
PROJECT 1: AC TO DC POWER SUPPLY

ELECENG 2E14 – Electronic Devices and Circuits I

Section C01

Viha Tripathi (tripav2)

400504615

Submitted: February 15th, 2025

Course Instructors: Dr. Yaser M. Haddara

The student is responsible for performing the required work in an honest manner, without plagiarism and cheating. Submitting this work with my name and student number is a statement and understanding that this work is my own and adheres to the Academic Integrity Policy of McMaster University.

Viha Tripathi

400504615

Summary

The objective of this project is to design and build a DC power supply that gives a current of 10mA at a voltage of $3V \pm 0.1V$, taken from a 120V (rms), 1 kHz supply source. The design consists of four key components: a transformer (for calculation purposes), a rectifier, a filter, and a regulator, which I opted not to use. In practical use, a function generator will replace the transformer to be used for testing purposes. The circuit is engineered to reduce ripple and maintain a constant output voltage.

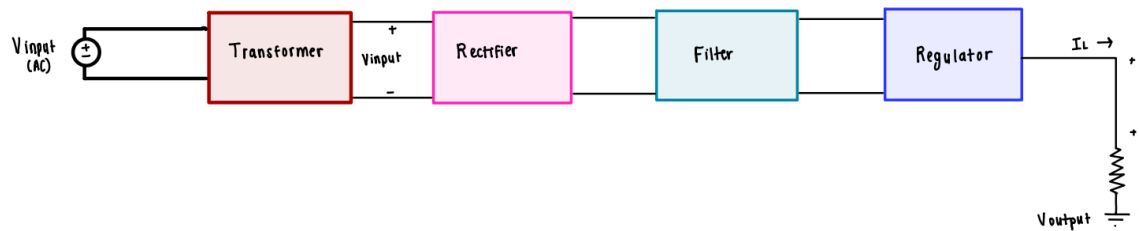


Figure 1: Block Diagram

I) Transformer

Although this design does not use a physical transformer, we simulate its action using the application of a suitable input to the circuit. With a turns ratio of around 22:1, a sinusoidal input of a peak voltage of 7.6V is obtained, or a root mean squared voltage of 5.37V. The input voltage of 7.6V was simulated using the Analog Discovery 2's integrated DC power supply. The transformer theoretically used in this project is a center-tapped transformer.

II) Rectifier

In the rectifying process, I chose a center-tapped full-wave rectifier for simplicity and efficiency. In this design, two diodes are connected to two different positive and negative voltage sources of equal magnitude. Compared to a four-diode bridge rectifier, a two-diode configuration of a center-tapped arrangement reduces the number of components used and minimizes experimental error potential. The use of a center-tapped setup is also in line with that of a center-tapped transformer, which gives two identical opposite-phase alternating current (AC) outputs. One half of a transformer's output is used at any given time, providing a unipolar waveform of output. The forward voltage drop of each of the rectifier's diodes must be considered in determining resulting output voltage, approximately $V_d = 0.7$.

III) Filter

For this experiment, I opted for a 100 uF capacitor in parallel with the load (310 Ω resistor).

IV) Regulator

A 3V Zener diode would be used to maintain a stable 3V output. However, I opted not to use a regulator in this circuit as it is not required.

V) Circuit Schematic

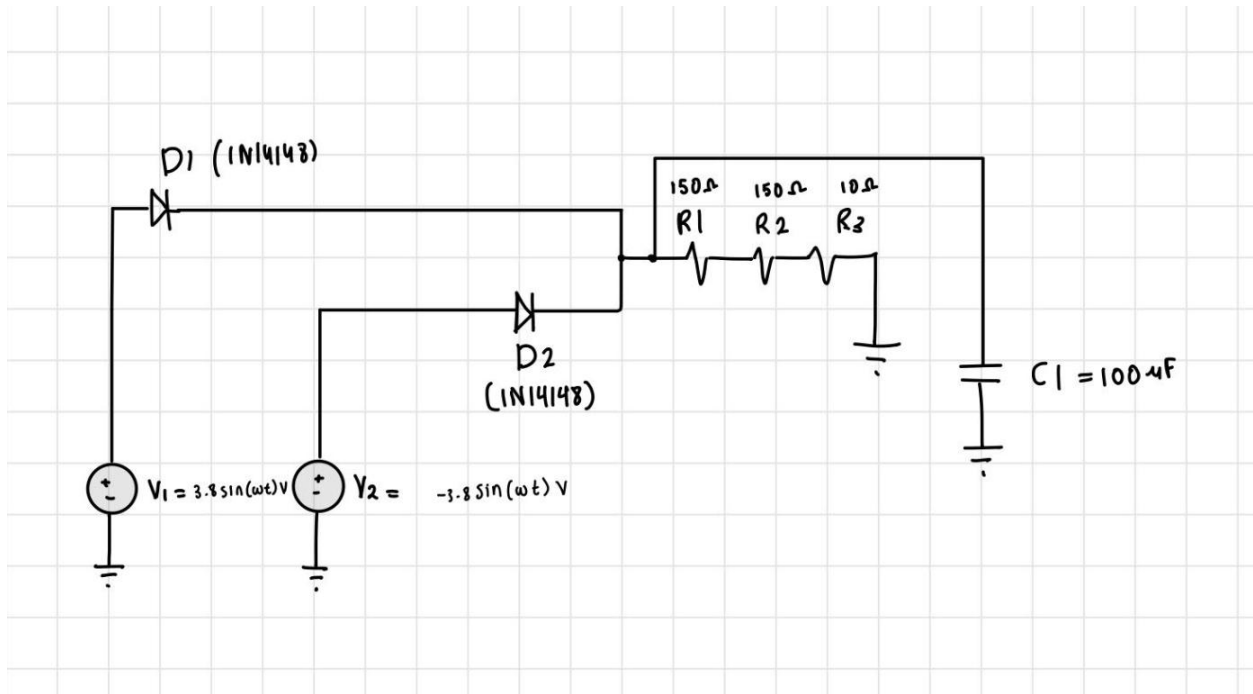


Figure 2: Circuit Schematic

VI) Component Calculations & Performance Expectations

Load Resistor

$$R = \frac{V}{I} = 300 \Omega \pm 10 \Omega = 310 \Omega$$

Rectifier

$$V_{DC} = V_{max} - V_D$$

$$V_{max} = V_{DC} + V_D$$

$$V_{max} = 3 V + 0.7 V$$

$$V_{max} = 3.7 \text{ V} + 0.1 \text{ V} = 3.8 \text{ V}$$

Therefore, the rectifier circuit inputs are:

$$V_1 = 3.8 \sin(wt)$$

$$V_2 = -3.8 \sin(wt)$$

Filter (capacitor)

$$I = 10 \text{ mA}$$

$$f = 1 \text{ kHz}$$

$$V_{ripple} = 0.1 \text{ V (to keep within spec)}$$

$$C = \frac{10 \text{ mA}}{1 * 0.1} = 50 \mu\text{F}$$

$$C = \frac{I_{out}}{f * V_{ripple}}$$

A 50 uF capacitance is the minimum capacitance required. A 100 uF capacitor will be used for a smooth output.

Diodes (assuming ideal)

$$V_D = 0.7 \text{ V}$$

Transformer

$$V_o = 2 * V_1$$

$$V_o = 2 * (3.8 \sin(wt))$$

$$V_o = 7.6 * \sin(wt)$$

RMS conversion:

$$V_{oRMS} = \frac{V_o}{\sqrt{2}}$$

$$V_{oRMS} = \frac{7.6}{\sqrt{2}}$$

$$V_{oRMS} = 5.37 \text{ V}$$

Turns Ratio

$$\text{Turns ratio} = \frac{N_1}{N_2}$$

$$\text{Turns ratio} = \frac{V_s}{V_o}$$

$$\text{Turns ratio} = \frac{120\text{ V}}{5.37\text{ V}}$$

$$\text{Turns ratio} = 22 : 1$$

VII) Expected Performance

According to calculations, the design requires that the input signal of the circuit used should be of an amplitude of 3.8V and a frequency of 1 kHz. The use of two such waveforms is required, each connected to one of the two center-tapped rectifier's diodes. The waveforms should be 180° out of phase; hence, when one waveform is in a half-cycle of a positive nature, the other is in a half-cycle of a negative nature, and vice versa. Such phase opposition ensures that when D1 is in a conduction condition, D2 is in a cut-off condition, and when D1 is in a cut-off condition, D2 is in a conduction condition. This switching action results in a waveform that is completely rectified at the output.

In order to compensate for the voltage drop of 0.7 V across diodes, the input amplitude was set at 3.8V rather than the required 3.0V. The adjustment ensures that rectified voltage, after accounting for the diode drop, reaches the desired voltage of 3.0V. The use of a capacitor in the circuit smooths the rectified waveform, hence reducing ripple to $\pm 0.1\text{V}$ of the desired voltage.

VIII) Design Considerations

Resistor

The first trade off that had to be made was to use two 150 Ω and one 10 Ω resistor rather than a 300 Ω resistor. Using 310 Ω instead of 300 ohms increased safety by slightly reducing current, minimizing heat dissipation, and ensuring components operate within safe limits.

Capacitor

The minimum capacitance calculated was 50 μF . However, this was not available, therefore, a 100 μF capacitor was used, which does not impact the circuit negatively but instead allows for smoother operation.

Power dissipation

$$P = VI$$

$$P = 3.1\text{V} * 10\text{mA}$$

$$P = 0.031\text{ W}$$

The resistors being used have a power rating of 0.25. Therefore, they may be used as 0.25 is less than the maximum power dissipated by the resistor.

Measurement and Analysis

I) Photograph of Actual Circuit

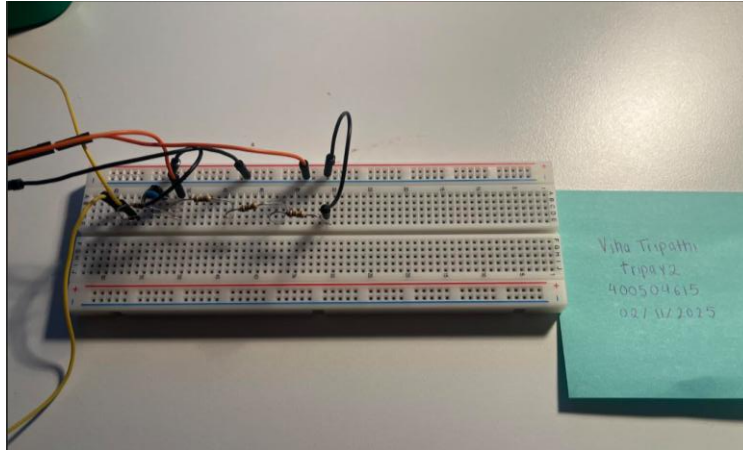


Figure 3: Physical Circuit

II: Measurement Procedure

To determine the performance of the circuit, the Waveforms software and AD3 was used. The waveform generator and oscilloscope was used to show the rectified wave after going through the center tapped full wave rectifier. The voltage drop through the load resistor as well as the voltage at the nodes where the diodes were connected. Along with this, the current was measured using the math function of the oscilloscope. By taking the channel 1 output (voltage) and dividing it by 310, the current wave was displayed.

III) Measurement Results

The waveform generator provided a sine wave input of a voltage of 3.8V and a frequency of 1 kHz, one of which was phase-shifted by 180 degrees to provide the rectifier function. The use of a vertical offset was not utilized.

The initial voltage across the load resistor was about 2.9 V, which is in the range of acceptable voltage of $3.0 \text{ V} \pm 0.1 \text{ V}$.

With a load resistor of 310Ω , the measured current was in the range of 9.8 A to 9.9 A. The range was expected given that there is a small variance in resistor value and is in line with our design tolerances.

VIII) Oscilloscope Output

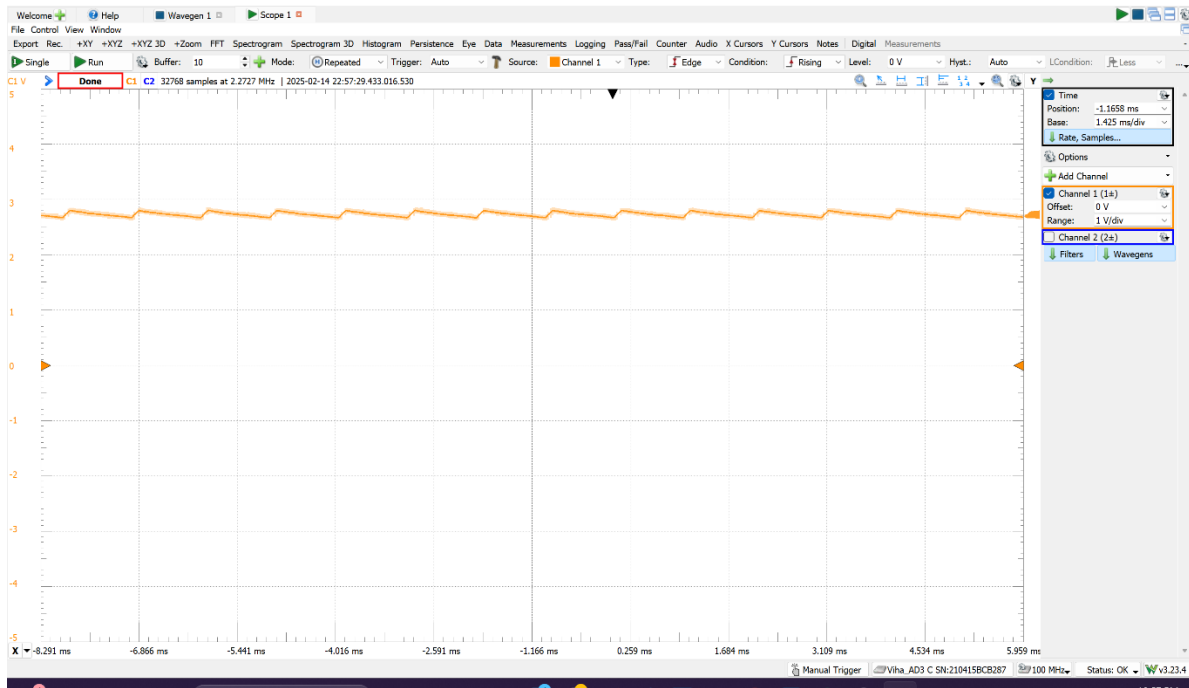


Figure 4: Oscilloscope Output Voltage

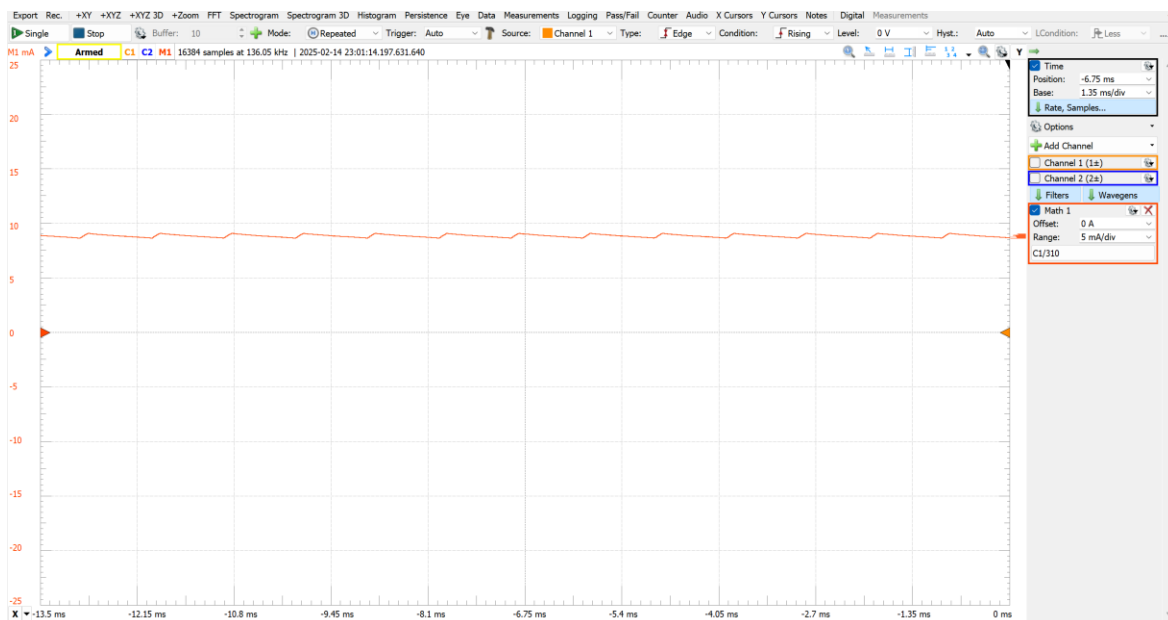


Figure 5: Oscilloscope Output Current

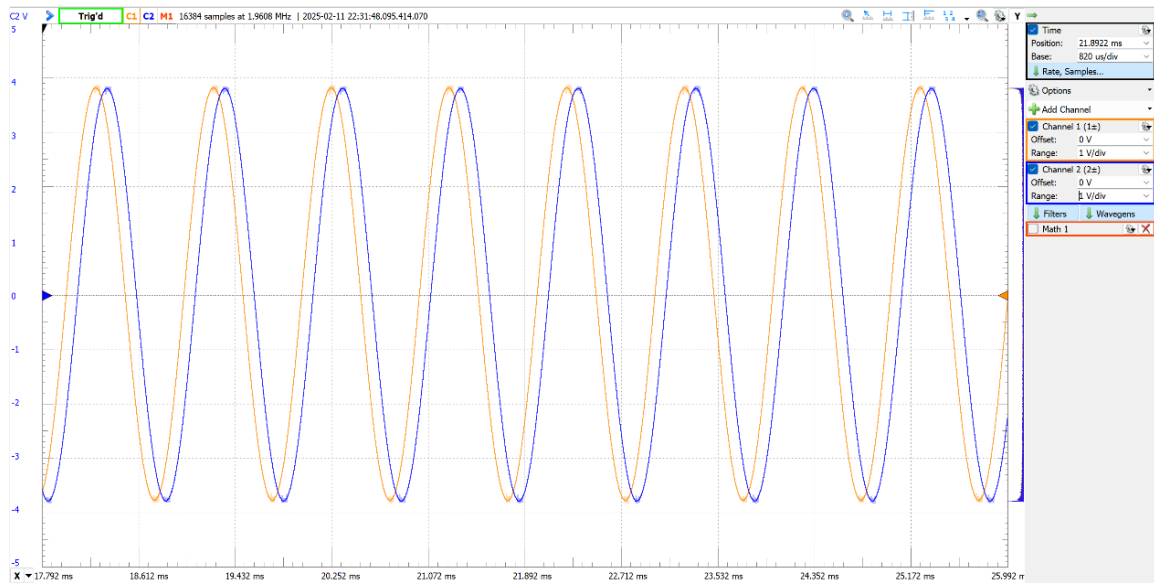


Figure 6: Oscilloscope Input Sine Voltages

Simulation

I) Circuit Schematic From Simulator

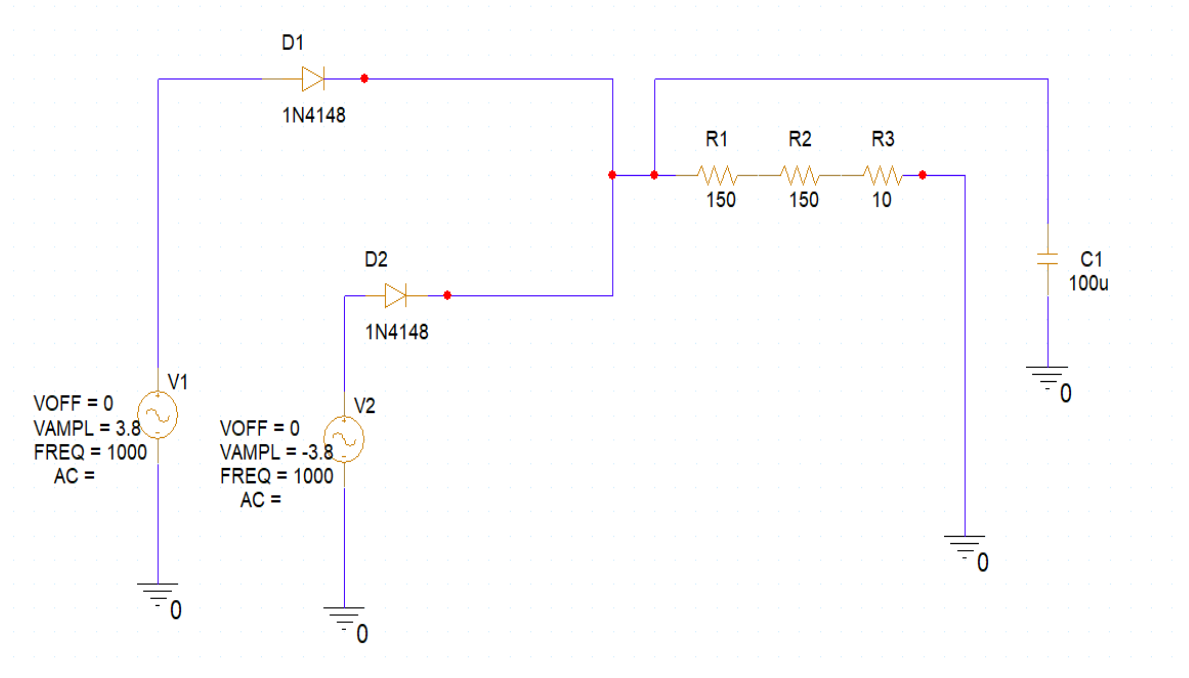


Figure 7: PSpice Circuit Schematic

II) Netlist

```

1: * source PROJECT 1 CIRCUIT
2: V_V1      N00088 0
3: +SIN 0 3.8 1000 0 0 0
4: V_V2      N00105 0
5: +SIN 0 -3.8 1000 0 0 0
6: R_R1      N00097 0 310 TC=0,0
7: X_D1      N00088 N00097 awb1n4148 PARAMS:
8: X_D2      N00105 N00097 awb1n4148 PARAMS:
9: C_C1      0 N00097 100u TC=0,0
10:

```

Figure 8: Netlist

III) Simulation Conditions

This simulation was performed as a transient sweep, and data was recorded from 0 to 0.1 ms with maximum step size of 0.1ms.

IV) Simulation Output

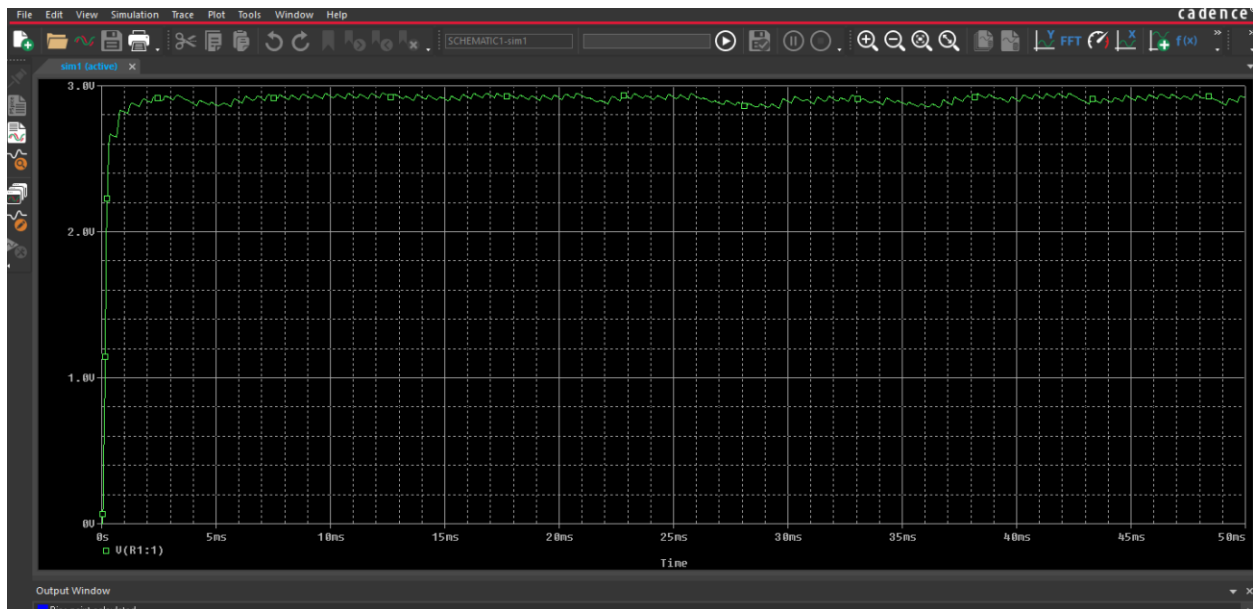


Figure 9: PSpice Output Voltage Graph

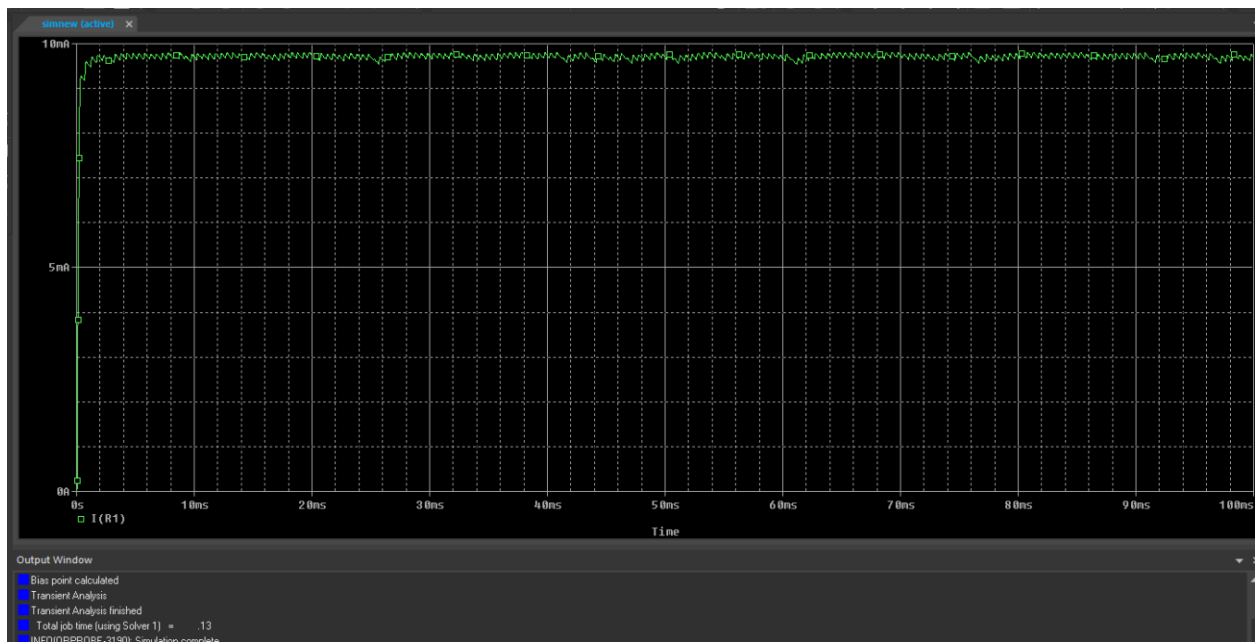


Figure 10: PSpice Output Current Graph

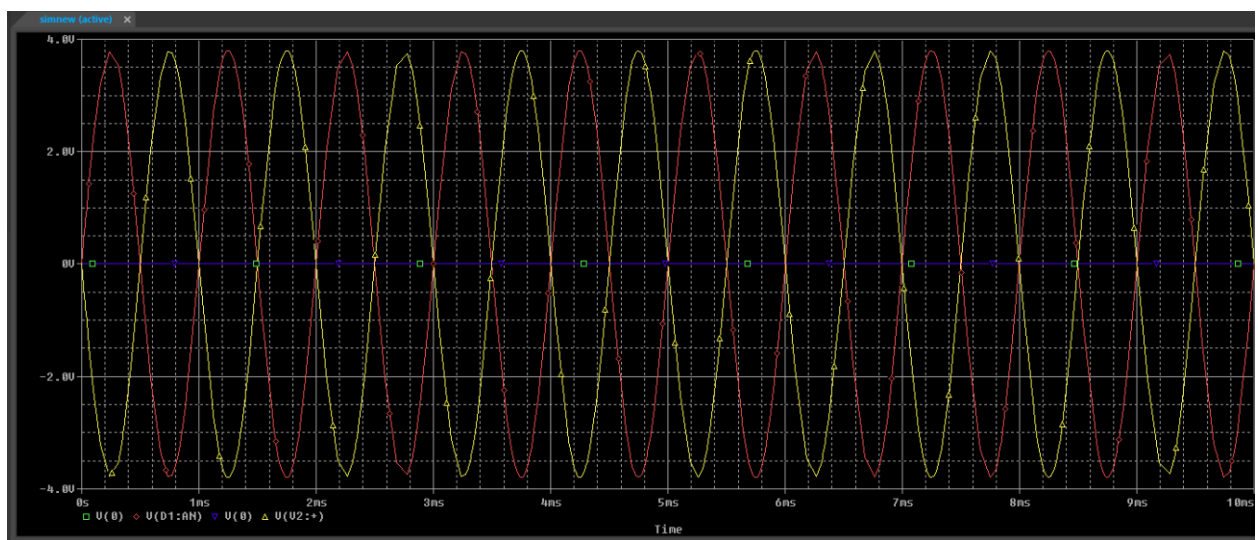


Figure 11: PSpice Input Voltages Graph

Discussion

I) Results and discrepancies

	V_{out} (V)	I_{out} (mA)
Simulation	2.9	9.9
Measured	3.1	10
Actual	2.85	9.1

Here are the percent differences between the simulated, measured, and actual values for both output voltage (V_{out}) and output current (I_{out}):

For Output Voltage (V_{out})

- Simulation vs. Actual: 1.75%
- Measured vs. Actual: 8.77%

For Output Current (I_{out})

- Simulation vs. Actual: 8.79%
- Measured vs. Actual: 9.89%

The disparities found between simulated, measured, and actual values stem from an abundance of influences. The relatively minor voltage deviation seen between simulated versus actual measurements (1.75%) shows that simulation accurately modeled the expected circuit behavior. In contrast, the greater deviation found in terms of current (8.79%) implies that some real world influences, such as component tolerance, variation in resistor values, or leakage in capacitor values, did not accurately get captured by the simulation. Larger deviation found by the measured values (8.77% voltage, 9.89% current) stems from various influences. Resistor tolerance, as well as capacitor tolerance, causes minor changes that impact both voltage measurements and current measurements. Despite these inaccuracies, the measured results fall well within an acceptable threshold, validating the conclusion that the circuit operates as expected. While error by the measurable result by current is greater, it does not always signify poor circuit design, but it does show the need to account for practical realities.

II) Design Limitations

A limitation was the absence of an inductor in the filtering stage. An LC filter would have offered superior ripple rejection and overall circuit efficiency than the capacitor filter implemented. Another limitation is the 1N4148 diodes which had to be used. They have a forward voltage drop of 1 V, which is different from the theoretically assumed value of 0.7, making it inconsistent with my theoretical calculations. Along with this, since the AD3 is not capable of calculating current directly and needs the implementation of the math function, this posed as a limitation. A more accurate reading of the current could have been obtained if current could be directly measured as there may have been some discrepancies in resistance of the resistors.

III) Problems encountered and troubleshooting

Many challenges were encountered during design, simulation, and implementation of the circuit. One of the key challenges was interference caused by noise that affected readings on the oscilloscope and made it more difficult to obtain accurate voltage and current readings. Also, unstable or loose connections on the breadboard caused some trouble, requiring careful inspections and reassembly. Another complication was caused by initial misplacement of diodes in the rectifier that made it ineffective in converting AC to DC, a problem that required rectification. It is crucial to note that diode orientation is important and must be correctly oriented in the circuit—an aspect that was initially neglected. The measurement of small ripple voltage was also a challenge due to the limitation of sensitivity of the oscilloscope, requiring careful adjustment of the probe settings. Lastly, the process of selecting capacitors required iterative processes to necessitated troubleshooting processes such as improvement of measurement procedures, and adjustment of component values to get desired performance results.

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.