

KUMU A‘O CUBESAT: ELECTRICAL POWER SUBSYSTEM

Jordan S. Torres
Department of Electrical Engineering
University of Hawai‘i at Mānoa
Honolulu, HI 96822

ABSTRACT

The objective of the electrical power subsystem (EPS) of the Kumu A‘o CubeSat will be to receive, store, and distribute the power required by the satellite. Solar energy will be received by solar cells that will be placed around the perimeter of the CubeSat. This energy will be converted and stored using a single lithium-ion battery. The battery will be monitored using a fuel gauge that the command and data handling (C&DH) will communicate with. From the battery, power will be distributed to the respected subsystems of the satellite through the use of DC-DC converters. All components of the EPS will be selected to be able to withstand the space environment. The Fall 2007 semester was devoted to the design of the EPS while the Spring 2008 semester was spent learning the PCP software and prototyping of the EPS.

INTRODUCTION

The translation of “Kumu A‘o” is “source of learning”. The Kumu A‘o CubeSat will provide engineering students at the University of Hawai‘i of Manoa with the opportunity to get hands on experience with a satellite project from “the cradle to the grave”. The Kumu A‘o CubeSat is a LEO (Lower Earth Orbit) mission, meaning the orbit of the satellite will reach the altitude of about 2,000 km. The Kumu A‘o Cubesat team is composed of electrical and mechanical engineers from the University of Hawai‘i at Manoa. I am the EPS engineer with team lead, Matthew Patterson.

Prior to the fall 2007 semester, a requirements list was created by the previous CubeSat team. From the requirements list, the EPS established voltages and currents that will be required. With these parameters set, a tentative block diagram of the EPS was created. In October 2008, we were able to complete the preliminary design review (PDR) where each subsystem presented their requirements and possible designs for their subsystem. The first PDR that was given was incomplete and needed to be worked on. The EPS had several components that hadn’t been selected. By the second PDR, the EPS had made all component selections. With the second PDR, the EPS and the rest of the team would move on to the next stage of design.

The next step of the design process was the critical design review (CDR). At this design phase, every part of each subsystem should be finalized and ready for fabrication. The CDR was completed on March 2008. In the CDR, the finalized block diagram and the components of the EPS were proposed. In the CDR, the EPS introduced the expected power output factoring in rotation of the Cubesat and intensity from the sun while in space. During the time allocated for questions, key issues were brought up. One problem found was that one of our DC-DC converters didn’t provide enough current at the output. Another problem that was found was in the connections between the different components. Our presentation didn’t have a section that went over the specific connections between all the components.

DESIGN

At the end of the fall 2007 semester, the design of the EPS's block diagram was finalized. The finalized block diagram of the EPS can be found in Figure 1.

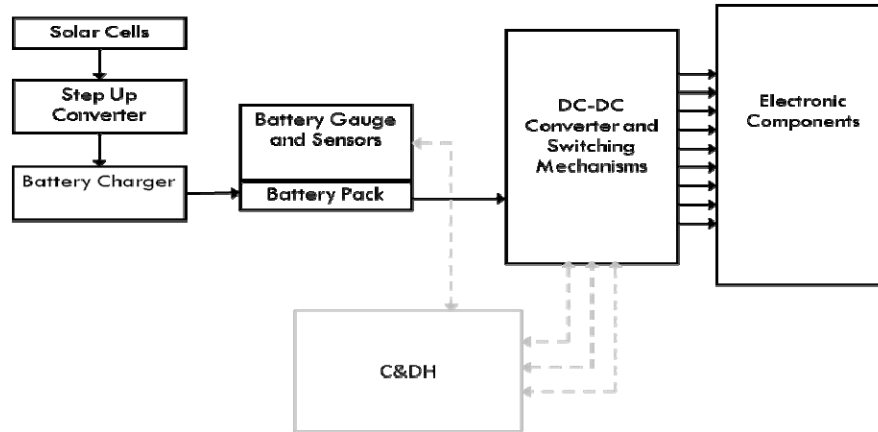


Figure 1: EPS Basic Block Diagram

Solar cells (COMPLETE)

The solar cells we will be using are Spectrolab's Improved Triple Junction (ITJ) solar cells. Spectrolab produces solar cells with higher efficiency, but due to high prices, Spectrolab advised for the use of ITJ cells that have a 26.8% efficiency. Each cell has an output of 2.23 V at 450.8 mA. Each side of the CubeSat will have two solar cells connected in series as shown in Figure 2.

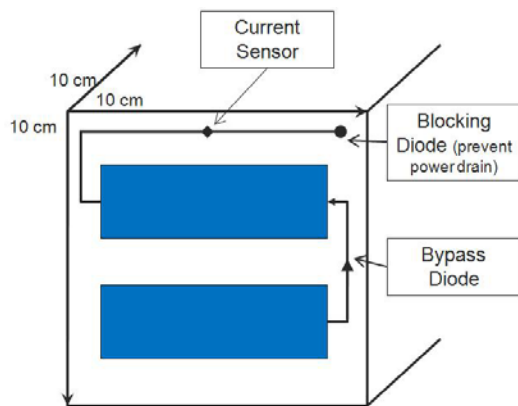


Figure 2: Placement of Solar



Figure 3: Soldered Practice Solar

The reason for this design is to acquire the most efficient output of voltage and current. Diodes will be placed at the output of each cell to avoid degradation from current going the wrong way. The output voltage from each side of the Cubesat will be 4.46 V with a current of 450.8 mA. Each side of the satellite will also have a current sensor which will provide a way to determine the orientation of the satellite. After acquiring a set of practice solar cells, we were able to test the soldering of the solar cells to the PCB. Figure 3 shows a soldered practice solar cell. The cell was soldered to the copper layer by tinning the bottom layer of the solar cell and the copper layer, then using a heat gun to solder the two components together.

Battery charger (COMPLETE)

The battery charger that will be used is the MAX1898. The key features of this chip are the charging current and its compact size. The MAX1898 was chosen due to its output current capability of 1.4 A. The prototyped circuit MAX1898 battery charger in Figure 5 was assembled according to Figure 4.

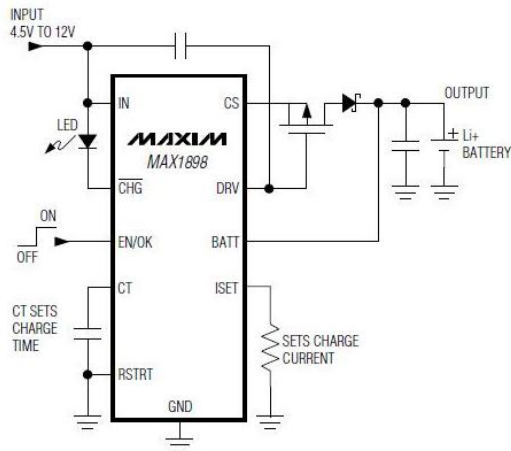


Figure 4: MAX1898 Circuit

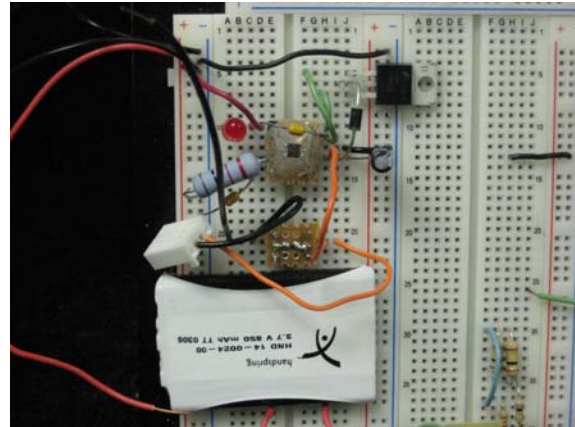


Figure 5: Prototyped MAX1898 Circuit

The only component that needed to be solved for was the output charge current. The equation used was: $I_{FASTCHG} = 1400 \text{ V} / R_{SET}$. Since an output current of 1.4 A is desired, R_{SET} will be 1 k Ω . The battery used for testing is a lithium polymer composite taken from a PDA. When testing the circuit, we found that it gave off the correct output, but when the board was tapped, the charger shut off. When this was found out, we figured out that the charger goes into a fault mode when the load voltage drops more than 200 mV from the way that we designed the circuit. This was resolved by floating the RSTRT pin. By floating the RSTRT pin, the fault mode is disabled. At an initial voltage of 2.74 V, the battery was charged at 888 mA.

Battery (COMPLETE)

The battery selected for this mission is the Saft MP176065. The Saft MP176065 is a single lithium-ion battery and was chosen for its capacity of 7 Ah and energy density of 26.25 Wh. The Saft battery has a mass of 146 g, which is within the allocated mass requirements set for the battery. The output voltage of the battery ranges between 2.5 V and 4.2 V, having a nominal voltage of 3.7 V.

Fuel gauge (COMPLETE)

The fuel gauge chosen for the monitoring the battery is the Texas Instruments' BQ2060 gas gauge. The initial reason for choosing this fuel gauge was for its capabilities specified for the lithium ion battery. The BQ2060 was created with the PCB software. In doing so, we saw how much room the circuit's components took up. The components of the BQ chip took up about three quarters of the PCB. Due to the lack of space on the PCB, we were forced to select another fuel gauge. The replacement fuel gauge is the BQ27200, which has many of the same functions as the BQ2060 with less external components.

DC-DC Converters (INCOMPLETE)

The DC-DC converters along with the battery charger were tested through prototyping during the spring 2008 semester. Each chip was tested individually before integrating the chips with the rest of the system. This method was used to avoid any inconsistencies of the data sheets and the component selection.

The first chip assembled was the **MAX1771** chip. The MAX1771 will be used as the adjustable output DC-DC step-up converter. One MAX1771 chip will be designated as 6 V for the Yaesu transceiver. Six other MAX1771 chips will be used to step up the voltage coming out of each side of the satellite to 7 V. The 7 V output of each converter will be the input to the battery charger. The first step in the circuit design was determining the values of all the components. The equation used to set the values of resistors R1 and R2 in Figure 5 is:

$$R2 = R1 \left(\frac{V_{out}}{V_{ref}} - 1 \right). \quad V_{ref} \text{ is set at } 1.5 \text{ V and } V_{out} \text{ is set to } 6 \text{ V. With these constants, we solved}$$

for R1 and R2 and found that $R2=3*R1$. We set $R1=10k\Omega$ and $R2=30k\Omega$. Once all the values of the components were acquired, prototyping of the breadboard was initiated. Figure 6 shows two MAX1771 chips from Figure 5 prototyped on a breadboard.

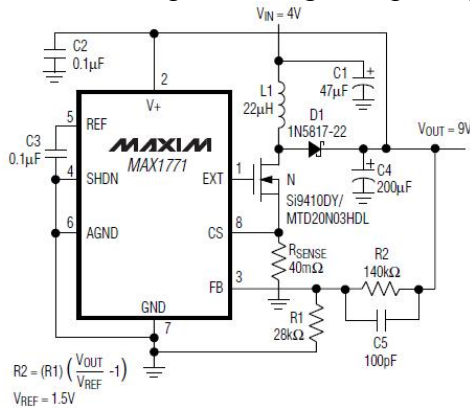


Figure 5: MAX1771 Circuit

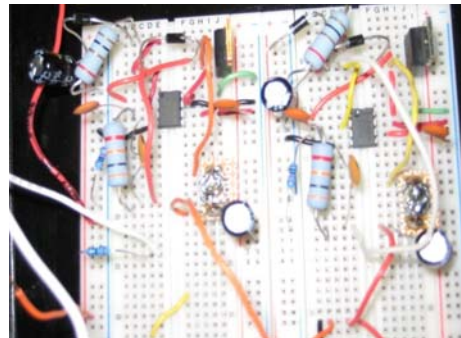


Figure 6: Prototyping of 2 MAX1771 Chips

After prototyping the schematic of Figure 5, it was found that the output of the circuit was oscillating. To solve this issue, a 100 Ω resistor was placed in parallel with the output capacitor (C4 in Figure 5). This shunt resistor stabilized the output reading. Figure 6 shows the final breadboard circuit design of the two adjacent MAX1771 chips. Once the output was stabilized, the voltage and current reading were taken. After prototyping three different MAX1771 chips, it was found that each chip had different output characteristics. The outputs of each of the chips can be found in Table 1. The output currents were in the range of 600 mA, which are insufficient.

Input Voltage	Theoretical Output Voltage	Output Voltage - Chip1	Output Voltage- Chip 2	Output Voltage- Chip 3	Placement of Test Probe
3 V	6 V	6.254 V	6.306 V	2.844 V	B4 Diode
3 V	6 V	6.741 V	6.504 V	3.271 V	Aft Diode
3.30 V	6 V	6.254 V	6.316 V	3.013 V	B4 Diode
3.30 V	6 V	6.67 V	6.467 V	2.994 V	Aft Diode

Table 1: MAX1771 Output Voltages

The MAX1771 chip has posed many problems for the EPS. The next step is to make solid connections between components using a protoboard to avoid fluctuations. If that is unsuccessful, a new chip will be used.

The MAX1703 will be used as the fixed DC-DC step-up converter for the terminal node controller (TNC). The TNC requires 5 V at with a current of 40 mA. The circuit in Figure 8 was assembled according to the circuit in Figure 7.

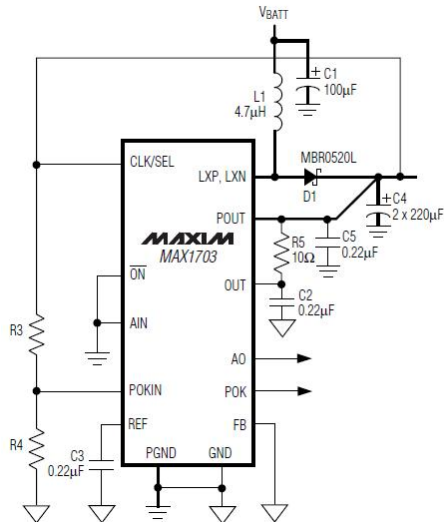


Figure 7: MAX1703 Circuit

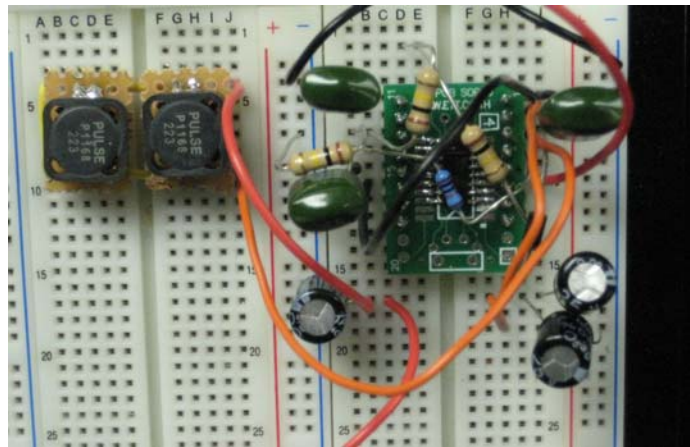


Figure 8: Prototyping of MAX1703 Circuit

The resistors R3 and R4 in Figure 7 were determined using the equation $R3=R4(V_{TH}/V_{REF} - 1)$. The datasheet specified V_{REF} equal to 1.25 V, V_{TH} equal to 3.75 V, and R4 to be less than or equal to 270 kΩ. With these specifications, R4 was set to 100 kΩ and R3 was set to 200 kΩ. When tested, the MAX1703 did not provide the desired output. The battery that was used for the testing of the MAX1898 battery charger was used as the input of the MAX1703 circuit. When connected, the output was nowhere near what is needed. This complication may be due to the use or wrong components. However, a new converter is being looked at currently.

The MAX1921 will be used as the DC-DC step-down converter for the flight module. The flight module requires 3.3 V with a current of 40 mA. Since the MAX1921 has a set output voltage of 3.3 V, the circuit design was simple. The components of the circuit in Figure 10 were selected from a table given in the chips datasheet.

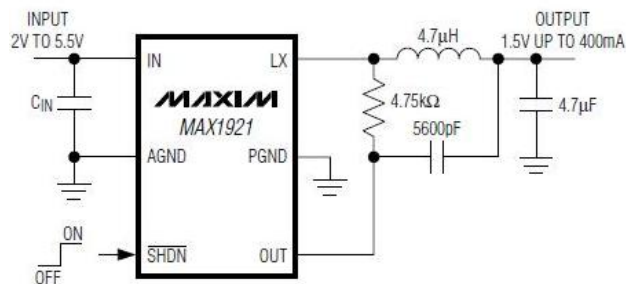


Figure 9: MAX1921 Circuit

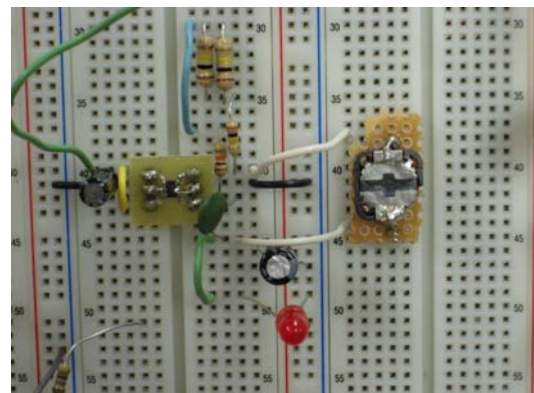


Figure 10: Prototyping of MAX1921

Although all the components were assigned, the circuit didn't produce the desired output. Leaving all other components the same, we adjusted the resistor until the correct output was acquired. We found that using a 360 k Ω , instead of the 4.75 k Ω resistor in Figure 9, gave us an output of 3.269 V with a current of 327 mA. The 1.07 W that was produced is sufficient power for the flight module.

PCB SOFTWARE

The first half of the spring 2008 semester was devoted to the learning of the PCB software. The software chosen for the PCB layout is Advanced Circuit's PCB Artist. Through the PCB software, we were able to plot the components of the EPS on a PCB. The first step in using the software is to create the chip. The library of PCB Artist was found to be limited to what chips were available so the creation of the specific chip was required. Once the chip was created, the creation of the circuit's schematic was done. The final step in the schematic design was placing the schematic on the PC104 template which was created by teammate Jeremy Chan. This template had the holes and the PC104 pins placed on the PCB. Once this was completed, the schematic was converted to PCB by software. This stage of the design process gave problems because the software was not able to create all the tracings that were required. even with a four layer board, the software wasn't able to make all the connections.

CONCLUSION

After making minor adjustments to the EPS, the Kumu A'o CubeSat team was able to complete both the PDR and the CDR in the Fall 2007 semester. The Spring 2008 semester was dedicated to learning the PCB software along with the prototyping of the EPS. Through the utilization of the PCB software, it was apparent that not everything can be done with the software alone. Also, we were shown how important prototyping is to the validation of a datasheet's circuit design. Prototyping gave us a chance to expose the problems that were present in the current circuit designs. By recognizing the problems, we can make adjustments now, and avoid problems later in fabrication.

ACKNOWLEDGEMENTS

I would like to thank the Hawai'i Space Grant Consortium for granting me the opportunity to participate in such an amazing program. I would also like to thank all of the Kumu A'o Cubesat team for being a cohesive unit. Also, thank you to mentors Lloyd French and Byron Wolfe who gave valuable advice throughout the semesters.

REFERENCES CITED

Maxim-ic Website: DC-DC Converters

Available [Online]: <http://www.maxim-ic.com/index.cfm?CMP=KNC-8AO766069986>

Saft Batteries Website: Li-ion Battery

Available [Online]: <http://www.saftbatteries.com/>

Spectrolab Website: Improved Triple Junction Solar Cells

Available [Online]: <http://www.spectrolab.com/prd/space/cell-main.asp>

Texas Instruments Website: Fuel Gauge

Available [Online]: <http://www.ti.com/>