ELECENG 2EI4 Project 1 Report
Instructor: Dr. Yaser M. Haddara

Haseeb Shaikh (400521659)

February 9th, 2025

Summary:
This project involves designing and constructing a DC power supply that delivers 10 mA at 3V ± 0.1V from an AC source that is 120V (rms) at 1 kHz. The design consists of a circuit which includes a transformer, rectifier, filter and may or may not include a regulator.

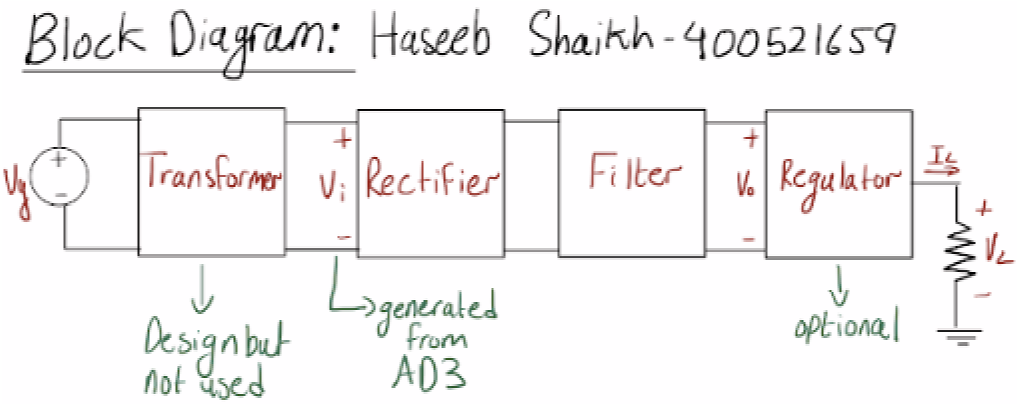

Figure 1: Block Diagram

Design

I. Transformer:

My DC power supply design does not include a physical transformer. The input voltage required for this design is AC Input with 4.6 V amplitude based on my calculation shown in Figure 3. The turns ratio (N1 :N2, where N1 is the primary windings, and N2 is the secondary windings) of the transformer that would be needed for the input to be generated from a 120 V AC source is 37:1, based on my calculations shown in Figure 3.

II. Rectifier:

The rectifier topology I chose for my design is a Full-wave bridge rectifier which consists of four 1N4148 diodes. Rectifier is considered one of the crucial stages of the DC power supply from an AC source as it converts a purely AC input to a unipolar output which gives it a nonzero average voltage. Full wave rectifiers utilize both halves of the sinusoidal input (AC source) by inverting the negative half cycle [3]. Whereas, half wave rectifiers utilize only the positive halves of the cycles and essentially, discarding the negative halves cycles, making it less efficient conversion from AC to DC as it results in a significant ripple in the output and not performing ideally as a rectifier [3]. In a half-wave rectifier, the single diode only conducts when it is forward biased such as in the positive half cycles of the AC input. In a full-wave bridge rectifier, two of the four diodes are always forward biased and the other two are always reverse biased depending on the positive and negative half cycle of the AC input which allows two of the diodes to always conduct. This enables full-wave rectifiers to deliver a higher average DC output and a ripple frequency that is double the input frequency which results in a more smoother and stable DC output [3].

The full-wave bridge rectifier was a more suitable design then the full-wave center-tapped transformer as, it does not require a center-tapped transformer, simplifying the circuit design as well as its considered to be more efficient due to maximizing transformer utilization and elevating the output voltage to generate a more constant and consistent DC output [3].

Relevant diode parameters include maximum forward voltage drop representing voltage drop when current passes through diode, peak reverse voltage representing the maximum reverse voltage the diode can withstand without breaking down and maximum current rating representing the highest continuous current the diode can handle without being damaged. These values were obtained through the datasheet for 1N4148 diodes. Based on the datasheet, the diode features a maximum forward voltage drop of 1 V at forward current value of 10 m but a minimum forward voltage drop of 0.8V is assumed for simplicity according to LTSpice, peak reverse voltage value of 100 V, and a maximum forward current rating of 200 mA [2].

III. **Filter:**
The filter that was used in my design was a filter capacitor essentially, a capacitor in parallel with a load which significantly reduces the variations in the rectifier output voltage. In my filter design, I used a 25 uF capacitor by determining that the average current supplied by capacitor should be equal to the output current which is given as 10 mA, with peak to peak ripple 0.2 V due to the given tolerance $\pm\ 0.1$ V and a 2000 Hz frequency as the full wave rectifier doubles the input frequency. The values calculated for the capacitor and resistor are shown below in figure 3.

IV. **Regulator:** I chose not to include a regulator in my design.

V. **Complete Circuit Schematic:**

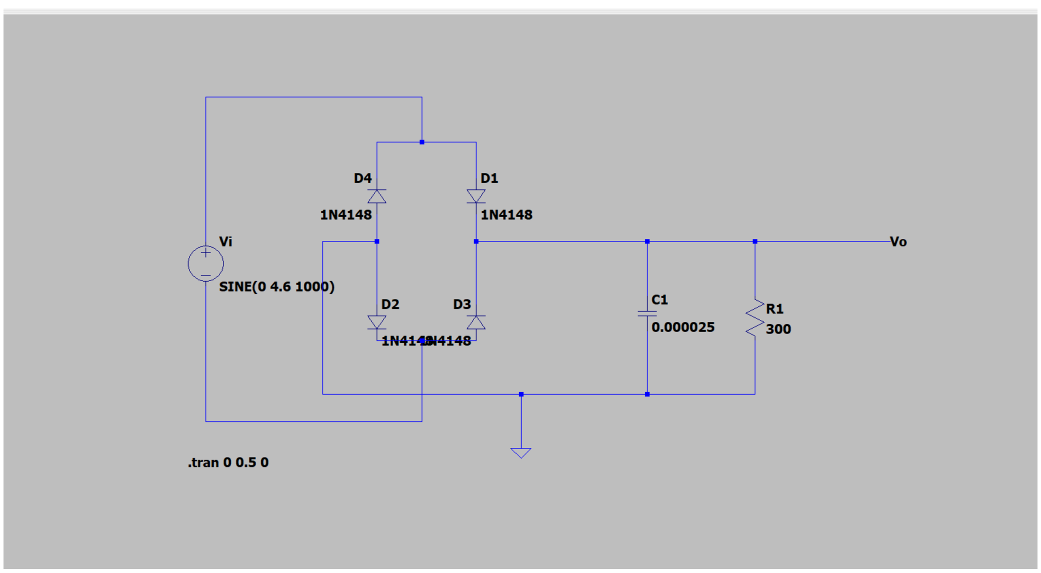

Figure 2: LTSpice Circuit Schematic

VI. Calculations:

V_{in} (Input Voltage):

$V_D = 0.8V$ (For 1N4148 diode)

$$V_{in} = V_{DC} + V_{D1} + V_{D2}$$
$$V_{in} = 3V + 0.8V + 0.8V$$
$$V_{in} = 4.6V$$

Transformer: 120 rms

Transformer ratio: $\dfrac{\text{Primary voltage}}{\text{Secondary voltage}} = \dfrac{E_p}{E_s} = a$

$$E_p = 120 \text{ rms} = 120\sqrt{2} \text{ V}$$
$$E_s = 4.6V$$

$$a = \frac{120\sqrt{2}}{4.6} \doteq 36.89 \doteq 36.9$$

Capacitance (C):

$$V_{rpp} = \frac{I_L T_{out}}{C}$$

$T_{out} = \dfrac{1}{F_{out}}$, $I_L = 10mA = 0.01A$

$$C = \frac{I_L}{F_{out} V_{rpp}} = \frac{(0.01A)}{(2000\text{ Hz})(0.2V)} = 2.5 \times 10^{-5} F = 0.000025 F$$

Resistance (R_L):

$$R_L = \frac{V_o}{I_o} = \frac{3V}{10mA} = 300\,\Omega$$

Figure 3: Theoretical Calculations

VII. Expected performance

Considering the design, the 300 ohms resistor's voltage should be equal to the output voltage value which should look like a DC output equal to 3V within a tolerance of $\pm\ 0.1V$ and the current through the resistor should be approximately 10 mA according to my design calculations.

VIII. Design tradeoffs, design margins, component ratings, safety, and other issues

Due to some components not being provided in the componnent kits, some changes had to be made in the circuit. Since the kit did not provide a 300 ohm resistor was not included in the kit, a 330 ohm resistor was used instead since it's the closest value to a 300 ohm resistor and was recommended by the professor. The kit also did not provide a 25 uF capacitor so a 100uF capacitor was used instead as a higher capacitor value will only resist more change in voltage, resulting in decreasing the ripple voltage and voltage curve being more smoother DC output. The power ratings such as output voltage, output current were verified after design changes in the simulation before constructing the actual circuit to ensure accurate results and for safety purposes.

Measurement and analysis

I. Photograph of my actual circuit

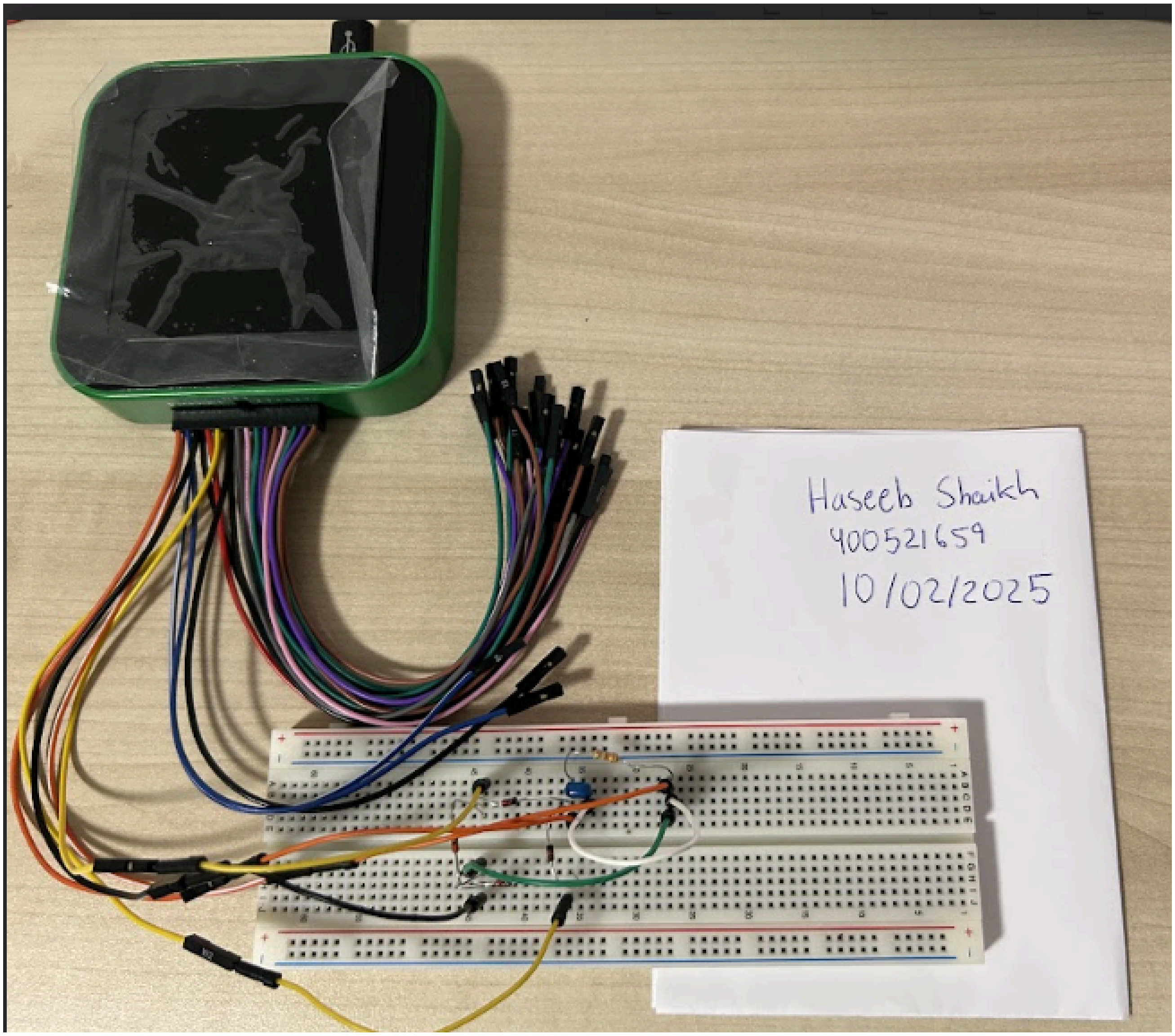

Figure 4: Physical Circuit connected to Analog Discovery 3 (AD3)

II. Measurement Procedure

The performance was determined through the use of AD3 and the WaveForms software. Through the use of Wavegen, the input voltage was set to a sine wave with amplitude of 4.6 V without changing other default settings as shown in figure 5. By connecting Channel 1 (orange wires) across the 330 ohm resistor and through the use of oscilloscope (Scope in WaveForms software), the output voltage (voltage across 330 ohm resistor) was obtained and the current across the 330 ohm resistor was measured by using ohm's law ($I = \frac{V}{R}$).

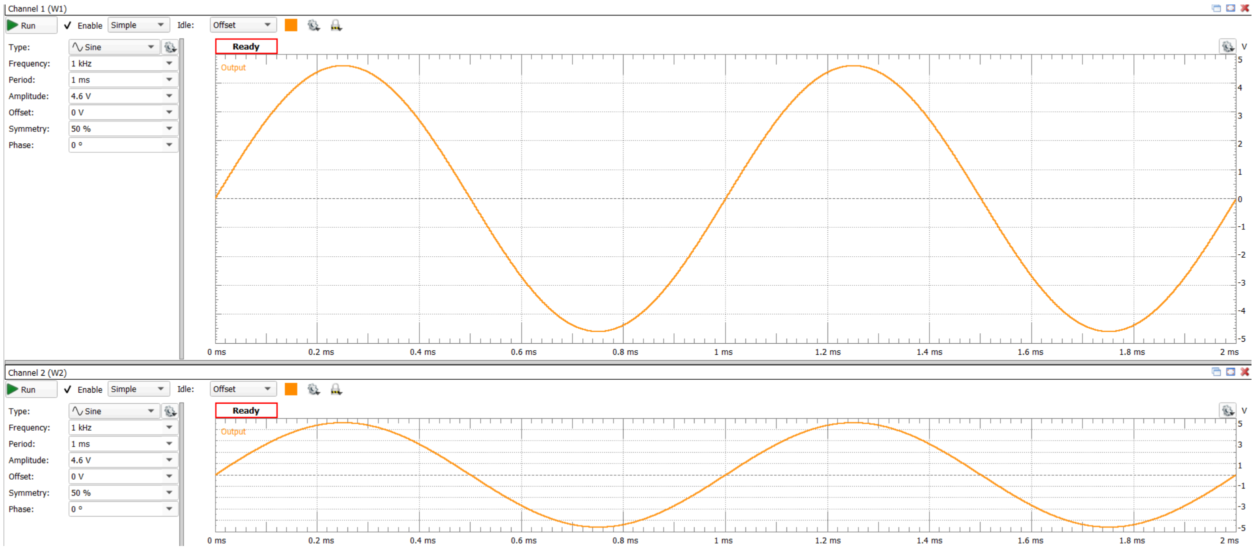

Figure 5: Input Variables (Wavegen in WaveForms Software)

III. **Key Measurement Results:** (As shown in figure 6)

Input Voltage= 4.6 V

Minimum Output Voltage = 2.941 V

Maximum Output Voltage = 2.941 V + 0.12128 V = 3.06228 V

Output Current = 8.912 mA

IV. Screenshot of your oscilloscope output

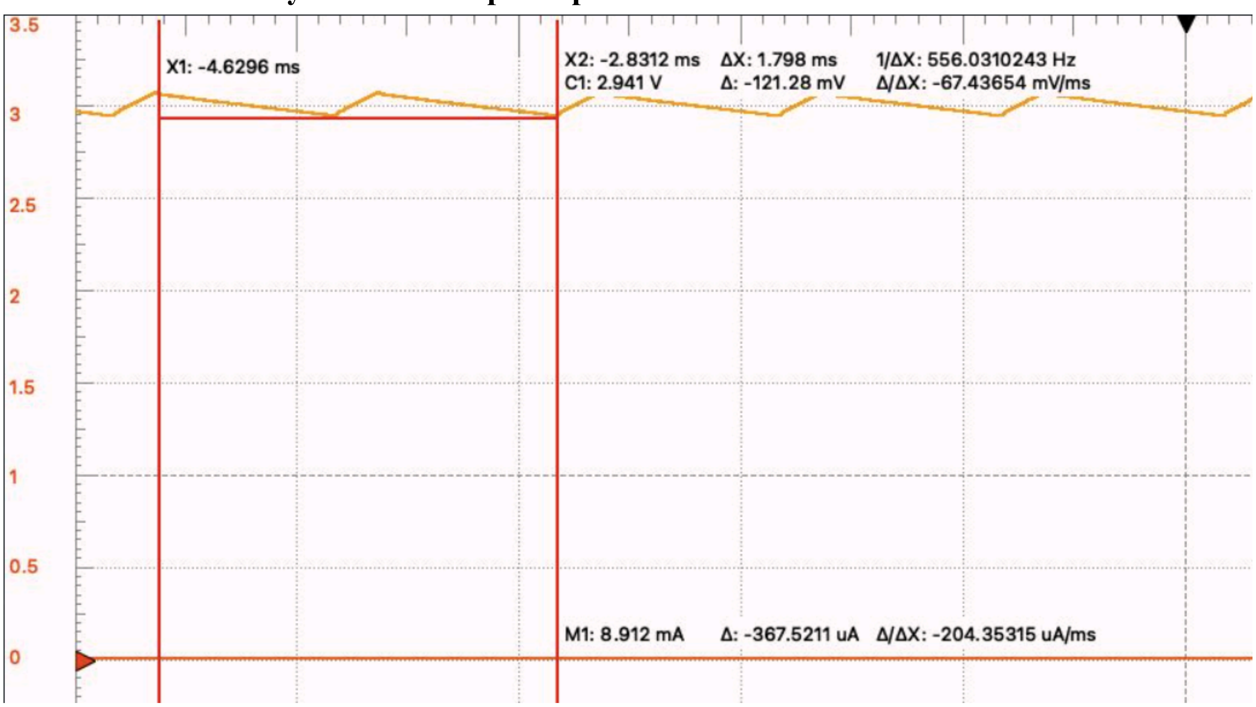

Figure 6: Oscilloscope in WaveForms Software showing Output Voltage in Orange and Output Current in Red)

Simulation: (LTSpice)

I. Circuit Schematic

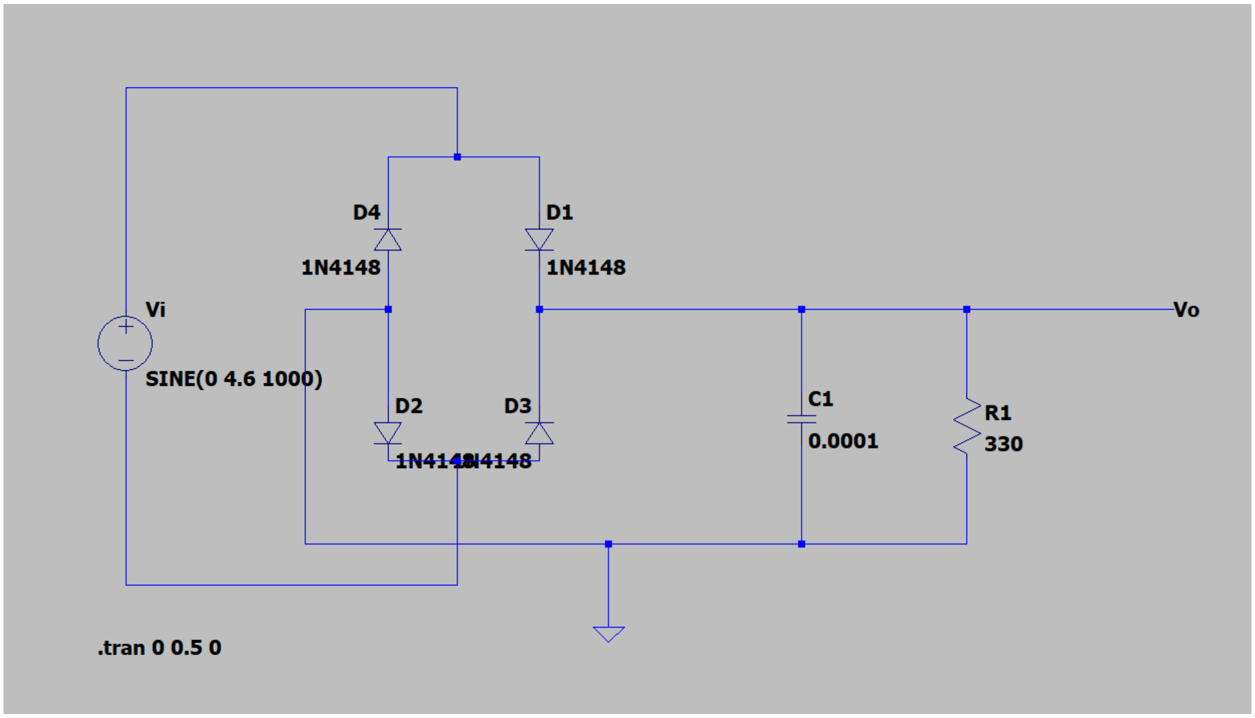

Figure 7: Updated LTSpice Circuit Schematic

II. Netlist:

SPICE Netlist: C:\Users\hasee\OneDrive\Documents\LTspice\2eiProjects\p1.net

```
* C:\Users\hasee\OneDrive\Documents\LTspice\2eiProjects\p1.asc
Vi N001 N002 SINE(0 4.6 1000)
C1 Vo 0 0.0001
R1 Vo 0 330
D1 N001 Vo 1N4148
D2 0 N002 1N4148
D3 N002 Vo 1N4148
D4 0 N001 1N4148
.model D D
.lib C:\Users\hasee\AppData\Local\LTspice\lib\cmp\standard.dio
.tran 0 0.5 0
.backanno
.end
```

Figure 8: LTSpice Netlist for Circuit Schematic

III. Simulation Conditions

The type of simulation that was used was transient analysis with a simulation time of 2 seconds and the diode model for all four diodes was the 1N4148 diode.

IV. Simulation Output:

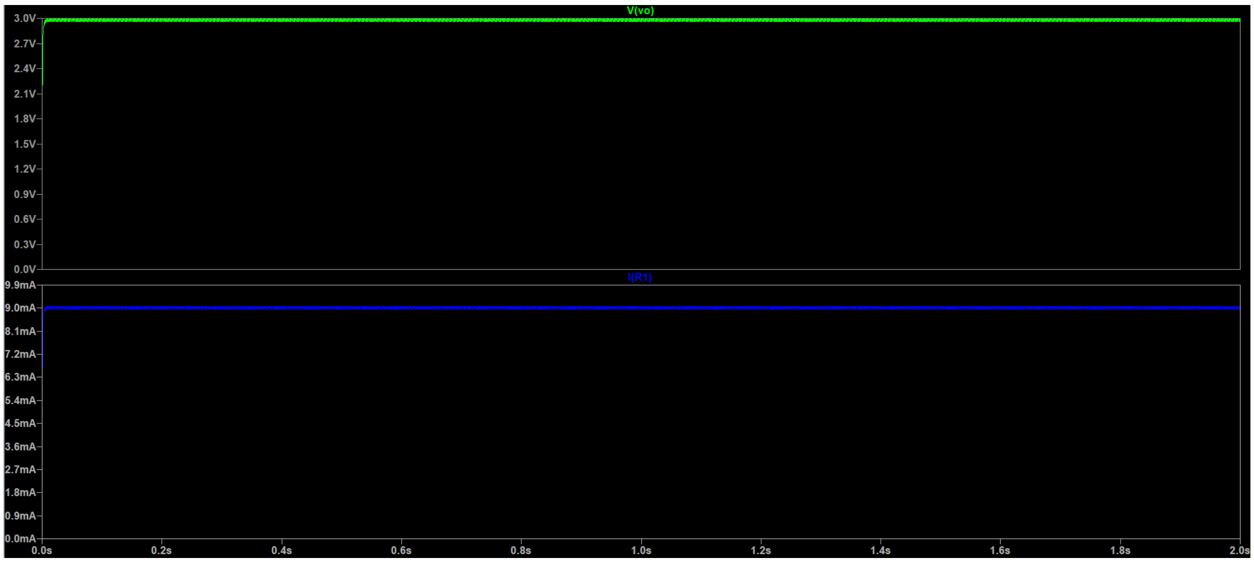

Figure 9: Simulation output for Output voltage, Input Voltage and Output Current

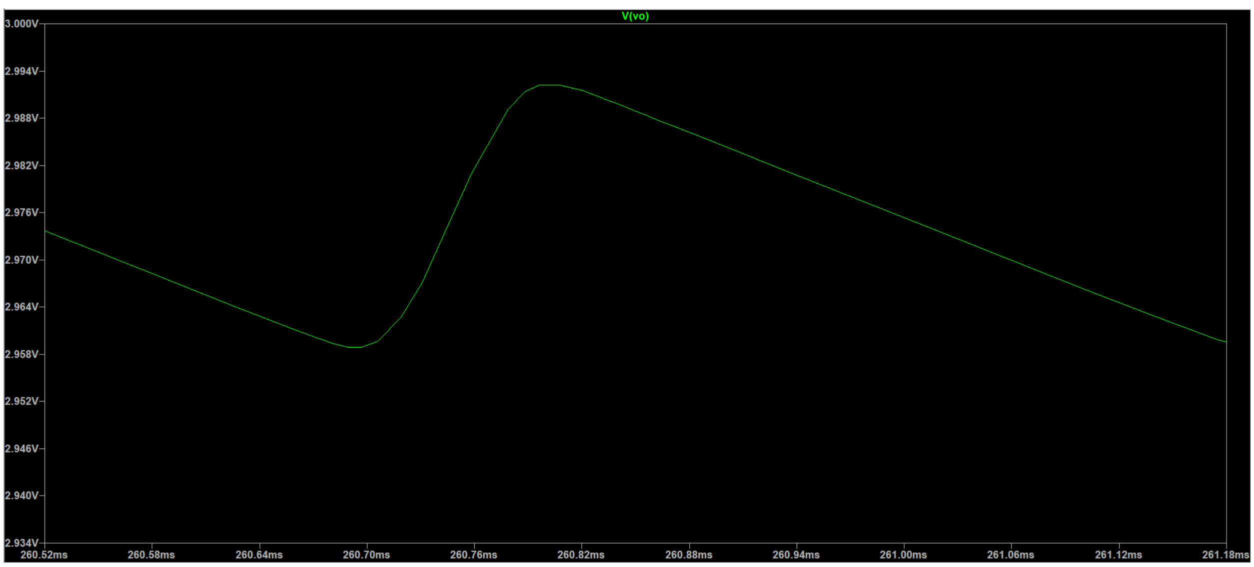

Figure 10: Close-up Simulation Output for Output Voltage

Figure 10 shows the Output voltage range and that it meets design specification of a desired range of 2.9V to 3.1V as the minimum output voltage equals approximately 2.958 V and the maximum output voltage range equals approximately 2.994 V.

Discussion

I. Results from design, simulation, and measurement.

The results between design, simulation and measurement have slight variations. Based on my theoretical calculations, the value of the resistance should equal 300 ohms, capacitor value should equal 25 uF, AC Input voltage should have a 4.6 V amplitude to meet the design specification which require the output voltage to be between the range of 2.9 V to 3.1 V and output current being 10 mA. The purpose of the simulation was to verify that the design and component values yield a desirable result before constructing the actual circuit. In the Simulation, different component values from the design were used due to not having access to some of the components with values required for the design. Through the use of simulation, we verified the results based on new values of resistor and capacitor that are expected to yield a similar behaviour than the design. In the simulation, a 330 ohms resistor was used instead of the 300 ohms as recommended by the course instructor, 100 microfarads capacitor was used instead of 25 microfarads capacitor as a high voltage value only decreases the ripple voltage which results in the voltage curve being a more smoother DC output and the same AC input voltage with 4.6 V amplitude was used. After executing the simulation, the output voltage was in the range of approximately 2.958 V to 2.994 V meeting the design requirement and the output current was approximately 9 mA which does not meet the desired output current of 10 mA as a higher resistor value of 330 ohms was used in the simulation. In the actual physical circuit, Same input variable including the AC input of 4.6 V amplitude was used. Through the use of Analog Discovery 3 and WaveForms software, the oscilloscope measured the output voltage of approximately 2.941 V to 3.06228 V meeting the design requirement and the output current was approximately 8.912 mA which was measured through the use of ohm's law and is lower than the desired output current of 10 mA due to using a higher value resistor than the design.

II. Discrepancies observed.

The discrepancy between the design and simulation was found in the output current which is the current across the resistor. Since the value of the resistor in the design was calculated to equal 300 ohms and in the simulation a 330 ohms resistor was used due to not being provided with a 300 ohms resistor in the components kits, this discrepancy was expected as a higher value resistor would have higher voltage drop and would result in a lower current flowing through the resistor. A slight discrepancy was also observed between the simulation output and the oscilloscope output which is always expected because simulator software gives ideal results whereas, real life hardware implementation of a circuit would never be ideal due to many factors such as in my case, internal resistance in components, varying forwards voltage drop across diodes and more. However, simulation results and oscilloscope results were very similar which demonstrates that the physical circuit was well constructed and the experiment was successful.

III. Limitations of the design and the limitations of the measurements.

The most major limitation of the design was not being provided a 330 ohms resistor in the component kits and were recommended to use 330 ohms resistor instead of the desired 300 ohms resistor. This largely affects the output current as it ideally reduces the output current from 10 mA to 9mA due to a higher voltage drop across the resistor. 300 ohms could be considered the maximum resistance value to yield an output current of approximately 10 mA. Additionally, another limitation was only one type of diode that could be used for the circuit which was the 1N4148 model as it had forward voltage drop that could vary from 0.8 to 1V which could contribute to yielding slightly inaccurate results. Lastly, one limitation of the measurement is not being provided with dedicated ammeter in the AD3 and Waveforms software as a custom math channel had to be used instead which divided the output voltage with 330 according to ohm's law to give the output current which may be slightly inaccurate than the actual current.

IV. Problems encountered in measurements and the troubleshooting steps you took

Many problems and issues were encountered in construction of physical circuits and using AD3 to obtain the output voltage and current. One problem I encountered was getting a lower output voltage than the desired 2.9 V to 3.1 V. To find a solution to these problems, I had to check if the circuit was constructed properly and checked for any loose or wrong connections. However, the circuit was well constructed and nothing seemed off. After comparing my physical circuit with the circuit schematic to see if both match, I realized I needed a second AC input wave with a phase difference of 180 degrees based on my circuit schematic that could supply voltage on a different connection of diodes as there are two nodes where the AC input voltage source is connected.

References

[1] A.S. Sedra, K. C. Smith, T.C. Carusone, and V. Gaudet, Microelectronic circuits, 8th ed. New York, NY: Oxford University Press, 2019.

[2] ABRA electronics, “1N4148 Diode Small Signal Fast Switching 0.3A 100V,” https://abra-electronics.com/ics-semiconductors/diodes-rectifiers/switching-diodes/1n4148-diodesmall-signal-fast-switching-03a-100v-1n4148.html (accessed Feb. 8, 2025).

[3] “Half wave and full wave rectifier: Function, comparison, and applications,” ROHM, https://www.rohm.com/electronics-basics/ac-dc/rectification (accessed Feb. 8, 2025).