diode bridge cannot be exactly 3.3 V + two 1 V diode drops. It is therefore necessary to work around this and a convenient solution is to use a buck converter.

Figure 2 shows an early simulated design of the full-wave rectifier circuit. Notice that the input to the diode bridge has been abstracted to an ideal sinusoidal voltage source. Because this model is not the case in the physical implementation, an updated schematic has been created to better simulate the physical circuit. To model a transformer in LTSpice, two coupled inductors must be used. The relative values of the inductors correspond to the turns ratio. To create the desired 12:1 transformer, a 144:1 inductor ratio is needed. Figure 3 shows the updated simulation with

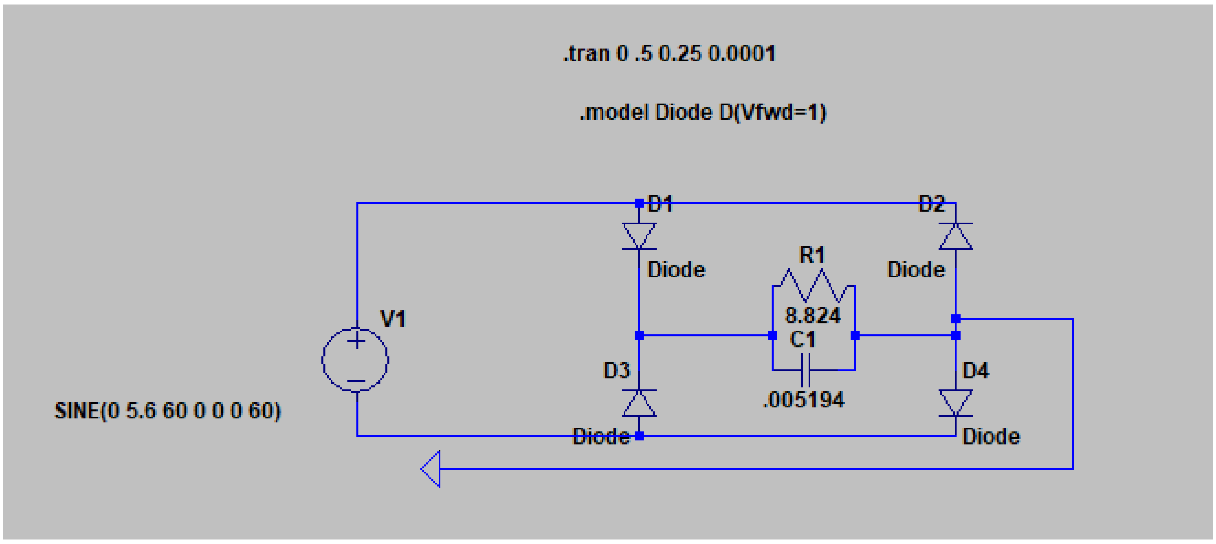

Figure 2: Draft Rectifier Simulation Schematic

the included transformer. Figure 4 shows the output voltage waveform. Notice that this is not the final output voltage that is desired; however, this is the close to the expected output of the rectifier bridge as previous discussed. This output must be cascaded with a buck converter to achieve the 3.3 V. Because there will be large components, such as the transformer and outlet

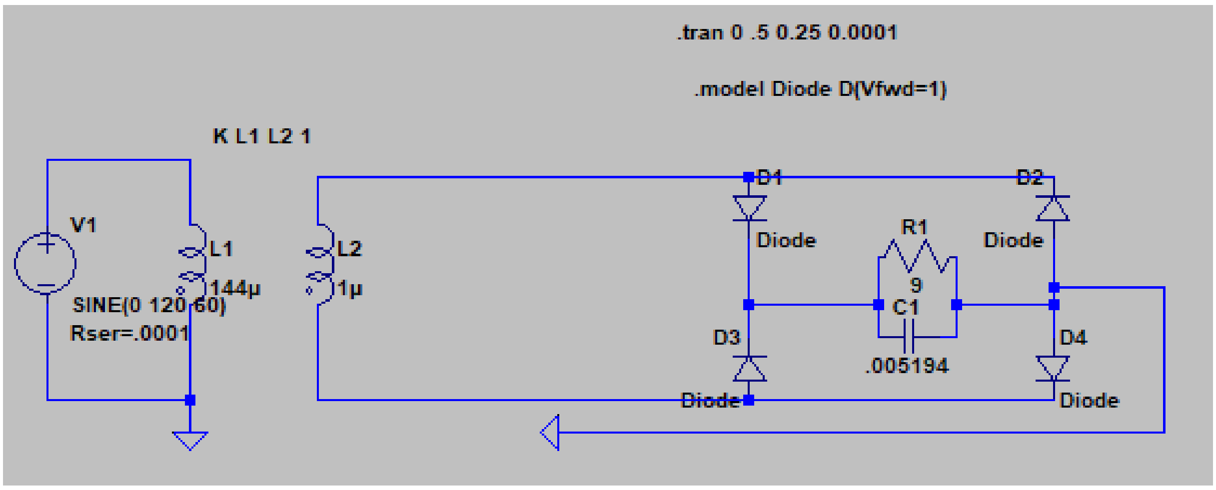

Figure 3: Updated Rectifier Simulation Schematic

connectors, a PCB will not be developed to incorporate all parts in the schematic. The PCB will most likely have a diode bridge, filtering capacitor, and the buck converter. PCB layout has not begun as of yet but will commence shortly after the physical circuit has been validated on the breadboard.

To validate functionality of this circuit, the following requirements must be met. The average voltage must be 3.3 V. This is a standard operating condition for both the microcontroller and

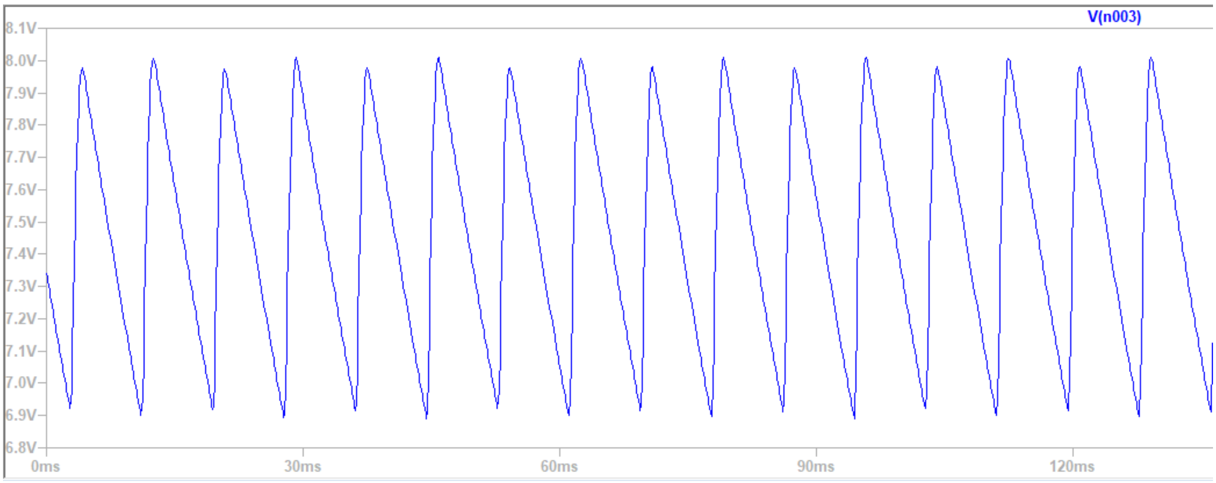

Figure 4: Updated Rectifier Simulation Output Voltage

the Bluetooth module to operate. The second condition is that the voltage ripple must be less than 0.6 V. This is the range between 3.0 V and 3.6 V. This is necessary because below 3.0 V or above 3.6 V, the components will shut down. Therefore it is imperative that the ripple be kept lower than the maximum range. The validation process would involve scoping the signal at the output of the buck converter and confirming the waveform. Ideally, the correct waveforms should be obtained at all outputs of each components. Therefore the output of the transformer and also the rectifier can be scoped. This allows a straightforward debug process if the output of the buck converter is not the desired 3.3 V. Figure 5 shows the simulated ideal output voltage waveform. In the case of failure, or an undesired output, the aforementioned debug process will occur and components will potentially be changed such that the failure is corrected.

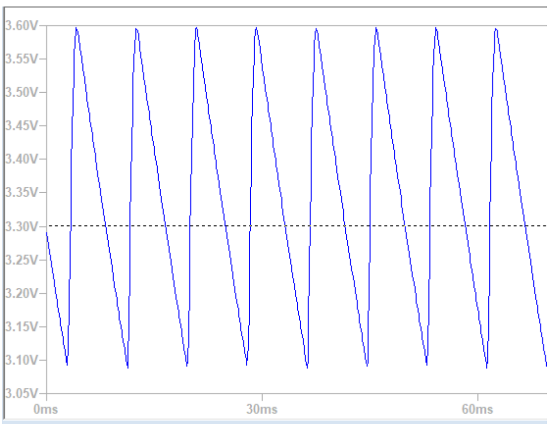

Figure 5: Rectifier Simulation Ideal Output Voltage

Lastly, the entire AC-DC converter must be tested with the rest of the components. Initially, modular testing will be done. Each individual component will be powered by the converter by itself. If all required functionality is capable when powered by the converter, the converter will be successful. If the converter is successful for both components, the converter will be tested on both components simultaneously. If the converter passes the system test, the functionality of the converter has been validated. In the case of failure, the output voltage of the converter will be monitored. Failures may occur due to the output voltage going out of the allowed ripple range caused by a load transient. This can occur when one of the components powers on or transmission begins. Circuit components will be recalculated in the case that failures occur and prototyping must restart with the new values.

By November 13th, a breadboard prototype of the entire AC-DC converter will be complete. Components have all been purchased so the converter is ready to be built. Scope data will be captured and validated with the requirements. By November 20th, modular and system testing will be complete. And hopefully by Thanksgiving break, a PCB for the smaller components of the converter will be developed with the rest of the components on the main hub.

In the case of total failure of the AC-DC converter, a charger that converts wall voltage to 3.3 V has been purchased to deliver power to the rest of circuit. The output of the charger has been scoped and has been tested to be able to modularly power the Bluetooth module. This will be a fail-safe option and will only be used for powering the microcontroller and Bluetooth module while the converter is being developed and validated.

2.2 Sensor Node

For the sensor node, a DC-DC step down (buck) converter will be designed. The source of the circuit will be a 4.5 V battery, which is obtained by connecting three AAA batteries in series. There are three components that the converter will be powering: a bluetooth module, BT832F[10], an ECG sensor, BMD101[9], and a microphone[6]. Maximum load can be calculated by obtaining the maximum operating conditions of each component.

$$\begin{aligned} P_{OUTmax} &= P_{BTmax} + P_{ECGmax} + P_{MICmax} \\ &= 26.64\ mW + 3.24\ mW + 2.16\ mW \\ &= 32.04\ mW \end{aligned} \tag{3}$$

Therefore the equivalent load can be modeled as a resistance with the value:

$$R = \frac{V^2}{P} = \frac{(3.3\ V)^2}{32.04\ mW} = 339.9\ \Omega \tag{4}$$

The current drawn from the load can then be calculated to be

$$I_{load} = \frac{3.3\ V}{339.9\ \Omega} = 9.7\ mA. \tag{5}$$

Knowing these values, components can be selected for the physical design. These maximum stress values must be determined prior to building so components do not go over their rated limits. There are three design routes that are available and are being pursued. The first method is to use the LM2596[3] buck converter chip. This package abstracts the entire switching process; the inductor and filtering capacitors must still be selected. The second method is to use a 3.3 V low dropout regulator. This process is the simplest and due to the very low load current, this method may be the most power efficient. Lastly the third method, and the most ambitious, is to design the entire buck converter using the LM25141 buck controller. This method abstracts the closed loop control only. Therefore switches, inductors and capacitors must all be designed.

There has been progress with the physical design of the DC-DC converter. The 4.5 V source was successfully constructed. A series battery connector was purchased and three AAA batteries were placed. The output was measured to successfully deliver approximately the expected 4.5 V. Figures 6 and 7 shows photos of the physical battery source.
A buck converter evaluation module using the LM2596 was also purchased to test the functionality of the part. Figures 8 and 9 shows the connected buck converter with the battery source. The multimeter measures the desired 3.3 V output.
Knowing that the LM2596 is able to deliver the necessary output voltage, the natural next step was to modularly test the converter with the Bluetooth module on the sensor node. This was

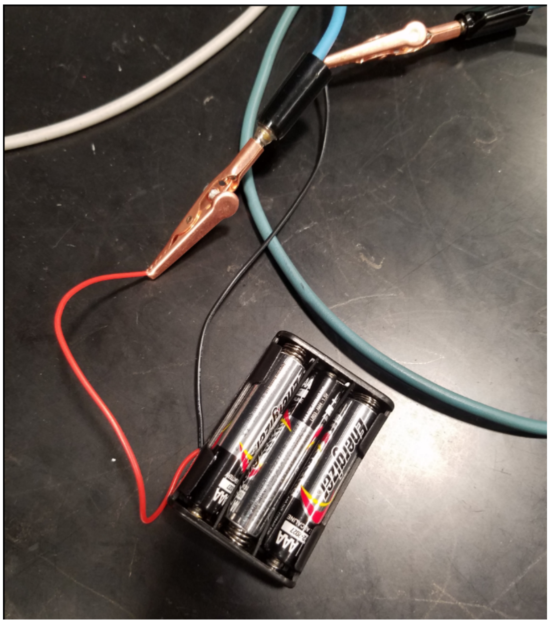

Figure 6: Battery Source

Figure 7: Battery Source Voltage

a success; the Bluetooth module was able to transmit while receiving power from the battery source. The LM2596 method is therefore a valid design path for the buck converter.

To begin design with the LM2596, the application notes for the part was studied [3]. Figure 10 shows a simple schematic with the labeled components and their values. Using the evaluation module, the resistor divider for the feedback can be determined and used for the final converter design. The benefit of using the buck converter is the typical high power efficiency. Having higher efficiency allows a longer operation time and therefore is important for the convenience of the product. It follows that the buck converter, itself, draws minimal power. This is important for the temperature of the overall device and can eliminate the need for a large heat sink - a typical requirement for voltage regulator devices.

The second design method for the DC-DC converter involves the use of a low dropout regulator. The LDO of choice is the TPS73033DBVR. This LDO provides a steady 3.3 V for a wide range

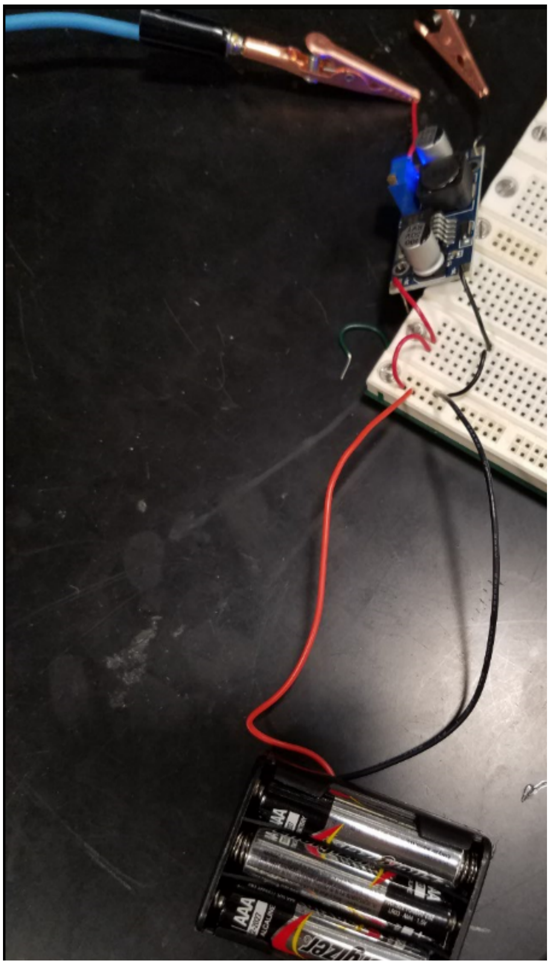

Figure 8: Battery Source + Buck Converter

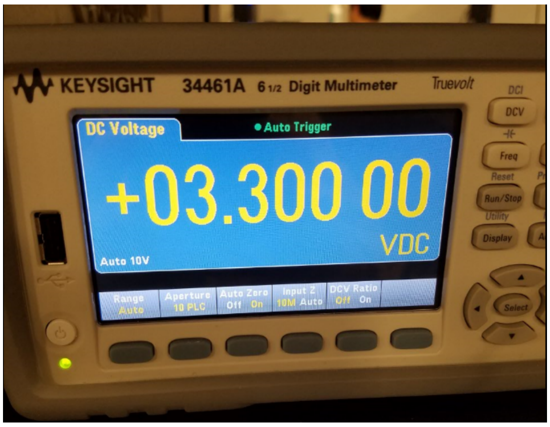

Figure 9: Output Voltage

of input voltages. This method is the least complex, but may end up giving the best efficiency. This is because an LDO typically has efficiency of $\eta = \frac{V_{out}}{V_{in}}$. Because 3.3 V is very close to 4.5 V, the LDO can have an efficiency of approximately 73%. Although this is not perfect, this is relatively good for an LDO. In terms of minimizing board space, the LDO is the clear winner. Additionally, switching converters, such as the buck in the first method, do poorly in terms of efficiency under low load conditions. Therefore, an LDO may be the best fit for the power management of this circuit. An interesting topic to note is that as batteries lose charge, the voltage begins to decrease. Therefore as V_{in} decreases, the efficiency for the the LDO method

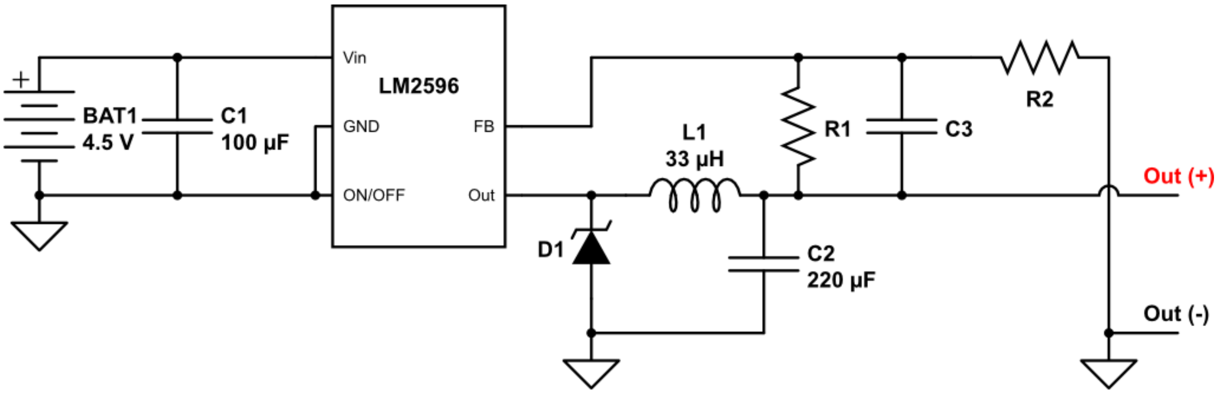

Figure 10: LM2596 Schematic

increases. This improves operation time at decayed voltages. Figure 11 shows a schematic with the necessary parts for circuit operation. Both circuits will be constructed and the efficiencies will be measured to determine which method is preferred.

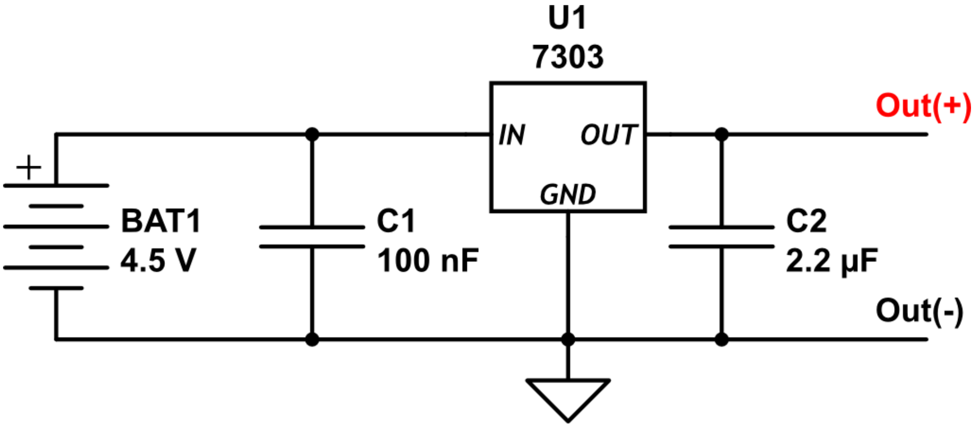

Figure 11: TPS73033DBVR Schematic

The last method is to design the buck converter with an abstracted switching control. This method uses the LM25141 [5] buck controller. The chip abstracts the closed-loop control for the switches but the rest must be designed for. This includes the switches, inductor, and filter capacitors. Design has begun on this method and parts have been ordered. This method provides additional complexity to the project and serves as a unique learning experience. Depending on time, this design path may not be pursued for the final circuit. Figure 12 shows a schematic of the designed values if this method is pursued.

To validate the DC-DC converter, the converter, regardless of design method, must supply 3.3 VDC. The ripple must be within the same 0.6 V range similar to the requirements for the components on the main hub. The output voltage waveform of the converter will be probed to confirm that this is within the requirements. Additionally modular and system level testing will take place for the DC-DC converter as with the AC-DC converter. Each individual component will initially be powered to see if all functionality is capable when powered by the converter. Once each component passes, the system will be powered altogether and tested with all components being powered by the converter. An additional validation test for the DC-DC converter will be the measurement of efficiency. Efficiency is important because the sensor node is battery powered and therefore operation time is a key factor for the overall product. Efficiency will be measured as a factor of output power over input power. The output power is given in Equation

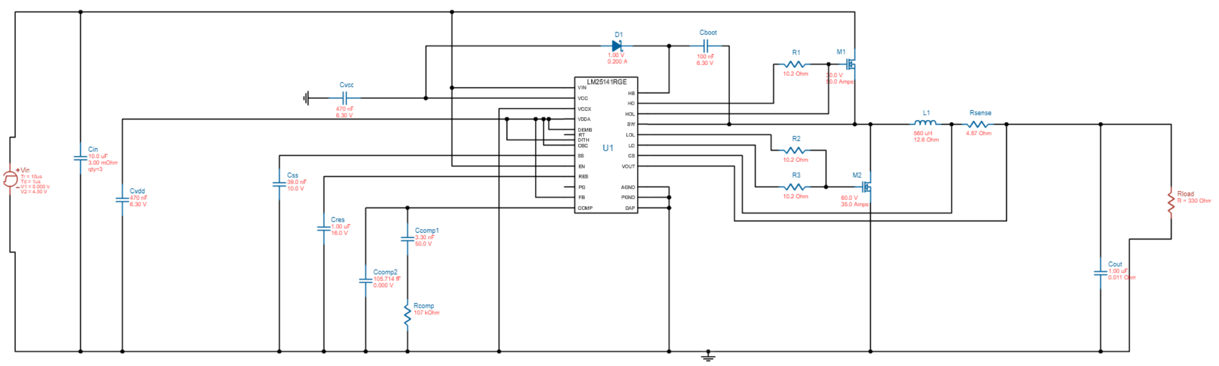

Figure 12: LM25141 Schematic

3. The input power will be measured with a very small resistor, about 100 Ω in series with the battery source. Measuring the drop across the resistor will allow a good approximation of the input power. By Thanksgiving break, the best method for the purpose of the sensor node will be chosen and PCBs will be built with the rest of the circuit components. In the case of failure for any of the methods, the constructed circuit will be debugged and component values will be recalculated. If total failure occurs for any of the methods, there are two more methods to achieve the desired result.

3 Conclusion

Overall progress has been steady with both converters. The rectifier for the main hub will be prototyped on a breadboard or perfboard soon; this is to be completed by November 13th. After the output of the rectifier has been validated with the waveform requirements, the next step will be to test the rectifier with the individual components in the main hub. The PCB design will commence shortly after the rectifier has been successfully validated; this is to be completed around Thanksgiving break. The buck converter is still in the development phase. The methods mentioned in the Design section for the buck converter will be compared to determine the best fit for the final circuit. The first buck method has already been validated to be compatible with the circuit components. The efficiency data will be measured by November 13th. Also by this date, the LDO circuit will be constructed. By Thanksgiving break, all efficiency data will be collected and compared to decide which is the preferred method for power conversion in the final product. Realistically, all of the power electronics should work. The more ambitious task is to add complexity to the power converters and not simply buy pre-designed parts.

The workload has been balanced among the three team members well. Each team member individually contributes similar amounts of work. For my personal contribution, I believe that I am on schedule and the amount of work I've put in has been sufficient for the timeline of this project.

This project follows the IEEE Code of Ethic[11]. Plagiarism will not occur in any form and if schematics or ideas are used from sources, the sources will be cited appropriately. The product also follows the Code of Ethics in that it should not cause harm to others.

The power electronics for this project is on course for completion by the Final Presentation deadline.

References

[1] BC2DA [Online]. Available:
http://www.mouser.com/ds/2/177/5c0060-67532.pdf

[2] 1N4004 [https://www.diodes.com/assets/Datasheets/ds28002.pdf]. Available:
https://gridconnect.com/bt832f-extended-range-Bluetooth-ble-module-mesh-network.html

[3] LM2596 [Online]. Available:
https://www.onsemi.com/pub/Collateral/LM2596-D.PDF

[4] TPS73033DBVR [Online]. Available:
https://www.onsemi.com/pub/Collateral/LM2596-D.PDF

[5] LM25141 [Online]. Available:
http://www.ti.com/product/LM25141/datasheet/abstract/SNVSAU14495

[6] MEMS microphones vs Electret microphones [Online]. Available:
https://ez.analog.com/thread/11410

[7] TI MSP432 Ultra Low Powered Microcontroller [Online]. Available:
http://www.ti.com/product/MSP432P401R/datasheet

[8] ESP32 WiFi & BLE module [Online]. Available:
https://www.seeedstudio.com/ESP-32S-Wifi-Bluetooth-Combo-Module-p-2706.html

[9] BMD 101 [Online]. Available:
http://m5.img.dxcdn.com/CDDriver/CD/sku.241178.pdf

[10] BT832F [Online]. Available:
https://gridconnect.com/bt832f-extended-range-Bluetooth-ble-module-mesh-network.html

[11] IEEE Code of Ethics [Online]. Available:
http://www.ieee.org/about/corporate/governance/p7-8.html