AGU ELECTRICAL-ELECTRONICS ENGINEERING SUMMER INTERNSHIP

FINAL REPORT

by

Student Name: Ahmet Yasir Cilvez

Student ID Number: 110110168

Company Name: TÜBİTAK Uzay Teknolojileri Araştırmaları Enstitüsü

Internship Dates: 02.06.2021 – 10.09.2021

Date of the Submission: 10.09.2021

SUMMARY

In the internship at TUBİTAK UZAY, first an over and under voltage protection circuit schematic and PCB design were created with using LTSpice and Altium Designer, second a pre-made BLDC motor driver schematic's PCB design were created with Altium Designer. During these processes all of simulations were made in LTSpice and PCB designs were built in Altium Designer. As the results of the internship period, one schematic and two different PCB designs were finished successfully.

1. INTRODUCTION

TÜBİTAK Space Technologies Research Institute (TÜBİTAK UZAY) is a sub-part of TÜBİTAK and they are connected with directly Turkey Ministry of Industry and Technology. TÜBİTAK UZAY was founded in 1985 to improve Turkey's capability on space technologies [1]. The Institute is working on mostly aerospace technologies, satellite technologies, remote sensing and data processing, communication technologies. They are the main contractor to develop next generation Turkish communication satellites for TURKSAT, known as TURKSAT 6A [1]. TUBİTAK UZAY has some groups in it that work on different parts of satellite and aerospace systems. The department of Platform and Payload Electronic Design that I did my internship develops the satellite's hardware which include all payloads and subparts of the systems.

2. PROJECT

A. VOLTAGE PROTECTION SCHEMATIC AND PCB DESIGN

According to requirements, under and overvoltage protection circuitry was created. The aim of this circuit was allowing to flowing current if the source voltage is between 7V and 12V even though the source gives a voltage of 5V-20V. The working principal diagram is shown in Figure 1. During this process, circuitry and PCB were created considering the following parameters: simple and applicable, reproducible, using components available in the market, low cost, low energy consumption, and high performance.

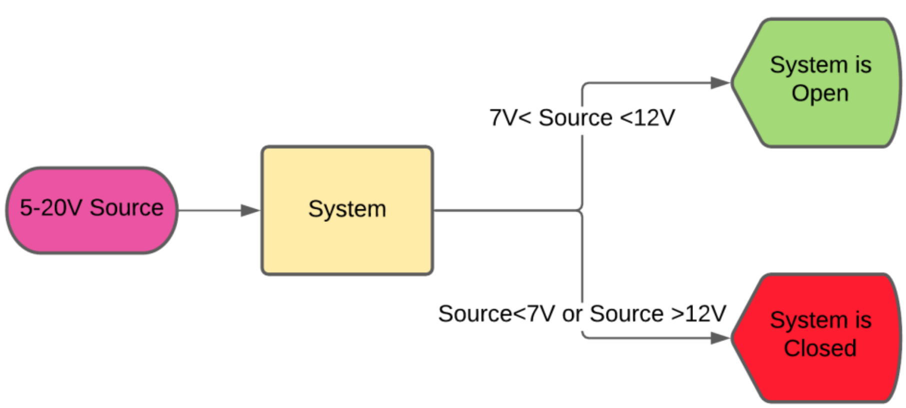


Figure 1 - The working principal diagram

A.1. Schematic Design

The schematic design includes subparts on it. All of the parts are explained in this section. The schematic circuitry diagram is shown below.

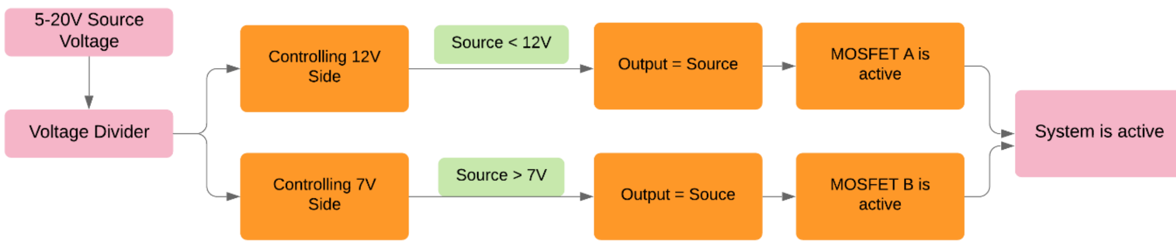


Figure 2 - The flowchart of the schematic circuitry

Window Comparator

Due to the fact that the system should check two boundary voltages which are 7V and 12V, there should be a dual comparator. The best and efficient way to compare a voltage with two voltages at the same time is using a window comparator circuitry design. One of the solo comparator diagrams is shown below.

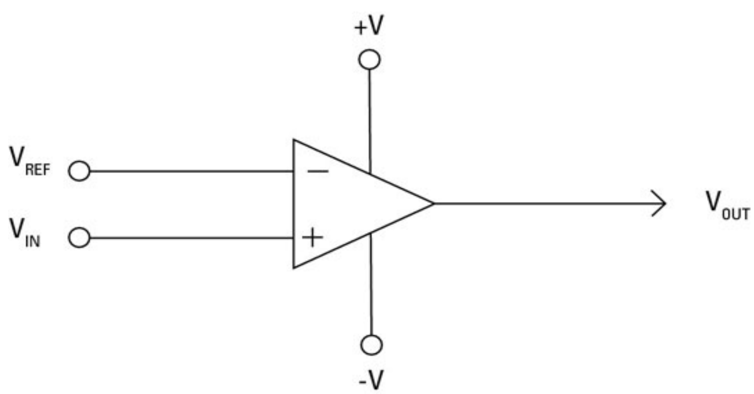


Figure 3 – A comparator schematic

To compare voltages, op-amps that are produced for voltage comparison are used generally. As shown in figure 3, the op-amp includes 2 input voltages, depends on the connection of them to + or – input pin of the component, the output voltage can be controlled. In detail, if the voltage that is connected to the positive input pin of the component is higher than connected to the negative input pin, the output voltage (Vout) is equal to nearly source voltage.

A window comparator is called when two comparison process are combined. The typical window comparator action diagram is shown below.

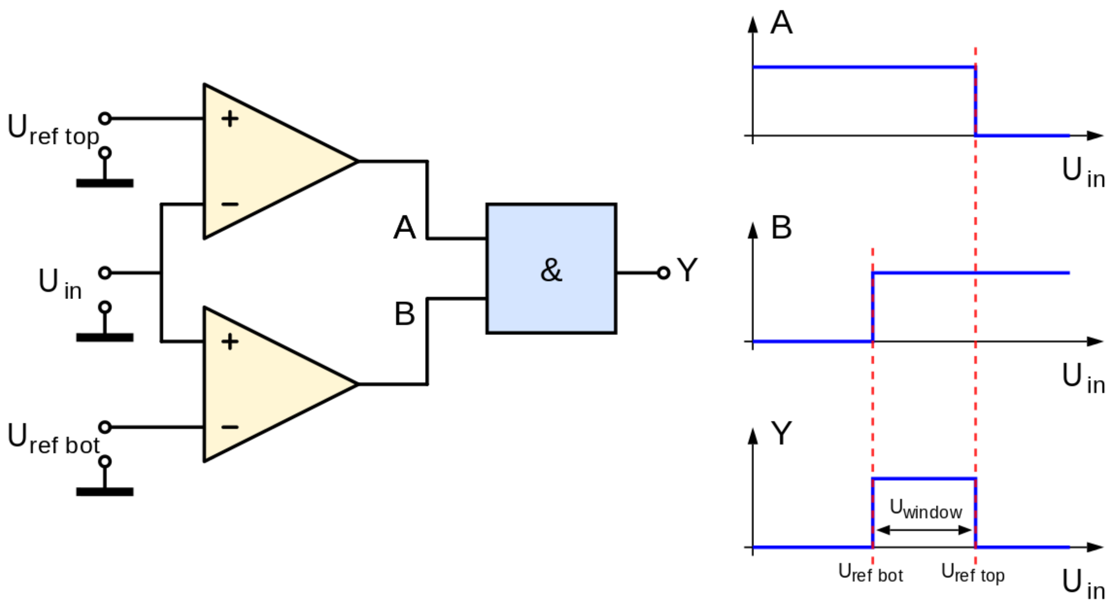


Figure 4 – Window comparator circuitry diagram and output graphs

As shown in figure 4 source voltage is connected as input reference (Uin) to both comparator reference inputs. However, this reference voltage has to be connected to reverse pins of two op-amps. In this way, one of them is controlling if the source is higher than the low boundary, other one controls if the source is lower than the high boundary voltage. A window comparator is used with commonly an AND gate at the output side of the circuitry.

Figure 5 - LT1017

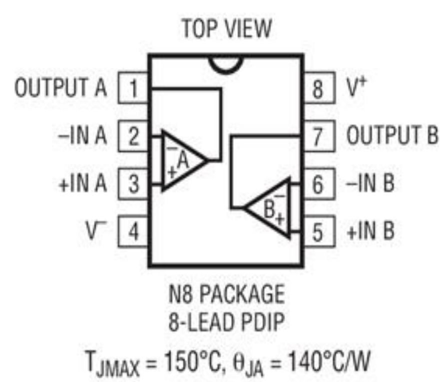


Figure 6 - Pinout of LT1017

LT1017 Micropower Comparator is selected and used for voltage comparison. The reasons for this selection are listed below.

- Low power consumption.
- Single supply sourcing 1.1V – 40V
- Too close output voltage to source voltage
- Having dual comparator inputs
- Available in the marketplace
- Large operation range temperature between -55 C and +125 C degrees
- Expected simulation results
- Available Spice model

Arranging Reference Voltage Values

Normally, the low-level boundary reference voltage of the op-amp should be equal to 7V when the high-level boundary reference voltage equals 12V. However, op-amps cannot be well driven in the condition of their supply voltages are equal to compared voltages. Due to low and high reference voltages that should be arranged to comparison truly, some changes were made. Firstly, the source-reference voltage is divided with a voltage divider because of reducing it from the supply voltage. The voltage divider structure is shown below.

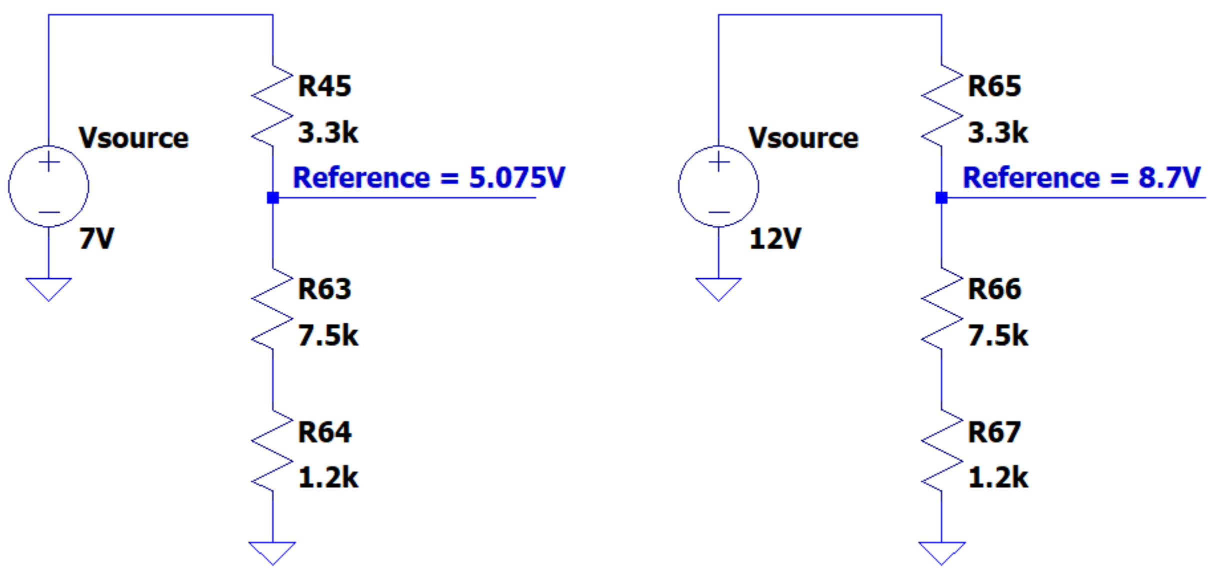


Figure 7 – Source-Reference voltage divider circuitry

As the output of the voltage divider circuitry, 7V source-reference is reduced 5.075V and 12V reference is arranged at 8.7V during op-amp's supply voltages are equal to source voltage. In this way, at least 1.925V difference between source and source-reference is generated.

Rearranging The High and Low Boundaries Input References

Figure 8 - 1N5238

Figure 9 - D1N4733

According to new low- and high-level boundaries, reference inputs of the comparator are rearranged. To provide stable reference voltages at the low and high boundaries reference input of the op-amp, Zener diodes are selected according to new reference voltages which are 5.075V and 8.7V.

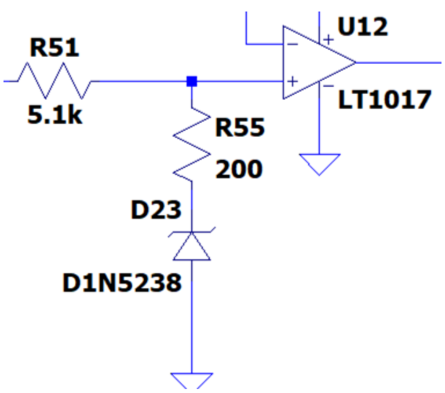


Figure 10 – High boundary reference input

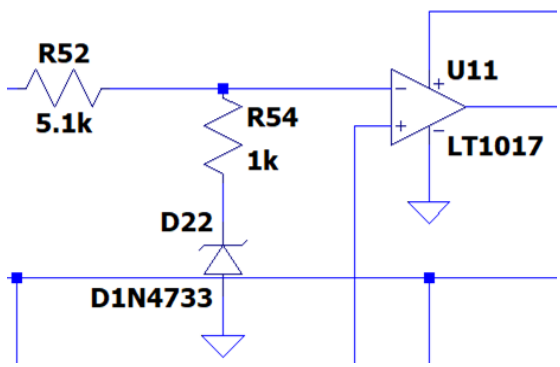


Figure 11 – Low boundary reference input

1N5238 for 8.7V and 1N4733 for 5.1V Zener diodes are selected to stabilize low and high reference voltages. Because of the 5.1K resistor that is used to reduce current on the pin, Zener diodes voltage drops are reduced by approx. 0.4V. That's why serial resistors were added to the Zener diodes voltage lines. In this way, leakage currents of Zener are used to generate a voltage drop on these additional resistors. As a result of this process, high and low voltage reference pins voltages are stabilized with changing resistor values to catch high or low-level voltage reference values.

In Tables 1 and 2, the relationship between source voltage, source-reference voltage, the voltage at the low and high boundary reference pins are shown. To test the system and get these results, near voltage values to low and high boundaries are given to the system as source voltage. As shown in Table 1, when the source voltage is 6.9V, the low boundary voltage is higher than the source-reference voltage, therefore the output of the op-amp is low. Unlike, while source voltage reaches 7V, source-reference voltage is higher than low boundary, therefore op-amp starts to give source voltage as output.

Source Voltage	Source-Reference Voltage	Low Boundry	High Boundry
6.9V	5.002V	5.02V	6.9V
7V	5.075V	5.042V	7V
7.1V	5.148V	5.065V	7.1V

Table 1 – Voltage values on some important points due to the source voltage

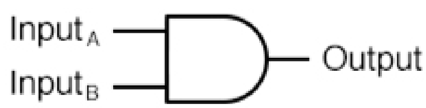
Source Voltage	Source-Reference Voltage	Low Boundry	High Boundary
11.9V	8.627V	6.007V	8.719V
12V	8.7V	6.025V	8.723V
12.1V	8.772V	6.043V	8.727V

Table 2 - Voltage values on some important points due to the source voltage

On the other hand, reactions of the high boundary controlling side voltages are obtained in Table 2. As shown there, when 12.1V is given as a source voltage, source-reference voltage is higher than high boundary voltage at the op-amp's pin. In this way, the output of the op-amp equals 0 voltage. Unlike, while 12V is the source voltage, high boundary voltage is higher than the source-reference voltage and the output voltage of this comparison equals to source voltage.

And Gate Structure

Since there are two output voltages coming from two comparison which are high and low boundaries, there should be an AND gate to control whole circuitry if source voltage is in the low and high limits. Because of the AND gate truth table shown in figure 12, the circuitry is active only both comparison's outputs are high at the same time.



A	B	Output
0	0	0
0	1	0
1	0	0
1	1	1

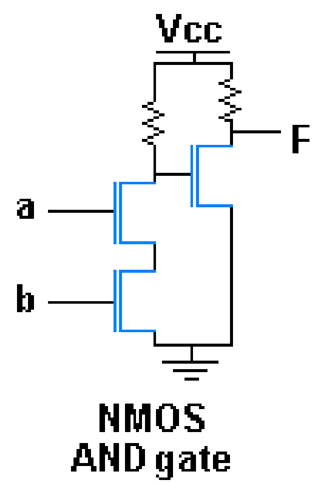


Figure 12 – AND gate representation and its truth table

Figure 13 – NMOS AND gate circuitry

A typical AND gate application with NMOSs is also shown in figure 13. The main disadvantage of this circuitry, except both NMOSs (a and b) are activated, there is a current flow on the output NMOS and power consumption is occurred (Figure 14). In addition, AND gate with BJTs (Figure 15) was set but there is a voltage drop with 1.5V from source voltage at the output line and there was not enough current on this line. That's why, AND gate with 2 NMOS structures is preferred in this project. In addition, NMOSs' voltage drops were quite less around 0.5mV per NMOS. The AND gate circuitry is shown in Figure 16.

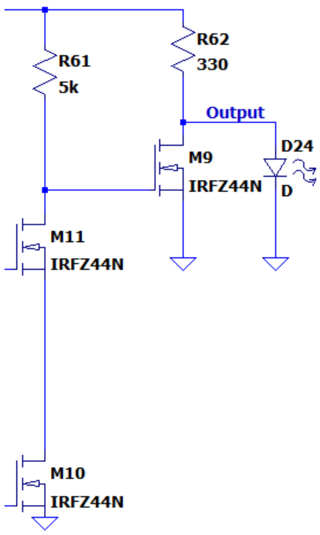


Figure 14 – AND gate with 3 NMOSs

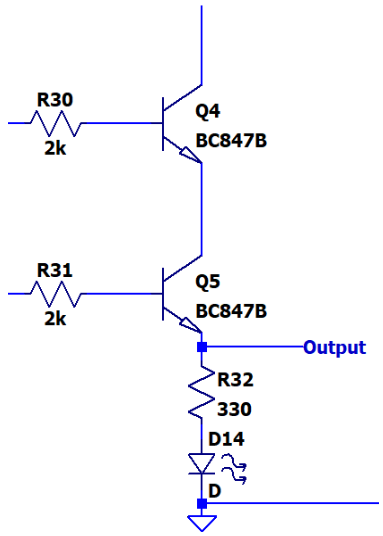


Figure 15 – AND gate with 3 BJTs

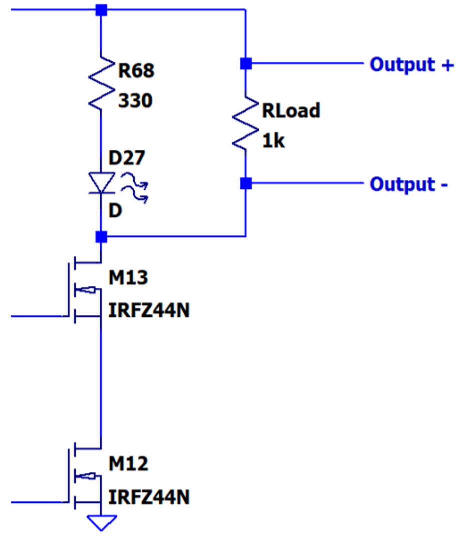


Figure 16 – AND gate with 2 NMOSs

In the last version of the AND gate circuitry, R68 and D27 LED are added for representing if circuitry is active or not and RLoad represents to rest of the circuitry that can be plugged to the voltage protection system.

Figure 17 - IRFZ44N

For the selection of the NMOS model that is available in the marketplaces, the following criteria are important: using for switching, enough current and voltage capacity, a wide range of operation temperature capability. Considering these parameters IRFZ44N MOSFET was preferred. IRFZ44N's some important features can be listed as:

- 60V drain to source capacity
- 48A current flowing capacity
- Usage in fast switching applications
- Low price and high accessibility
- -55 C to +125 C degree operating temperature

Whole Circuitry

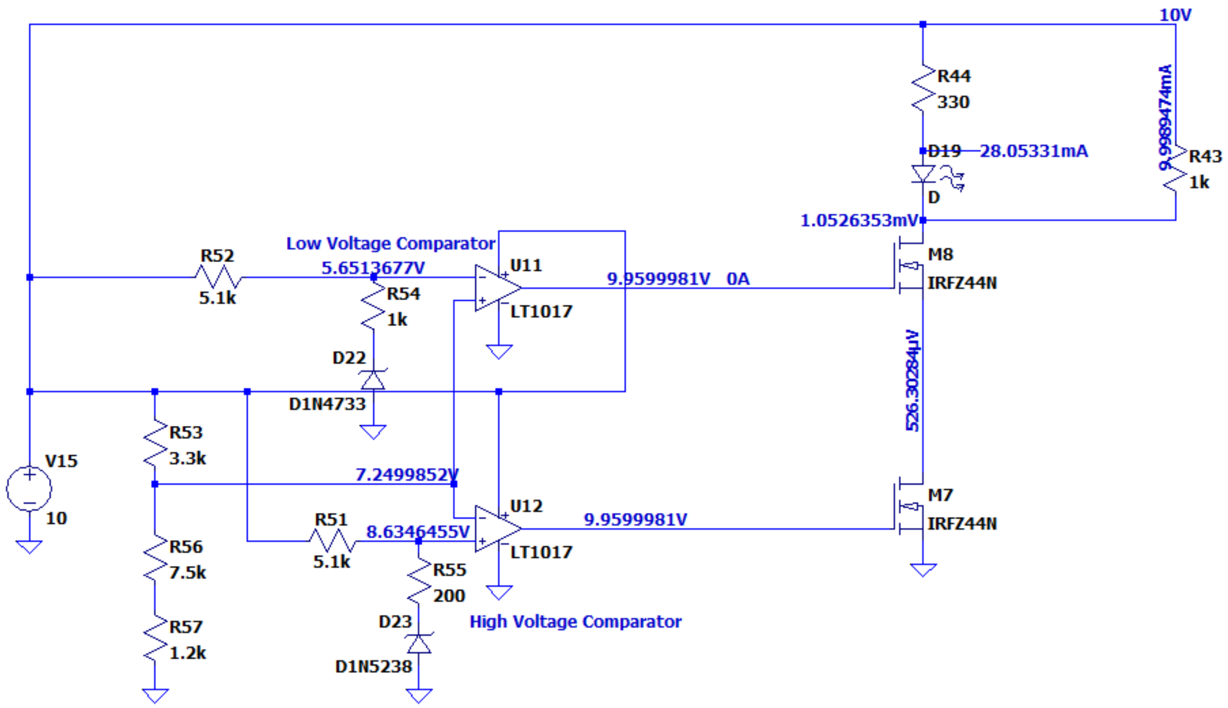


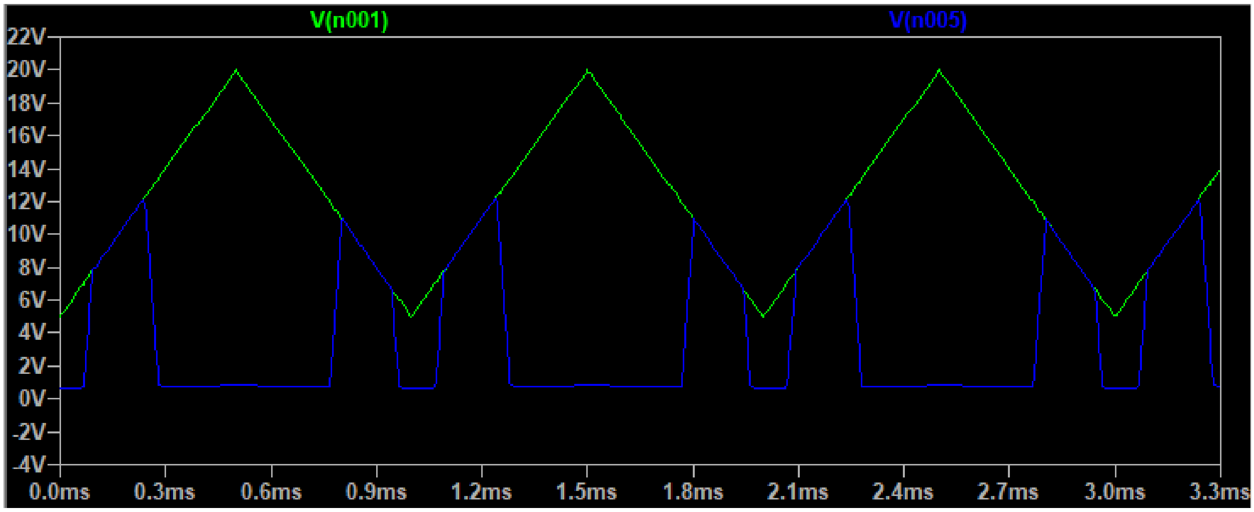
Figure 18 - Whole Circuitry

A.2. Simulation Results

Schematic simulation results coming from LTSpice are shown in this section.

Source Voltage vs. Output of The Hole Circuitry

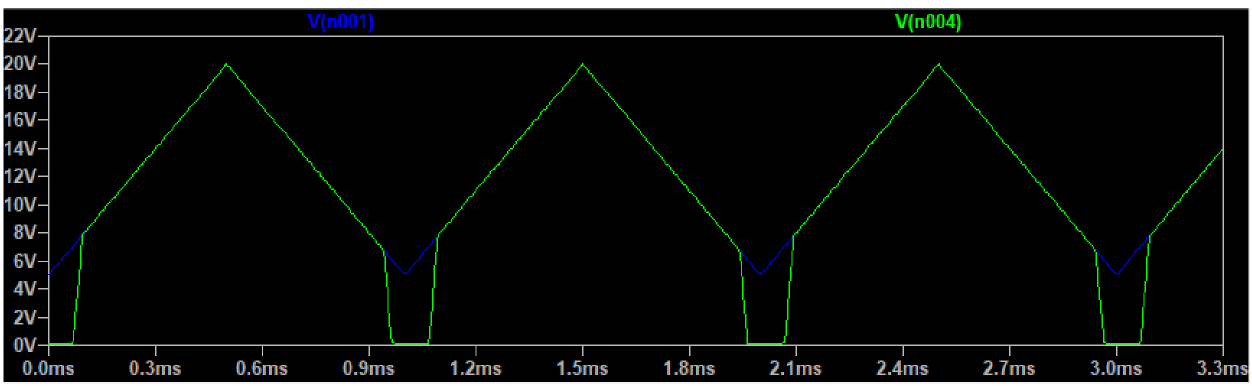
The relationship between the 5-20V source voltage and circuity switching behavior is shown in graph 1. As seen in this graph, the output voltage (blue line) of the circuitry is equal to the source voltage (green line) when the source voltage is in the 7-12V interval. Otherwise, the output voltage reduces to 0.



Graph 1 - Source Voltage vs. Output of The Hole Circuitry

Source Voltage vs. Low Boundary Controlling Output Voltage

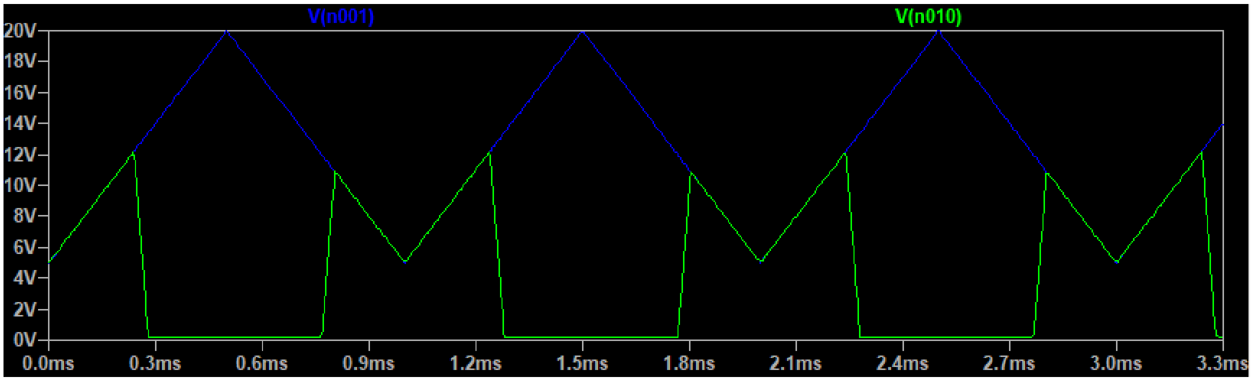
The graph of source and low boundary controlling voltage is shown below. As a result of this simulation graph, after the source voltage (blue line) reaches 7V and keeps increases, the output voltage (green line) equals to source voltage. Likewise, from 20V to 7V output voltage equals to source voltage.



Graph 2 - Source Voltage vs. Low Boundary Controlling Output Voltage

Source Voltage vs. High Boundary Controlling Output Voltage

The graph of source and high boundary controlling voltage is shown in Graph 3. As a result of this graph, after the source voltage is greater than 12 V and keeps increases, output voltage (green line) equals 0. Unlike, when the source voltage is less than 12V, output voltage equals to source voltage.



Graph 3 - Source Voltage vs. High Boundary Controlling Output Voltage

A.3. Printed Circuit Board (PCB) Design

Altium Designer is used for designing PCB of the circuitry. Steps are in below were followed while PCB is created.

1. Libraries and footprints of the components are detecting or generating in Altium Designer
2. Importing schematic components to PCB environment
3. Routing the connections
4. Adding Test Point's to PCB
5. Adding polygons
6. Adding vias to connect layers with each other
7. Arranging texts that including components' names
8. Reshape PCB's dimensions
9. Adding screw holes to mounting

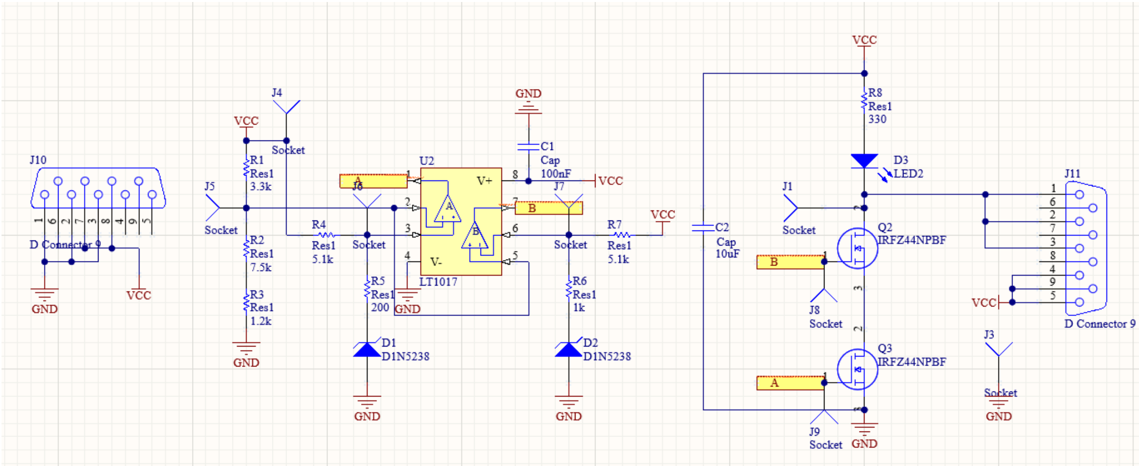


Figure 19 – Altium Designer Schematic

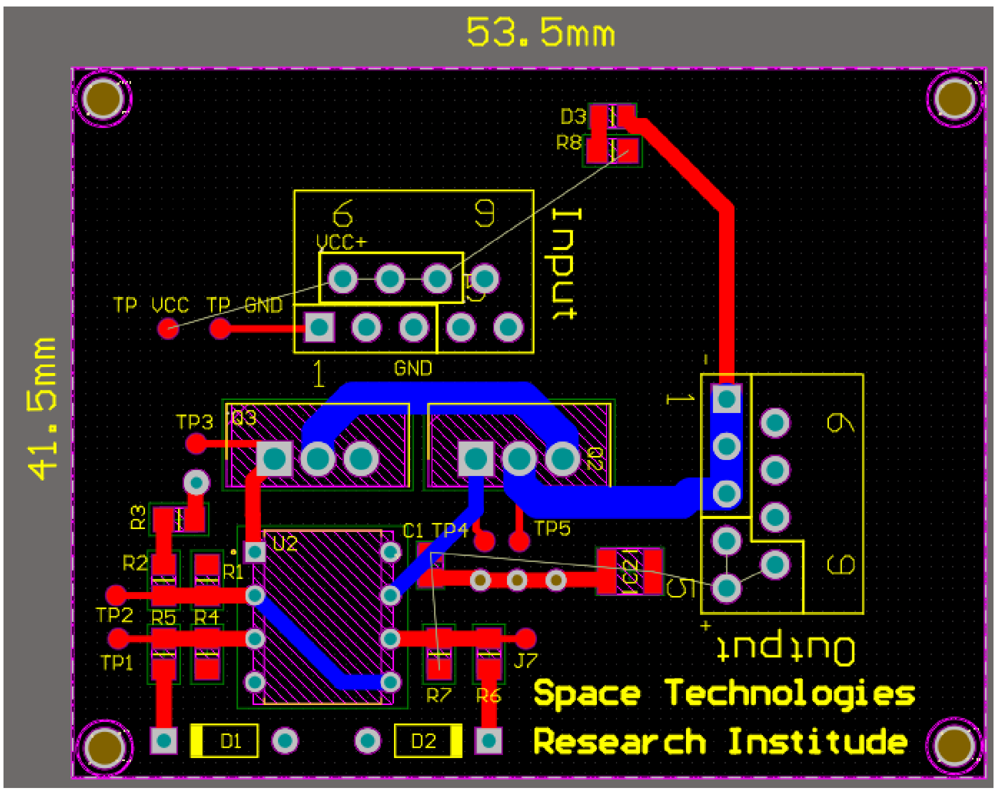


Figure 20 – PCB Design without polygons

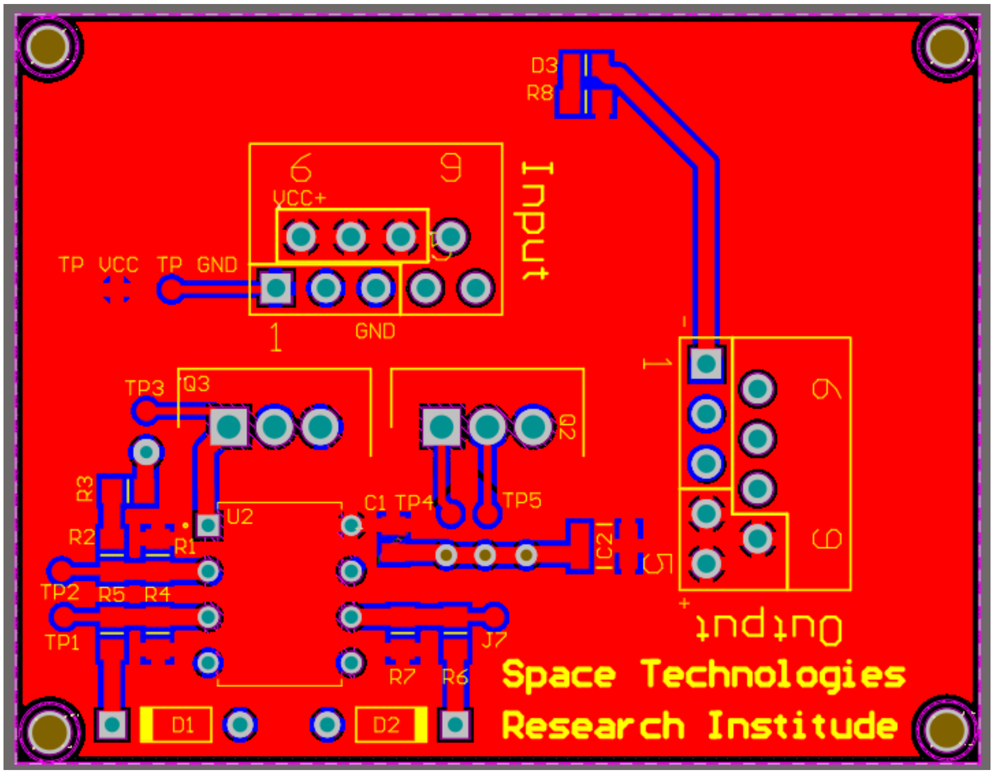


Figure 21 – PCB design with polygons

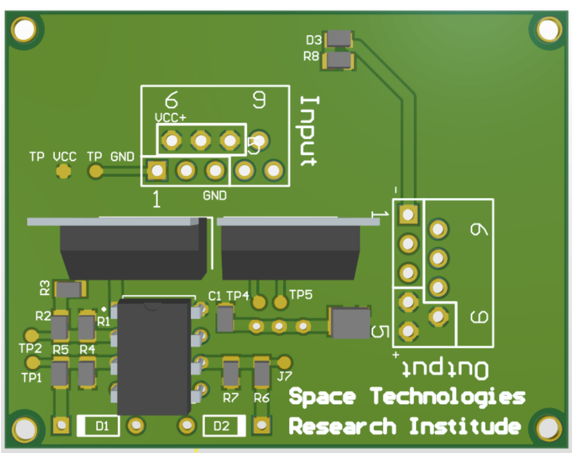


Figure 22 – PCB 3Dd perspective

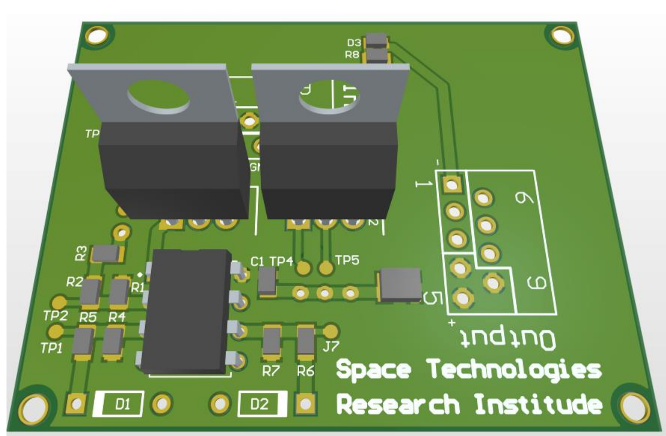


Figure 23 - PCB 3Dd perspective

PCB Current Flowing Capacity

The routes having the main load are determined as 2mm which can carry 4A and other routes determined as 1mm that can carry 2A with a max 10 C degree increase. The table of calculations is shown below.

Inputs:

Current	4	Amps
Thickness	1	oz/ft^2

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	208	mil
Resistance	0.00242	Ohms
Voltage Drop	0.00970	Volts
Power Loss	0.0388	Watts

Results for External Layers in Air:

Required Trace Width	2.03	mm

Table 3 - PCB current flowing capacity calculation

Cost of The PCB

To calculate the cost of the whole system, used components' prices were taken from Digikey and Özdisan. PCB board price was taken from JLCPCB. The price list and total cost are shown in Table 4.

Name	Unit Price $	Pieces	Cost
LT1017	8.17	2	16.34
IRFZ44	0.43	2	0.86
1N5238	0.16	1	0.16
1N4733	0.21	1	0.21
Resistors	0.018	8	0.144
100nF Cap	0.24	1	0.24
10uF Cap	0.87	1	0.87
2 Layers PCB	0.4	1	0.4
LED	0.024	1	0.024
		Total in $	19.248
		Total in TL	161.3

Table 4 – Cost of the PCB

Possible Improvements

At the end of the project, a presentation is organized by The Platform and Payload Electronic Design group and the projects were presented by internship students. Due to the feedback from the group, to more stabilize the voltage on the high and low boundary reference inputs, the location of Zener diodes can be chanced and they can connect as parallel with the resistor serial before. In this way, the reference pin's voltages can't increase as shown in Tables 1 and 2. Also, this change allows the usage of different kinds of op-amps because if voltages on the boundary inputs increase to a certain value, the op-amp cannot be driven healthy. Another possible improvement can be made in AND gate structure. The AND gate structure switches the GND side of the output and the load or rest of the circuit connected with the voltage source every time. If there is an unwanted event occurs and the load's GND pin touches any GND on the board, there can be overvoltage harming. It can be seen that the circuit works but it can be improved according to feedbacks from The Platform and Payload Electronic Design group.

B. BLDC MOTOR DRIVER PCB DESIGN

The second project was creating a PCB design for an existing schematic of a BLDC motor driver [2]. Altium Designer was used in this project.

BLDC (brushless DC motor) is one of most common types of motor using in sector because it use less energy and give high efficiency rates. That's why, drawing a BLDC motor driver PCB is preffered.

The same PCB drawing process in the project 1 is also valid for this project. The main difference is the routing and drawing polygon policy because motor driver should conduct more current than other PCB's. The PCB design is shown below.

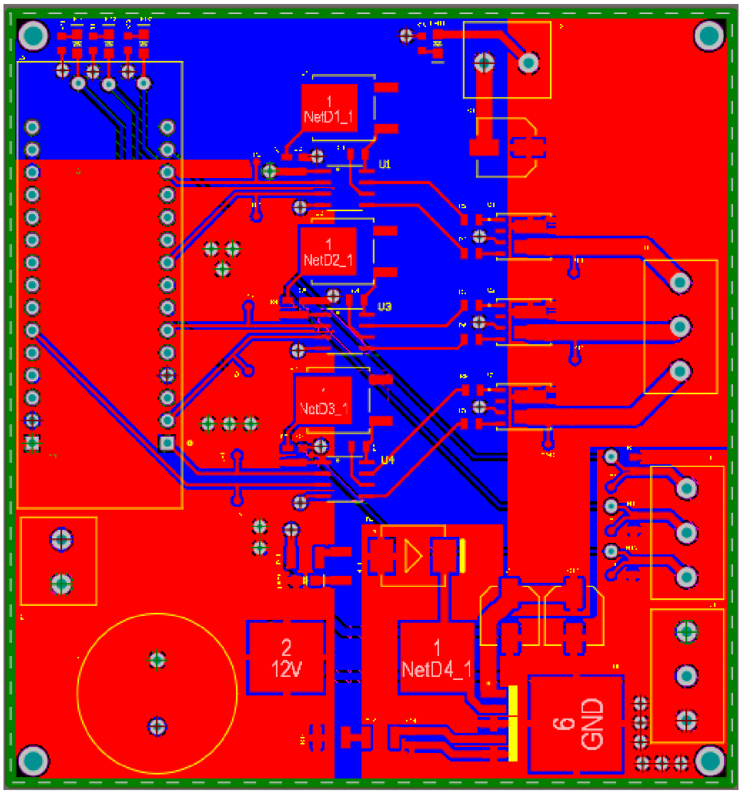


Figure 24 - Top seen of the PCB

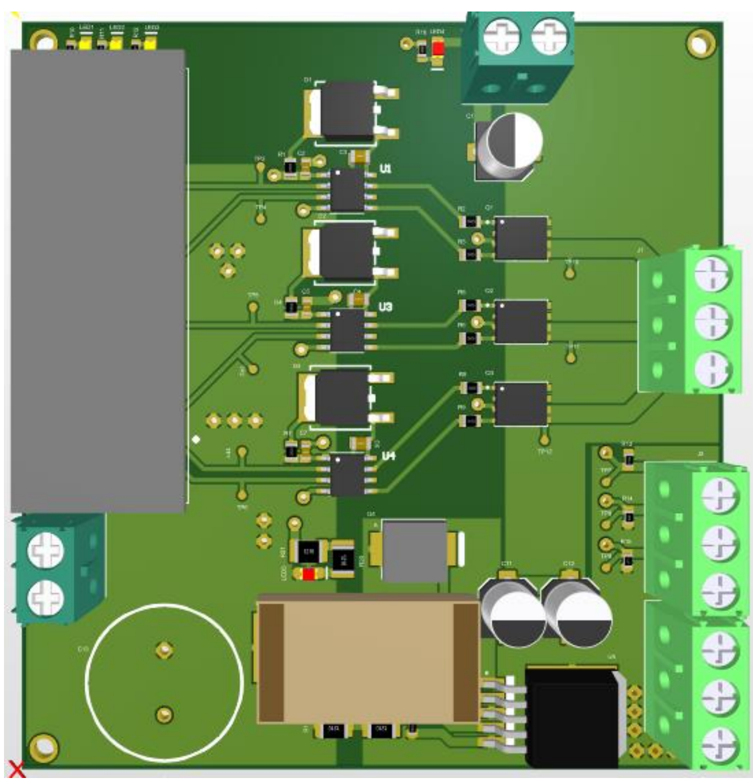


Figure 25 - 3D of the PCB

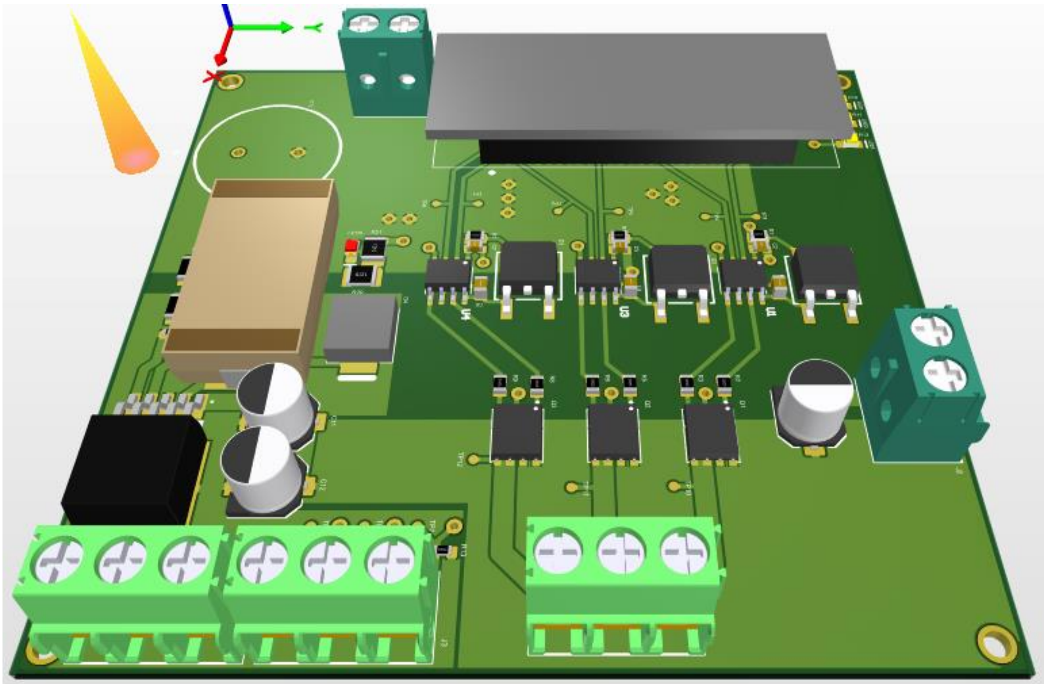


Figure 26 - 3D of the PCB

Similarities of the Internship Work and Faculty Courses

The first project was an R and D process. It starts from researching the needed system and continues drawing schematics, finding components, building PCB, and analysis of it. In this way, the project work is directly connected with Electronics 1 because of using MOSFETS and diodes, Electronics 2 because of BJTs, Circuit 1, and Circuit 2 because of Opamps, resistors, and capacitors. Also, circuit analysis is included in these whole courses.

Differences of the Internship Work and Faculty Courses

The project process is a little different from faculty courses. The possible reason for this is calculations and all simulations were done in mostly the computer environment, instead of hand calculations. Also, dealing with the real components is different from using ideal components in courses.

Current Challenges

Because the institute area is space technologies, the resources are limited and R and D works must be not much different than previous tested and experienced works. Also, there is not enough work in this area in Turkey. A space technology environment should exist very soon in

Turkey. In addition, the institute can increase their test and lab department to more fast testing and developing the circuitries.

3. CONCLUSIONS

To conclusion, from the first step to last step in the first project, I experienced how to approach an engineering problem and solve it in a logical way. In detail, I searched about the purposed system to understand previous works, I build the circuit to convert theory to simulation, then draw the PCB in Altium and learned lots of new information about LTSpice and Altium. Also, I have a chance to show all my effort and works with the Platform and Payload Electronic Design group by a presentation. They gave very important feedbacks and I also worked on my project with considering these feedbacks. In the second project, I improve myself more and more on Altium Designer because the circuit was more complicated than first project and it was a 4-layer PCB.

I earned lots of workplace, R and D experiences in this internship. Researching, building new circuitries, simulating, and asking questions about them to the supervisor gave lots of new information to me. Especially, I learned how space quality products are made and what is the process of them. Also, I found a chance to use my course and project experiences in a real business.

4. REFERENCES

[1] https://uzay.tubitak.gov.tr/tr/kurumsal/hakkimizda-0

[2] https://github.com/uwrobotics/MarsRoverHardware/

[3] Mishra, Mohit & Saw, Rajiv & Deepak, & Channi, Harpreet Kaur. (2018). Overvoltage And Undervoltage Protection System.

[4] https://www.watelectrical.com/overvoltage-under-voltage-protection-system/

[5] https://www.analog.com/media/en/technical-documentation/data-sheets/10178ff.pdf

[6] https://www.electronics-tutorials.ws/opamp/op-amp-comparator.html

[7] https://en.wikipedia.org/wiki/Window_detector