Electronic Devices & Application

Group Mini-Project (DC- PSU Design)

Hamid- Noor- Ofek-Toni- Zahra

Table of Contents

1. Introduction

A. Aim of the report

This report analyzes different designs of the linear DC PSU and implements the chosen design that best fits our specifications. We have also kept in mind different constraints that helped justify the chosen design. These constraints or factors include cost, availability of components we're allowed to use, simplicity, and power dissipation. The report illustrates first the thinking process that led us to the chosen design, calculations, circuit simulation through software-based testing, practical testing, how we built the prototype, and finally the evaluation of the results, calculations, and measurements.

B. Specification Breakdown

The chosen group project leader is Hamid Irfan, with student ID 240422932. Based on the specification requirement $A = 4, B = 2, C = 2, \& D = 9$. The figure below shows the voltage, current, and load resistor requirements calculated based on the formulas given to us. It's important to note that the load regulation has to be less than 5% and the line regulation less than 0.5%.

$$V_{o\,req} = \frac{1}{2}(A * B) + 4\,Volts = \frac{1}{2}(4 * 2) + 4\,Volts = 6.1\,Volts$$

$$I_{o\,req} = 5(CD) + 50\,mA = 5(29) + 50\,mA = 195\,mA$$

$$R_{min} = \frac{V_{o\,req}}{I_{o\,req}} = \frac{6.1}{(195 * 10^{-3})} = 31.282\,\Omega$$

$$R_{max} = \frac{V_{o\,req}}{I_{o\,req} \div 10} = \frac{6.1}{(195 * 10^{-3}) \div 10} = 312.82\,\Omega$$

Figure 1.0: Specification Requirements

2. Design Analysis

A. Choice of Rectifier & Reasoning

In the design of a DC power supply, the rectifier sub-system is required to convert the $12V\ r.m.s$ AC input to a stable DC output while meeting the specification requirements. After evaluating both the half-wave and the full-wave bridge rectifier, we chose to use the full-wave rectifier for our design. This section of the report explains the reasoning behind the choice, considering constraints and tradeoffs.

Initially, we wanted to use a half-wave rectifier due to the simplicity of the design, but the half-wave rectifier alone produces less average voltage compared to the full-wave rectifier. This is explained further through the formulas below.

$$Half - wave\ average\ voltage = V_{avg1} = \frac{V_p}{\pi}$$

$$For\ V_p = 17V, V_{avg1} = \frac{17}{\pi} = 5.4V < V_{req}(6.1V)$$

$$Full - wave\ average\ voltage = V_{avg2} = \frac{2V_p}{\pi}$$

$$For\ V_p = 17V, V_{avg2} = \frac{2(17)}{\pi} = 10.8V$$

In addition to this, we can add a filter capacitor to the rectifier to increase the average (DC) voltage and reduce the ripple voltage. Using a high enough capacitor value, we can make the ripple voltage negligible, which is crucial for good load and line regulation. It's also important to note that the full wave rectifier output frequency is twice that of the half wave, and since the ripple voltage is given by the following equation:

$$V_r = \frac{I_{tot}}{fc}$$

We can say that the ripple voltage for the half-wave is twice the size of the ripple voltage of the full-wave rectifier. Due to the high ripple, achieving the required load regulation (below 5%) using the half-wave is much harder than the alternative option, and it requires a much larger capacitor, which would result in increasing both the size and cost of the prototype.

Furthermore, a full wave rectifier offers additional advantages such as higher r.m.s current due to the rectification of both halves of the cycle, which lowers "stress" on the rectification diodes (different pairs of diodes conduct each half cycle). These advantages help us tackle real-world constraints and make it easier to meet our specified requirements. Hence, the full-wave rectifier does come with a slightly higher initial cost due to more components and additional complexity for troubleshooting. However, we chose to pay that "price" to satisfy the required output voltage and minimize ripple to ensure we meet the required load and line regulation.

B. Choice of Regulator & Reasoning

Voltage regulators are used to provide a constant output voltage despite changes in input voltage and load currents. We evaluated and designed the different topologies of the load regulators in order to meet our requirements. We chose to use a slightly modified version of the zener resistor in series. This section of the report explains why we choose that topology.

Initially, we tried to design a simple zener series resistor regulator. One challenge of the regular topology (Zener & Resistor) is that there is high power dissipation in the series resistor. Another challenge is that there will be very poor load regulation due to the very high zener current when R_{max} is used and very low zener current when the R_{min} is used.

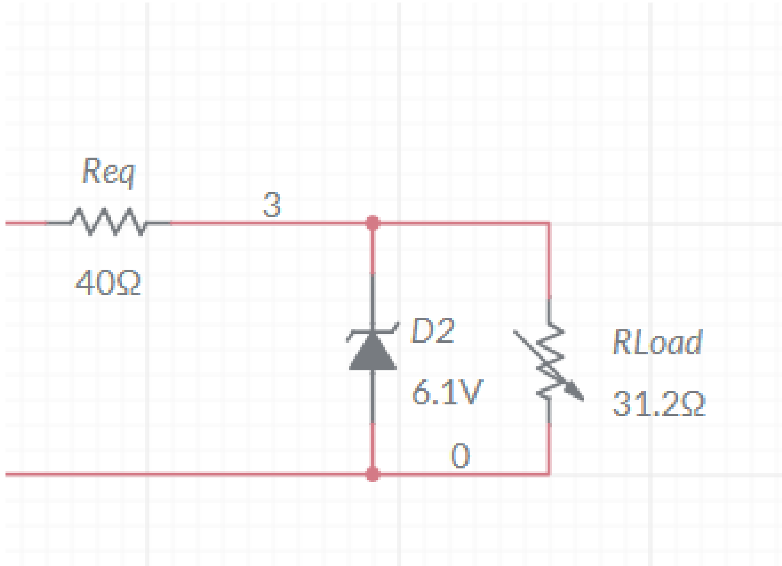


Figure 1.1: Zener & Resistor Regulator Original Topology

As opposed to the original topology in Figure 1.1, we decided to try a different topology, a regulator circuit with a zener and a transistor. This circuit is slightly more complex, but we chose to try it because it provides better load regulation as the zener current is less dependent on the load current. Its relevant circuit diagram is given in the figure below.

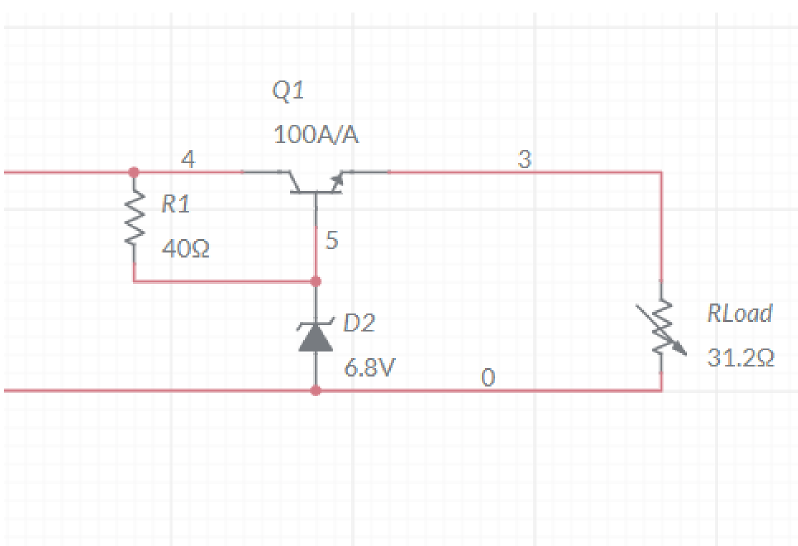


Figure 1.2: Zener & Resistor & Transistor Regulator

When we simulated the above circuit, a major challenge occurred. The power dissipation in the transistor under maximum current is much higher than the limit given by the transistor's datasheet. To explain further, the power dissipation in a transistor is a function of V_{CE} & I_c, since both are defined by our requirements, the only way to reduce the power dissipation in the transistor is to reduce V_c. We tried to do so by reducing the value of the filter capacitor used to create a lower DC voltage in the input of the regulator and also adding a resistor in series with the collector. However, this solution did not work for both loads because the collector current changes as the load changes, which ultimately means that the voltage drop across the series resistor changes as well. The zener would turn off as the ripple voltage pulls the collector voltage down. We weren't able to meet the power dissipation limit of the transistor, so we checked another type of BJT with a higher power rating, but a quick look at its datasheet proved it to be inadequate. Since the third regulator option holds the same challenges as the one just discussed, we decided to try and overcome the challenges of the zener series resistor option.

Recalling the main drawbacks of the first option:
1) High power dissipation in the series resistor
2) Poor load regulation

Starting with the power dissipation challenge, the power dissipation in the resistor is given by $P = IV.$ Since we ideally want a constant current and voltage across the resistor, and we were limited to using only resistors with a power rating of, $0.25W$ we decided to use multiple resistors that will split the total power dissipation needed without changing the total resistance hence keeping the same current and voltages (in the simulation we called that resistance R_{eq} for simplicity.

Now the load regulation measures the steadiness of the zener voltage with changes in load current. From the IV characteristic of the zener diode, we know that there is an optimal region of zener currents that provides the best regulation. We approximated it to be between $50 - 150mA.$ Once a zener diode operates outside that range of currents, it does not hold its zener voltage well. For the zener series resistor topology, the total current I_{tot} flowing through the resistor is divided into load current $I_l$and zener current I_z , making the zener current dependent on the load. Due to the wide range of load currents, the zener current of the topology mentioned falls outside the optimal range of currents. In other words, for changes in the load currents, the zener voltage changes, causing poor load regulation.

$$I_{tot} = I_{z(\max)} + I_{l(\min)} = I_{l(\max)} + I_{z(\min)}$$

$$I_{z(\max)} = \frac{P_D}{V_z} = \frac{1.3W}{6.2V} \approx 210mA$$

$$I_{l(\min)} = 19.5mA$$

$$I_{tot} = 229.5mA$$

$$I_{z(\min)} = 229.5mA - 195 = 34.5mA$$

From these calculations, we can see that our zener currents are outside the desired range, and this is at the absolute maximum rating of the zener currents. In addition, decreasing the maximum zener current $I_{z(\max)}$ to a safe operating region will further decrease the minimum zener current $I_{z(\min)}$. This is why, to keep the zener currents in the desired range, we added a parallel zener. By using this solution, we can choose a total current I_{tot} for which the corresponding two zener currents are within the optimal range of currents under all required loads.

$$I_{tot} = 2I_{z(\max)} + I_{l(\min)} = I_{l(\max)} + 2I_{z(\min)}$$

$$By\ setting\ I_{z(\min)} = 50mA\ , we\ can\ find\ I_{tot} = 295mA\ \&\ I_{z(\max)} = 137mA$$

From this, we can see that the minimum total current through the series resistor that would allow our zener currents to stay within the desired range is $295mA$. The logic we followed is that as long as we use the zener with currents in the desired range, the total current becomes less impactful. This makes our design more robust and can handle changes in the calculated current.

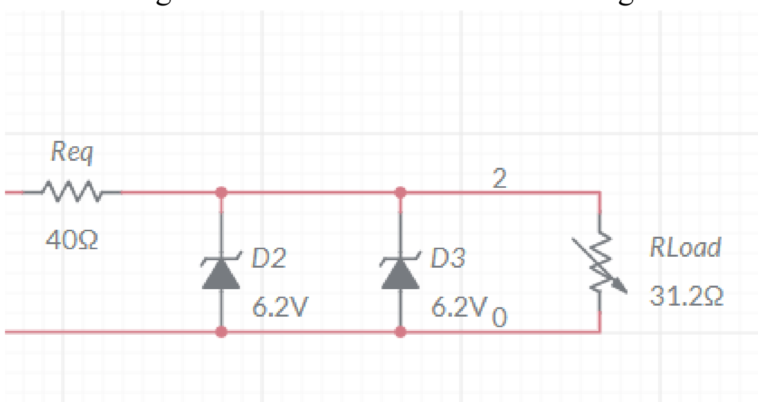


Figure 1.3: Our Chosen Regulator Design

We also considered the voltage across the resistor to be constant, but in reality, it changes with the ripple voltage. This means that as the ripple voltage changes, the current through the resistor will change as well. This will drive a small variation of the zener voltage; in other words, the ripple voltage drives changes in the output voltage. In order to meet the line regulation requirements, we had to make sure that the ripple voltage was negligible, meaning that the changes caused to the output due to the effect of the ripple voltage were small enough. The next challenge was to find a capacitor value that would reduce the ripple voltage and meet the required line regulation. An explanation of how we tackled these challenges will be further explained in the next sections. In conclusion, while adding another zener might seem like an unnecessary complication of the circuit at first, it allowed our design to meet the required load regulation and make our regulator more robust to the tolerance of the components and less dependent on the exact value of total current, while also ensuring safe operation of all components.

3. Design Calculations & Schematic

A. Components Used & Chosen Design

The components we used were Diodes 1N4004, Zener Diodes 6V2, and 16 resistors each 10Ω with 0.25W rating. We also chose a full-wave rectifier with an equivalent resistance and two parallel zener diodes. The figure below shows a 2-port network view of our PSU that is used to provide a general overview of the topology we used; each part is treated as a subsystem with a specific role. The prototype system consists of two sub-systems, the full-wave rectifier and the regulator. Below will be the circuit diagrams and calculations for each subsystem we chose.

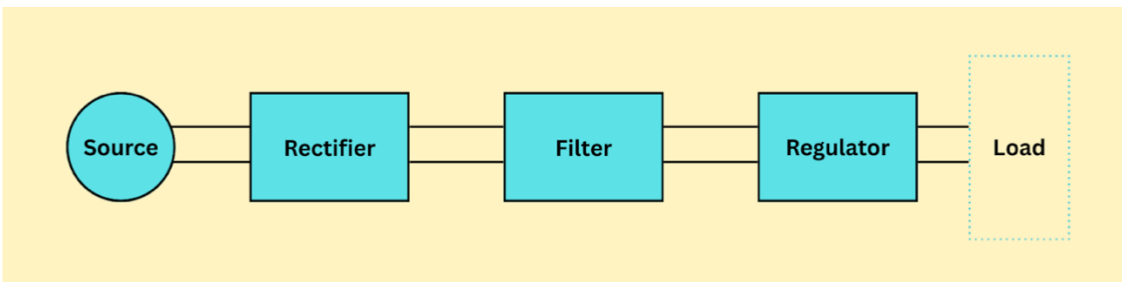


Figure 1.4: 2-Port Network View

A. Design Calculations & Schematic for Regulator Subsystem

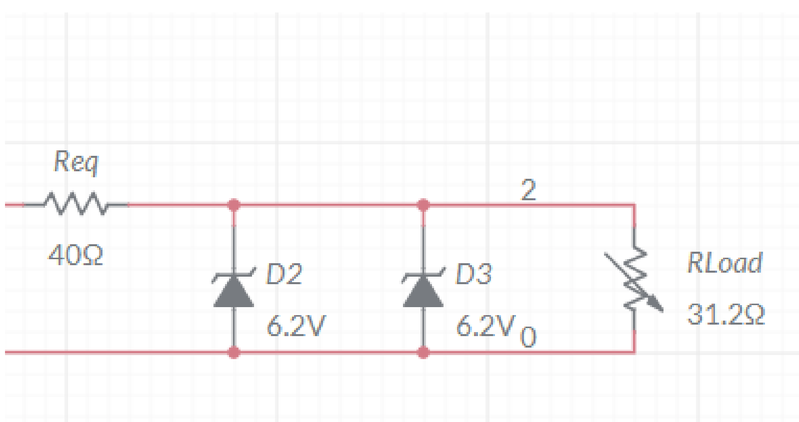


Figure 1.5: Regulator Subsystem

This regulator's output voltage is defined by the voltage of the zener diodes. We used zener diodes of 6.2V since they were available in the lab, and they will provide voltage within the 10% tolerance allowed. If the zener currents are within the optimal range of operation, the zener voltage will be held steady. This would apply even with a small variation in current. A rule of thumb is to operate the zener at around 50% of the maximum current allowed, which we found earlier in this report to be around 210mA. Hence, we approximated the optimal range of operation to be between 50-150mA as mentioned before. The following calculation shows that for a total current of 295mA, the zener current stays within the desired range:

$$I_{tot} = 2I_{z(\max)} + I_{l(\min)} = I_{l(\max)} + 2I_{z(\min)}$$

$$I_{tot} = 2(50mA) + 195mA = 295mA$$

$$I_{z(\max)} = \frac{I_{tot} - I_{l(\min)}}{2} = 137mA$$

To calculate the value of equivalent resistance needed in order to draw 295mA, we assumed the input voltage for these subsystems to be constant. This input voltage is equal to the output voltage of the rectifier after the filter capacitor, which we measured to be 18V.

$$R_{eq} = \frac{V_{in} - V_l}{I_{tot}} = \frac{18 - 6.2}{295 \times 10^{-3}} = 40\Omega$$

To determine the number of resistors needed to make the equivalent resistance and split the power dissipation between them, we will calculate the power dissipated in the equivalent resistance:

$$P = VI = (17V - 6.2V)(295mA) = 3.2W$$

Since one of our design constraints is that the resistors to be used have a power rating of 0.25W. Therefore, the minimum number of resistors required to effectively dissipate 3.2 watts is:

$$No.of Resistors = \frac{3.2W}{0.25W} = 12.8 \approx 13\ resistors$$

As it is recommended to use resistors at around 75% of the maximum power dissipation, we used 16 resistors. The actual optimal number of resistors we need to use is 17, but it was easier to get the needed equivalent resistance using 16 resistors, and the change is not significant in terms of power dissipation in the resistors.

B. Design Calculations & Schematic for Rectifier Subsystem

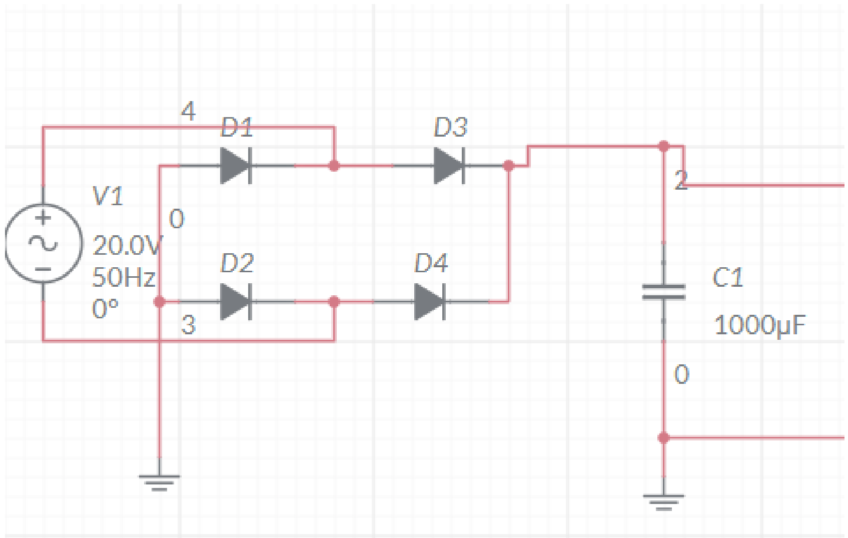


Figure 1.6: Rectifier Subsystem

The figure above is the circuit diagram for the full-wave rectifier sub-system, making use of 1N4004 diodes. No calculation is required for designing the full-wave bridge rectifier. The Peak inverse voltage of the used diodes is 400V, which is much higher than the case of use We measured the power supply to provide 20V peak and not 12V rms as specified, since the 1N4004 has a forward voltage drop of around 1V, the rectified voltage at the output of this subsystem is 18V peak. This has occurred in our measurements as expected. As discussed earlier, the value of the capacitor will determine the ripple voltage. This is given by:

$$V_r = \frac{I_{tot}}{fc}$$

The ripple voltage has a direct effect on the line regulation; this means that a big enough ripple will change the currents flowing through the zeners, hence changing the output voltage. A big enough capacitor can reduce the ripple voltage, therefore improving our line regulation. The line regulation requirement for this project was specified to be less than 0.5%. Since we couldn't find the IV characteristics graph of the zener used, we didn't know how much a specific change in the current would change the output voltage. We approximated that a ripple voltage of around 1V would not significantly affect the output voltage, because it produces a current variation around the total current we aimed for. In other words, recalling that

$$I_{tot} = \frac{V_{in} - V_z}{R_{eq}}$$

The formula above shows how the current through the equivalent resistor of the regulator is affected by V_{in} (which is the voltage at the input of the regulator subsystem), and noting that V_{in}

changes with ripple. We can evaluate the approximated change in total current. since the ripple voltage is measured as peak-to-peak V_{in} changes between

$$V_{in} = V_{rect} \; and \; V_{in} = V_{rect} - V_r$$

By substituting V_{in} in the total current, we can approximate how the ripple will change the total current. When we put in the numbers, we see that the total current changes between 270mA and 295mA. This is a small variation of I_{tot}, and since it's around our approximated value of total current, it shouldn't have a significant effect on the output voltage. To find the capacitor value we modified the equation of the ripple voltage to:

$$C = \frac{I_{tot}}{fV_r}$$

Given that $I_{tot} = 295mA, f = 100Hz,$ and $V_r = 1V$, we find that we need a capacitor of 2950μF to create the desired ripple, the closest value that was available in the lab was 2200μF. This value works well but the capacitor was big in size and took a lot of space. We tried to find a smaller capacitor for which the output would still be below the line regulation limit. The next available value in the lab was 1000μF. Following the same calculation, we can find that the total current stays within the range of 221mA and 295mA. For this capacitor, we calculated a line regulation of around 0.2% which is well within the required range. We tried even a smaller capacitor of 470μF, but we found that for that capacitor line regulation of around 0.5% which is we thought was too close to the required upper limit of the line regulation so we decided to leave headroom for the tolerance and real life limitation and use the safer option of the 1000μF which was well within the required range but smaller in size from the option we first checked (2200μF).

4. Simulation & Tracing

A. Analysis & Multisim Graphs & Measurements at Relevant Points

The figure below displays a simulation of the voltage coming out of the rectifier sub-system of our design circuit. We can see that the current I_{tot} measured using the green probe swings between 250mA and 300mA. This is a bit higher than expected. It isn't critical as it is still close to the range we expect; this difference is due to the difference between the ripple we see on the simulation and the one we measured. However, it won't affect the as both ranges of total current have the regulator operate in its reverse breakdown region.

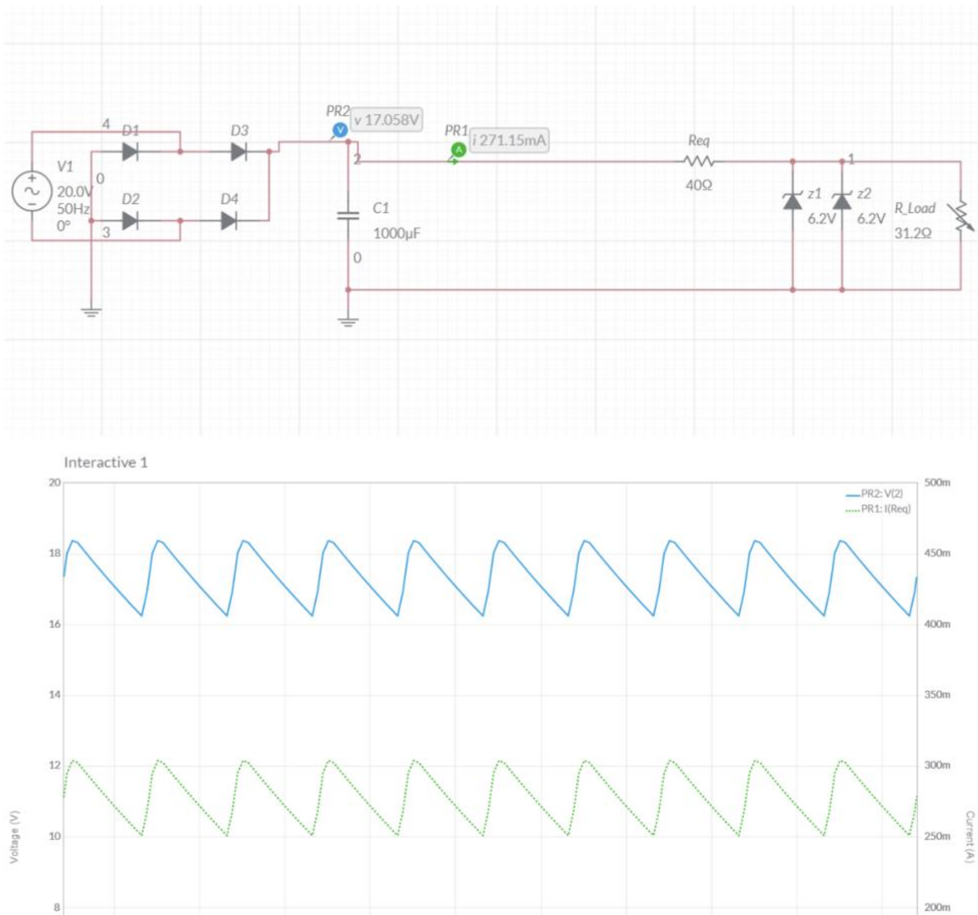


Figure 1.7: Rectifier Subsystem Simulation

In the other simulation below, we simulated the circuit using a load of 33Ω and 330 Ω to match the values of resistors that were available in the lab. This allows us to compare the tested prototype to the simulation. The figure below displays a simulation of the voltage and current across a load of 330 in the regulator sub-system.

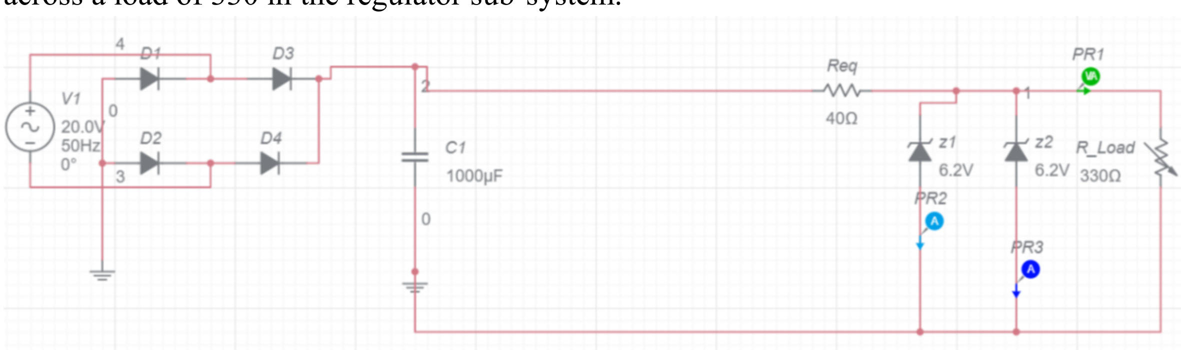


Figure 1.8: Regulator Subsystem Simulation

In this situation, we tested the zener and R_{max} currents and voltage we also verified that the zener stays within the expected range and maintains the regulation. In the graph below, we see that the 2 zeners split the total current with the load, and each zener current varies between

115mA and 140mA. The zener current drops due to the effect of the ripple, but it stays within the optimal range of currents we have approximately. The regulated voltage stays constant around 6.2V as expected. Both zeners stay within safe operation range in the context of power dissipation with currents much smaller than the absolute limit of 210mA.

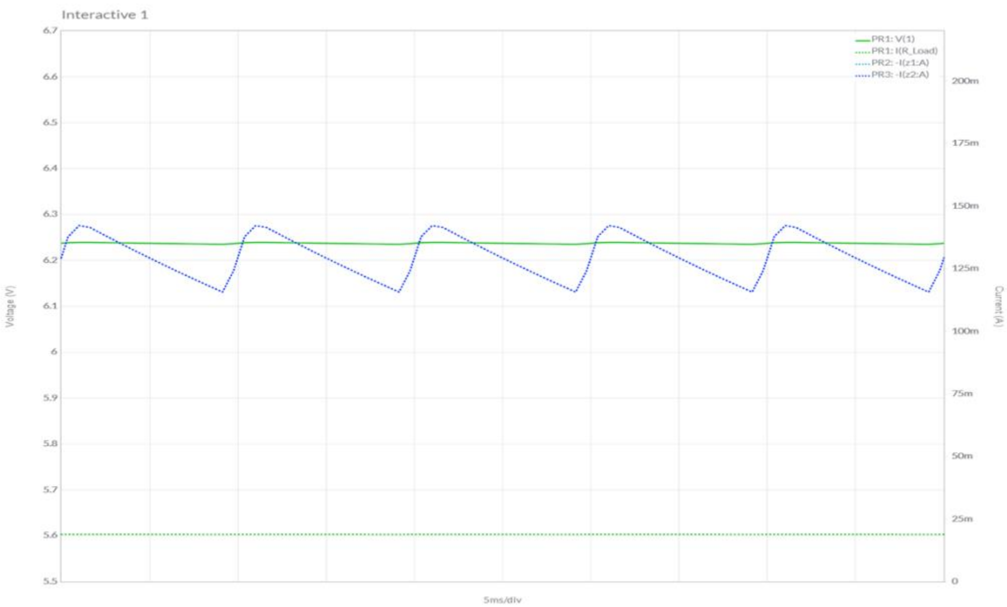


Figure 1.9: Zener Simulation

The image below displays a simulation of the voltage and current across a load of 33 Ω in the regulator sub-system. In this situation, we tested the zener and Rmin currents and voltage. We also verified that the zener stays within the expected range and keeps the regulation. In the graph below, we see that the 2 zeners split the total current with the load; each zener current varies between 30mA and 50mA. The zener current drops under the low limit of the optimal range of currents we have approximated. These drops in currents are due to the ripple voltage. From the graph, we see that the drops of zener current don't have a significant effect on the output voltage. Both zeners stay within safe operation range in the context of power dissipation with currents much smaller than the absolute limit of 210mA.

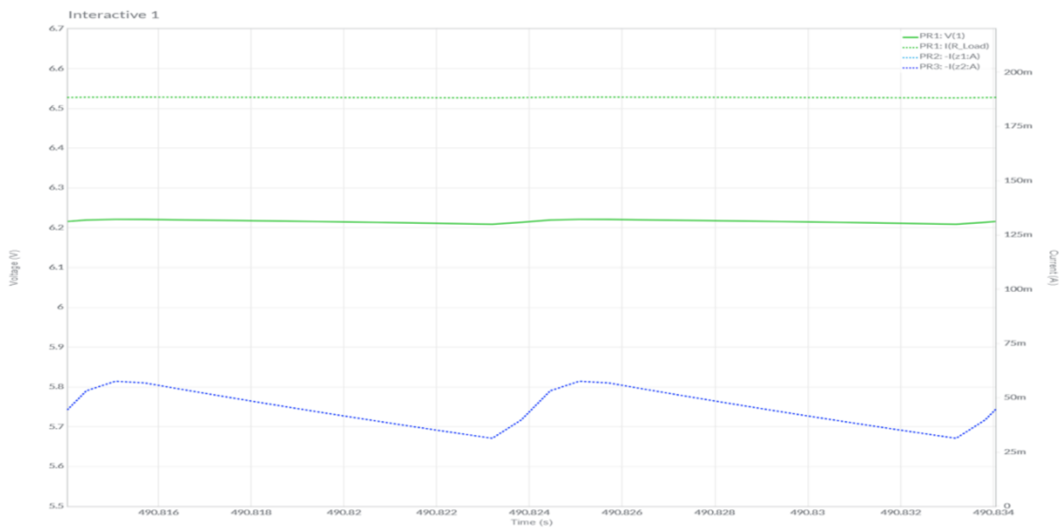


Figure 2.0: 33Ω Simulation

In the last simulation test we conducted, we verified that the resistor won't dissipate much power.

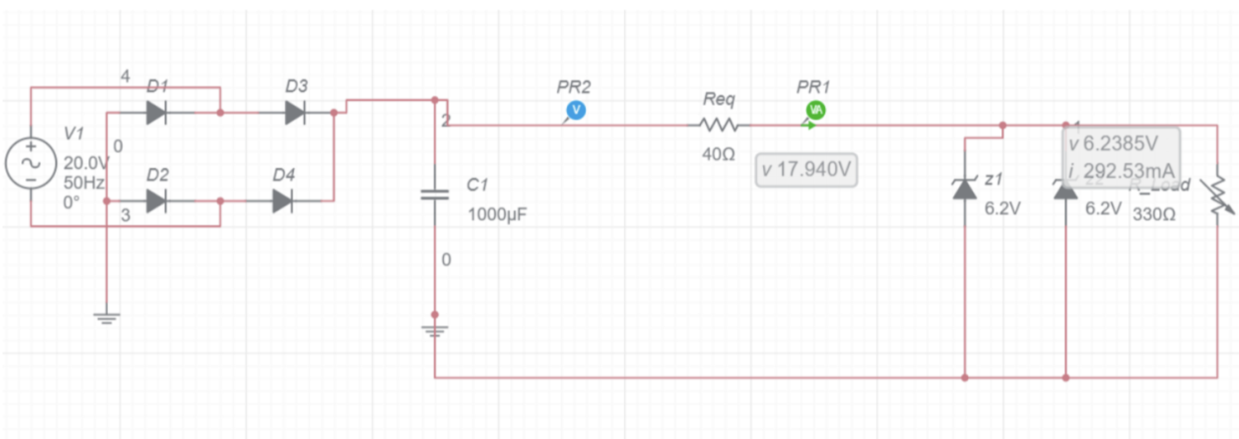


Figure 2.1: Power Dissipation

The voltage across the resistor is the difference of the probe voltages

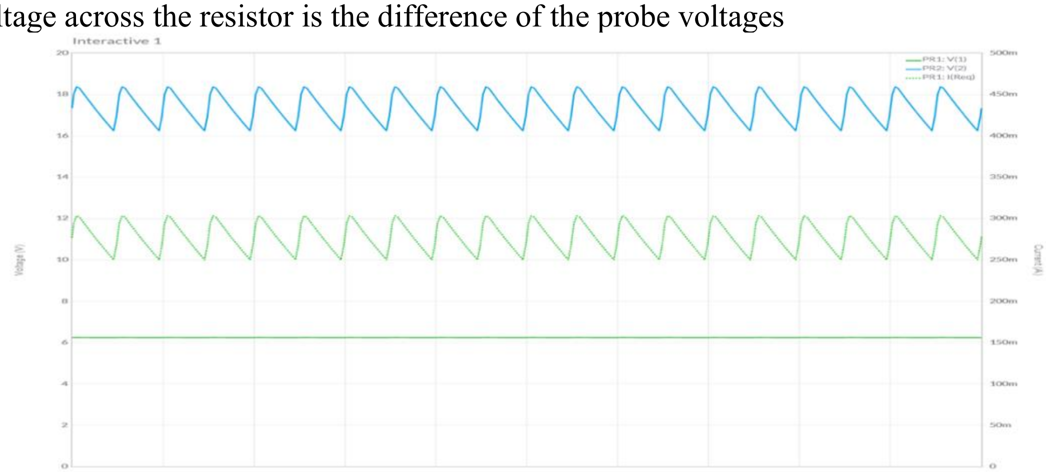


Figure 2.2: Probe Traces Simulation

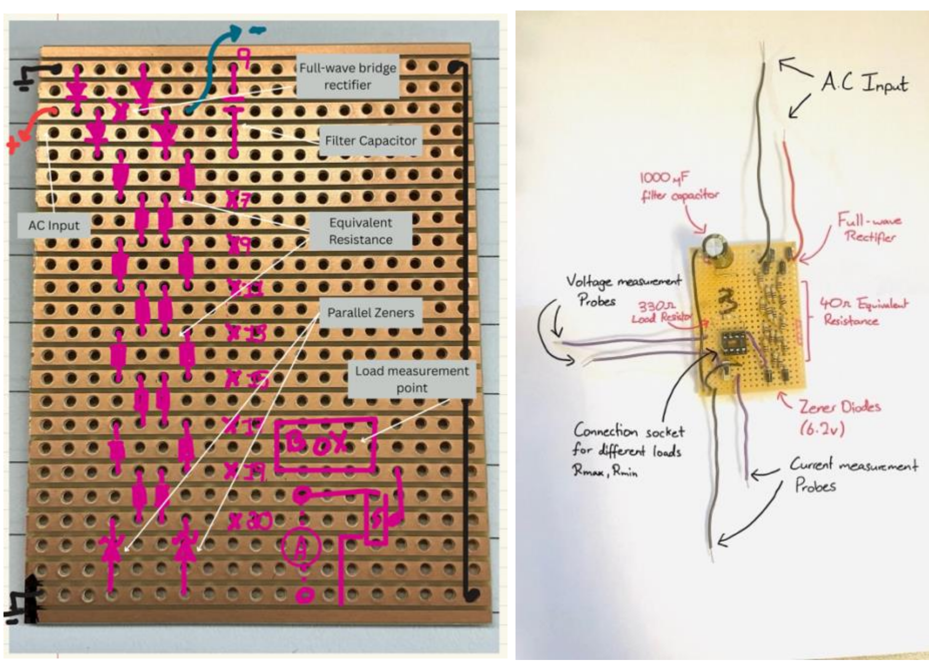
In the worst-case scenario, the peak voltage is 18.3V. The calculation below explains how, even in the worst-case scenario, the equivalent resistance will still be able to effectively dissipate the power. Our expected current value for the worst-case scenario is 300mA.

$$V_{req} = 18.3V - 6.2V = 12.1V$$
$$P_{req} = VI = (12.1V) * (300mA) = 3.63W$$
$$Number\ of\ resistors = \frac{3.63}{0.25} = 14.52 < 16\ resistors$$

By using the worst-case peak ripple voltage for our calculations, we still do not exceed the power limitations for the equivalent resistors. The minimum number of resistors we require to dissipate the maximum 3.63W of power is 15, and we are using 16, which is more than enough to dissipate the power across the equivalent resistance.

5. Prototype

A. Picture of Prototype & Layout Plan



Figures 2.3: Design Layout of Prototype on Stripboard & Soldered Design

B. Explanation of Layout Plan

Observing the above diagram, the crosses (x) on the layout refer to 'break points' that we created on the stripboard by using a Phillips screwdriver to break the copper connection rails to isolate parts of the circuit. This can be seen in the part labelled 'Full-wave bridge rectifier', where a break point was made so that the diodes aren't shorted, and the pairs of diodes can rectify half cycles independently. There is also the filter sub-system, where the filter capacitor is connected right after the rectifier in order to smooth out the pulsating DC voltage and provide a more stable and steady DC output that feeds into the regulator sub-system. Finally, we have the regulator sub-system that is a combination of the parallel zener's configuration as well as the R_{eq}(Equivalent Resistance), where we used resistors in series and parallel to produce an equivalent resistance of 40Ω.

Moreover, the component in the diagram labelled 'box' is an 8-pin connection socket, which we used to connect the different load resistors according to our specification. The switch is used to change between measuring the voltage across the load resistor (which would require a measurement instrument to be connected in parallel to the resistor) and measuring the current (which would require the instrument to be in series with the load resistor). In the soldered prototype, we added probes in the form of wires to measure the voltage and the current. These probes were connected in their respective configurations to the 8-pin connection socket. Two additional probes were also used for the A.C. input, which were connected before the rectifier sub-system.

6. Testing Plan

A. Introduction

Testing is an essential part of developing an electronic circuit, as it allows us to verify that the prototype design operates correctly and safely before final implementation. By carrying out systematic measurements, we can confirm that the circuit meets our specification values and identify any issues in the development process. The objectives of our testing process are defined below. We designed the prototype with testing in mind, and it was a key factor throughout our design process from the initial calculation phase through the simulation phase and eventually to the final prototype phase as well. This is because testing allows us to identify and fix mistakes in our design and circuit before we conclude our final design and prototype.

Objectives of Testing:

1. Verify the correct DC output voltage and current
2. Evaluate line regulation and load regulation
3. Measure ripple voltage
4. Compare measured results to expected values

Equipment Required

1. Bench AC source/transformer (as per design)
2. Different load resistor values (33 & 330)
3. Digital multimeter (DMM)
4. Oscilloscope with probes

Equipment Required:

1. Bench AC source/transformer (as per design)
2. Different load resistor values (33 Ω & 330Ω)
3. Digital multimeter (DMM)
4. Oscilloscope with probes

B. Method & Approach

We designed our circuit with testing in mind as one of our main priorities. The goal of our testing was to ensure that our measured output values were aligned with our calculations and simulation-based outputs. Our goal for testing is to make sure that the prototype meets its specification requirements and checks its accuracy, reliability, and performance. We used Multisim to simulate and test the circuit designs before we started assembling the circuits in the lab. The benefit of this is that we were able to get a brief idea of what values to expect for our output, bearing in mind that simulation values are not 100% accurate and that we should expect some variation in the outputs between the simulation tests and the practical tests.

For our chosen design, we needed to verify our calculations and calculate our load and line regulation with values from our circuit instead of just relying on simulation testing. A precautionary measure we took before we started assembling the circuit was testing all our gathered components in the electronics lab one by one to make sure that we were not dealing with any faulty components. This was done by using a digital multimeter (DMM) to measure the value of the resistors used and using simple circuits on a breadboard to test whether the diodes were working. Once we had tested the components and verified that they were all functional according to expectations, it was time to assemble the entire circuit on the breadboard.

We first assembled the circuit on a breadboard and made sure that the circuit was functional by regularly connecting it to the AC power source and observing the outputs. We also made sure that there were no shorts in the circuit or burnt components. We then tested the breadboard version of our circuit using both DMMs and an oscilloscope to check whether our outputs were in line with the expected outputs from our simulation. The DMM was mainly used to measure DC voltages and currents at different points in the circuit, such as the input voltage, the regulated output voltage, and the current drawn by the load. This allowed us to confirm that the output voltage stayed close to the design value as the load was changed.

The oscilloscope was used to measure the output voltage and waveform. Since this design is a DC PSU, we were ideally expecting a flat line as the waveform in resemblance to a true DC voltage supply. By viewing the waveform on the oscilloscope, we were able to check the ripple frequency and how the output responded when the different loads were connected or removed. Using both instruments together gave us a clear and reliable understanding of the circuit's performance on the breadboard at both the output level and at the individual component level.

After testing the circuit design on the breadboard and verifying that our output values were in line with our expected values based on our calculations and circuit simulations, we created the soldered prototype, and now it was time to test the values on the final prototype. We were prepared for some variation in the outputs because components soldered to a stripboard sometimes behave differently from those connected on a breadboard (e.g., added resistance from connecting wires in a breadboard). However, the difference between outputs was negligible.

Our prototype design, which has been added and labelled in the 'Prototype' section of this report, allows us to test both the load voltage and current with the help of a switching mechanism that we designed in our layout plan. With the help of this mechanism, we were able to effectively measure and compare the output values with those of our expectations. The results section below records our expected and actual values, as well as error rates and regulations.

C. Measured & Calculated Results

$$\boldsymbol{For\ the\ 33\Omega}: \ Expected\ V_o = 6.1V, Measured\ \ V_o = 6.17V$$
$$Calculated\ Error = 1.14\%$$

Figure 2.4.: Measured Output for the 33Ω

$$\boldsymbol{For\ the\ 330\Omega}: \ Expected\ V_o = 6.1V, Measured\ \ V_o = 6.37V$$
$$Calculated\ Error = 4.42\%$$

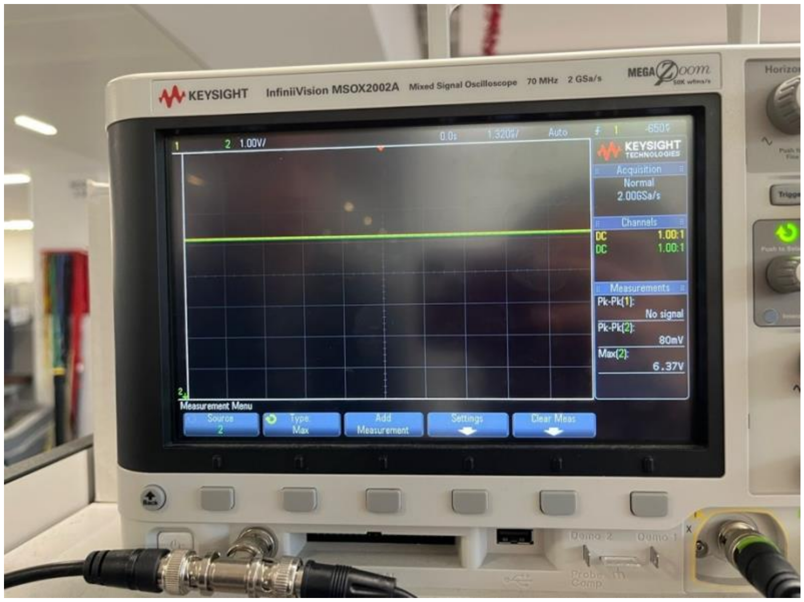


Figure 2.5: Measured Output for the 330Ω

$$\textbf{For the 33}\boldsymbol{\Omega}: \ Expected\ I_o = \frac{6.17V}{33\Omega} = 186mA, Measured\ \ I_o = 192mA$$
$$Calculated\ Error = 3.23\%$$

Figure 2.6: Measured Current for the 33Ω

$$\textbf{For the 330}\boldsymbol{\Omega}: \ Expected\ I_o = \frac{6.37V}{330\Omega} = 19.3mA, Measured\ \ I_o = 19.0mA$$
$$Calculated\ Error = 1.58\%$$

Figure 2.7: Measured Current for the 330Ω

$$load\ regulation = \frac{V_{NL} - V_{FL}}{V_{FL}} \times 100 = \frac{6.37 - 6.1}{6.1} \times 100 = 4.4\% < 5\%$$

$$line\ regulation = \frac{\Delta V_o}{\Delta V_i} \times 100 = \frac{70mV}{36} \times 100 = 0.194\% < 2\%$$

Testing Criteria	Simulation Testing	Prototype Testing
V_o for R_{max}	6.30	6.37V
V_o for R_{min}	6.21	6.17V
I_o for R_{max}	18.8mA	19.2mA
I_o for R_{min}	188mA	183mA

Figures 2.8: Comparison with Simulation Outputs

D. Evaluation & Discussion

One of the main discussion and evaluation points for our report is the disparity between the measured outputs in our simulations and practical testing. By comparing the values measured in the simulation section of the report and the testing section of the report, it is apparent that there is a significant difference in the measured values. In order to reach a conclusion and evaluate this difference, we must understand the fundamental difference between simulation testing and practical hardware-based testing.

Simulation tools are extremely useful for setting expectations in testing and planning, but they represent an idealized version of reality, while real circuits must deal with practical limitations and imperfections. In simulation software, most components are modelled using ideal or near-ideal parameters. Resistors often have exact values, power supplies are perfectly stable, and connections have no resistance. In contrast, real components have manufacturing tolerances. For example, a 330Ω resistor with a 5% tolerance can actually be anywhere between 313.5 Ω and 346.5 Ω. When several such components are used together, these small variations can add up and affect the measured output.

Another major factor is that real components are not ideal. Diodes have voltage drops that vary with temperature and current, transistors have parameter variations such as gain (β), and op-amps have limitations such as input bias currents, offset voltages, and finite gain. In simulations, these effects are often simplified or fixed, whereas in real life, they change with operating conditions. Temperature also plays a role, as components heat up during operation, causing their electrical characteristics to drift slightly. Measurement tools and physical construction also contribute to differences. Multimeters and oscilloscopes have their own accuracy limits and can slightly load the circuit being measured. Breadboards introduce additional resistance, capacitance, and noise due to long wires and imperfect contacts, which are not present in simulations.

Upon evaluating the difference between our tests (simulation testing vs practical testing), discussing and evaluating our test results in accordance with our testing plans is key. The testing plan developed for this prototype was effective in verifying the performance of the DC power supply under both no-load and loaded conditions. By following a structured approach using DMMs and oscilloscopes, the key design requirements, such as output voltage accuracy, current capability, load regulation, and line regulation (all in reference to specification values) were successfully assessed. Testing the circuit across different load values ensured that the PSU was evaluated under realistic operating conditions, rather than only ideal cases.

The measured output voltage showed good agreement with the expected value of 6.1 V. Under a 330 Ω load, the output voltage was measured at 6.37 V, which is slightly higher than expected but still acceptable for a linear regulator and well within the 10% tolerance criteria. When a 33 Ω load was applied, the output voltage decreased to 6.17 V. These small variations indicate that the regulator maintains a relatively stable output despite changes in load, demonstrating effective voltage regulation. The calculated loaded cases further confirm that the circuit performs within the specified limits. The output current was also within bounds for both load values. This shows that the power supply is capable of delivering the required current without significant deviation or instability.

Load regulation was calculated to be approximately 4.4%, which satisfies the design requirement of being less than 5%. This result indicates that the output voltage does not significantly drop as load current increases, reflecting good regulator performance. Additionally, the calculated line regulation of 0.194% demonstrates that changes in input voltage have very little effect on the output voltage, confirming that the PSU provides a stable output even when the input varies. Power dissipation measurements also showed that all components operated within their power rating limits. The use of multiple resistors to form the equivalent resistance successfully distributed power dissipation, ensuring that no single resistor exceeded its 0.25 Watts power rating. This confirms that the design complied with the imposed component constraints and that the prototype is safe during operation.

Overall, the testing plan successfully validated the design objectives of the DC power supply. The results show that the circuit meets all specified performance criteria and behaves predictably in close accordance to calculations and simulations. Any small deviations observed are consistent with expected real-world component tolerances and do not negatively impact the overall functionality of the PSU.

E. Limitations & Improvements

Despite meeting the specification requirements, the large number of resistors required to achieve the desired equivalent resistance increased the physical size of the circuit and made the soldering process more complex and time-consuming. Another limitation is the dependence of the output voltage on the zener diode characteristics. Although the use of parallel Zenner diodes improved the load regulation, the output voltage is still sensitive to the diode's tolerance and temperature variations. This could lead to small voltage drifts during long-term operation or under changing ambient conditions.

An improvement would be to replace the multiple low-power resistors with fewer resistors of a higher power rating. This will reduce the component count, simplify the physical layout, and improve mechanical reliability while maintaining the same electrical performance. Further improvements could also include the addition of protection features, such as current limiting, to enhance robustness. Using an integrated voltage regulator IC could improve output voltage accuracy and temperature stability. This change would make the design more suitable for practical, long-term use while still meeting the original specification requirements. Another improvement that could potentially benefit the design is using more zener diodes in parallel. This would allow us to make the regulator more robust to small changes due to ripple or tolerance.

7. Conclusion

The prototype produced as a result of this project successfully met its objective of acting as a functional DC PSU operating within design constraints and limitations. Our group collaborated effectively for designing, implement, and validating a design that satisfies the given technical specification. Through a structured design process that included theoretical analysis with the support of calculations, software-based simulation, breadboard testing, and final prototype construction, a functional PSU was achieved. The final design met the required output voltage of approximately 6.1 V and delivered the specified load current for each load that remained within the current limitations while remaining within the defined limits for both load regulation and line regulation. The components also remained within safe power dissipation levels, and risks associated with high current and high power were averted.

Experimental results confirmed that the load regulation was approximately 4.4%, meeting the requirement of being less than 5%, and the line regulation was measured at 0.194%, comfortably within the specified limit of 0.5%. The thorough testing process that we carried out at various stages of the design throughout its development demonstrated that the prototype behaved consistently with calculations and simulations under different load conditions. Although minor deviations were observed, these were expected due to real-world component tolerances and did not affect overall performance. In conclusion, the testing results verify that the final prototype satisfies the specification requirements and demonstrates a technically sound and well-validated DC power supply design, as outlined in the project report structure and assessment criteria.

8. Team Contribution Table

Task	Hamid	Ofek	Noor	Zahra	Toni
Analysis of technical specification & design options	X	X	X	X	X
Design Calculations for the chosen design	X	X	X		X
Simulation & Their analysis	X	X			
Building of the prototype	X	X	X	X	
Testing planning, execution, and result analysis	X	X	X		
Evaluation of the design and discussion	X	X	X	X	X
PowerPoint Presentation			X	X	
Technical Documentation (Writing & Editing)	X	X	X		X