Design Project 1

AC – DC Power Supply

ELECENG 2EI4

Dr. Yasar M. Haddara

February 9th, 2025

Aaron Ghosh - ghosha20 – 400512786

Summary:

The goal for this design project is to create a DC power supply, capable of producing a current of 10mA at 3V ± 0.1V from a 120V voltage source at 1 kHz. In order to accomplish this, the circuit required a rectifier, filter, and potentially a regulator to convert AC voltage input into a DC output. The circuit was designed and simulated on LTspice and physically tested using the AD3's function generator to provide the sinusoidal amplitude, bypassing the need to use a transformer. The rest of the circuit want built using various components from the 2EI4 kit.

Design:

Circuit Schematic:

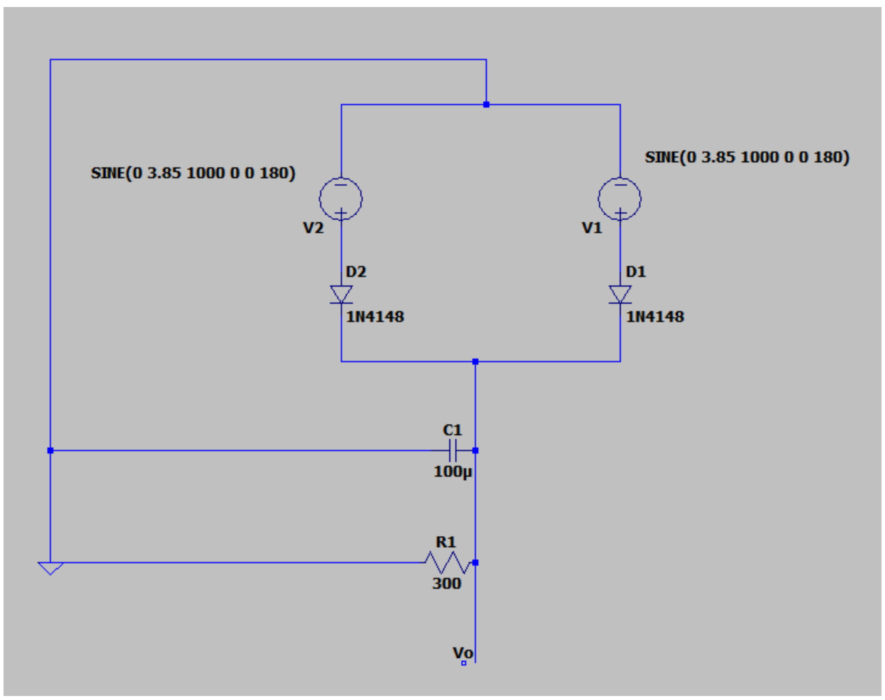

Figure 1: Circuit schematic designed on LTspice

Load Resistor:

With the use of Ohm's Law, the effective value for the load resistor can be determined using the required current (10mA) and voltage (3V). This provdes us with the resistance of 300 Ω.

$$R = \frac{V}{I} = \frac{3\text{V}}{10\text{mA}} = 300\Omega$$

For the physical demonstration aspect of this project, two 150 Ω resistors were used in series as there was no 300 Ω resistor available.

$$Req = 150\Omega + 150\Omega = 300\Omega$$

Filter:

The filter used for this project is a parallel RC circuit. This design for a filter was chosen as the components were already available as they were left over from a previous course (2CI4). An alternative option for a filter would be an LC circuit, however a RC would suit our purpose better as it handles low voltage inputs much smoother than an LC circuit. LC circuits also tend to generate more heat and cost more.

The following formulas were used to determine the capacitance value for the capacitor based on the design requirements for this project.

$$Vrpp = \frac{I}{fC}$$

$$C = \frac{I}{fVrpp}$$

By rearranging the peak-to-peak voltage, we can find the capacitance value with respect to the current (10mA).

$$f = 2fin = 2(1000)Hz$$

The peak-to-peak ripple voltage is 0.2V as the tolerance ± 0.1 is only the amplitude which can be doubled, representing the full voltage range.

$$C = \frac{I}{fVrpp} = \frac{10mA}{2000Hz \times 0.2V} = 25\mu F$$

After plugging everything together, we can see that we need at least a 25µF capacitor to be within our required range, however, this is an ideal situation and does not take into account any fluctuations that come with physical components. Therefore, to allow us to create an operationally physical demonstration. A larger capacitor was used, ensuring that there is a smoother DC voltage output and account for any fluctuations. A 100µF capacitor was chosen for this purpose.

With a new capacitor, it can be seen that the following tolerance of ± 0.025V, results with a peak-to-peak voltage value of 0.05V.

$$Vrpp = \frac{10mA}{2000Hz \times 100\mu F} = 0.05V$$

Rectifier:

The rectifier used for this project is called a full-wave center-tapped rectifier (FWCT). It was chosen because it can convert AC voltage to DC voltage very efficiently. A FWCT also has the benefit of lowering the ripple voltage as both the positive and negative half-cycles of the AC waveform are used for rectification, allowing it to smooth out the output. It was also chosen for simplicity as it only required two diodes to implement, making it a strong idea with our limited resources.

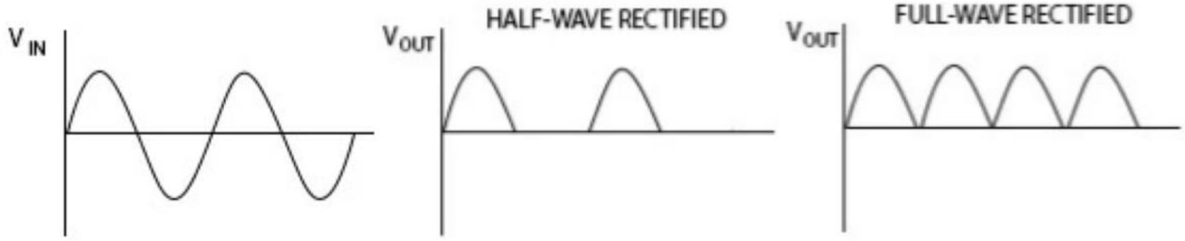

Figure 2: Half-wave vs Full-wave rectifier

As seen in the figure above, a full-wave rectifier has double the frequency, making the entire process of smoothing and filtering out fluctuation much simpler.

Even with a full-wave rectifier, there are two possible ways of moving forward and implementing the design. A full wave bridge or a full wave center tapped rectifier. For this project, the FWCT rectifier was chosen for its simplicity. It only required two diodes compared to the four needed for the full wave bridge. Moreover, it was more efficient as each half-wave of the sine wave passed through two diodes instead of four. The disadvantages and limitations of the FWCT rectifier are beyond the scope of this project.

To determine the input voltage for the rectifier, the following calculations and values were considered.

$$Vo = 3V$$

$$Vp = Vo + \frac{Vrpp}{2} = 3 + \frac{0.05}{2} + 0.7 = 3.725V$$

$$+V = 3.725\sin(wt)$$

$$-V = -3.725\sin(wt) \ \boldsymbol{or} \ -3.725\sin(wt + 180°)$$

Vo represents the output voltage at the load and Vp is the peak voltage which also accounts for the tolerance determined previously. The 1N4148 diode was used for the rectifier and has a breakdown voltage at 100V. As per the diode data sheet, the voltage drop at the diode is 0.7V at 10mA, indicating that the diode is within specifications and will operate safely in the design.

Regulator:

For this project, incorporating a rectifier was unnecessary as the final output already met the requirements. However, in larger power supply designs, a regulator plays a crucial role by stabilizing voltage fluctuations, preventing potential risks like short circuits and overheating. If left unregulated, voltage can exceed the required amount (voltage spikes), leading to circuit damage. For our circumstances, a 3V reverse-bias Zener diode in parallel with the load would capped the voltage at 3V. However, because the input voltage was managed by the AD3 function generator, the output voltage remains within the stable range, making a regulator unnecessary.

Transformer:

For the sake of this project, a transformer will not be used but only be designed. It is represented by two sinusoid waveforms with amplitudes of 3.85 V at 1kHz with a phase shift of 180°.

To find the coil ratio, all the calculations must be done respect to in root mean square (rms) values as this is the standard for transformers. The input voltage is 120V AC rms, however our waveforms Vp of 3.725V is not in rms and must be converted before continuing, as seen below.

$$Vo = 2Vp$$

$$Vrms = \frac{2A}{\sqrt{2}} = \frac{3.725}{\sqrt{2}} \approx 5.268V$$

$$\frac{N_{primary}}{N_{secondary}} = \frac{V_{primary}}{V_{secondary}}$$

$$\frac{N_{primary}}{N_{secondary}} = \frac{120V}{5.268V} \approx 22.779 \rightarrow 23 : 1$$

Therefore, approximately 23 primary turns (120V rms) are needed for every 1 secondary turn to achieve Vrms = 2.634V, resulting in a peak voltage of 3.725V.

Measurement & Analysis:

Physical Circuit:

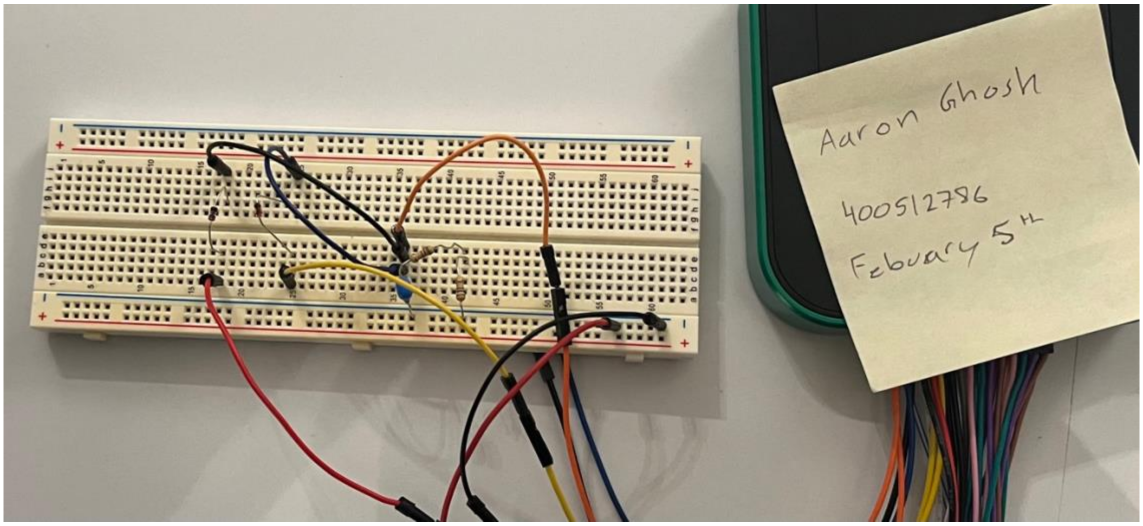

Figure 3: Physical circuit

Procedure:

To measure the output voltage and current, measurements were taken at node Vo as per the circuit schematic diagram in Figure 1. The voltage was displayed using channel 2 and the load current was displayed using the math channel (C2/300), representing (Vo/300Ω) (See Figure 5).

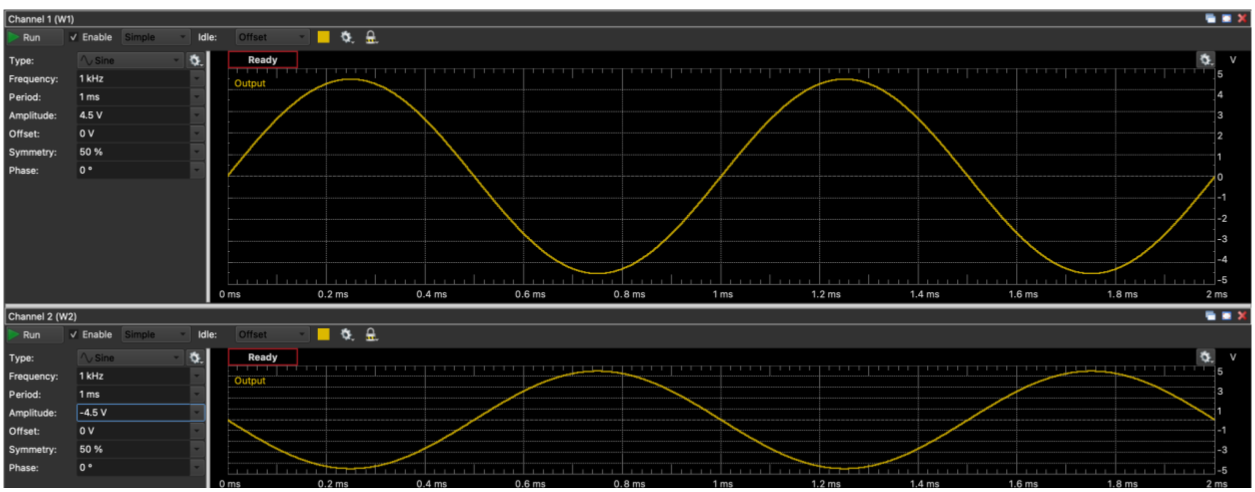

Figure 4: AD3 input voltages on Waveforms

To ensure that every component in the circuit works as intended, multiple measurements were taken throughout the entire design process at each step. This ensures that the circuit is correctly implemented, starting with the rectifier. The rectifier's design was tested first as it would need to correctly rectify the AC current before it can be filtered. Afterwards, the filter was tested and implemented, followed by the load resistor. This step-by-step procedure made the entire process more streamlined and troubleshooting a lot easier. It also added an extra layer of safety as the entire design would be implemented correctly as every step, ensuring nothing can get damaged.

Measurement:

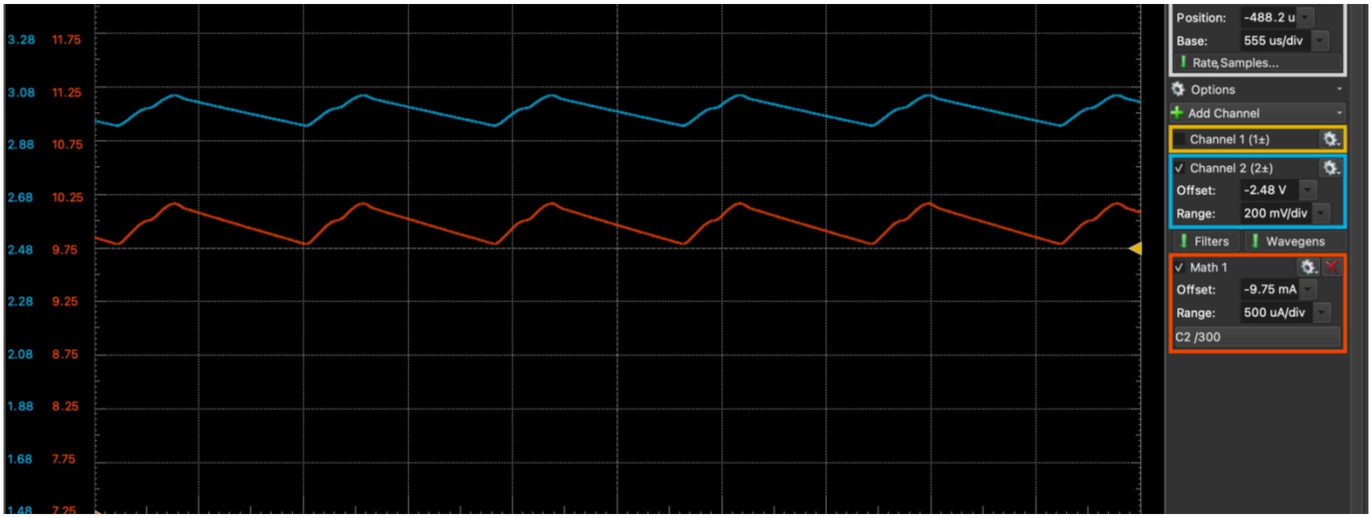

Figure 5: AD3 oscilloscope readings (blue is output voltage) (red is load current)

The two key measurements that were taken are Vo and I_L. The output voltage was determined through the AD3 oscilloscope reading as shown in Figure 5. Also shown is the current which was calculated and displayed using the math channel (Vo/300Ω).

$$Vo = 3.06V \; - 2.94V$$
$$I_L = 10.2mA \; - 9.8mA$$

$$Vrpp = 3.06 \; - 2.94 = 0.12V$$
$$Tolerance = \frac{Vrpp}{2} = \frac{0.12}{2} = 0.06V$$

The difference in the calculated theoretical voltage tolerance and actual measured circuit tolerance is expected as there is variation and inaccuracy with real components, specifically caused by the resistor, diodes, and capacitor.

The percent error can be calculated using the theoretical and measured values.

$$Error\,\% = \frac{V_{\text{Theoretical}}}{V_{Measured}} \times \; 100\% = \frac{3.725}{4.5} \times 100\% \approx 83\%$$

In order to stay within the output range, the voltage amplitude was increased from 3.725 to 4.5 (Figure 4) to account for the resistor, diodes and capacitor as they operate within their own tolerance ranges which can impact the results.

Simulation:

Simulated Circuit:

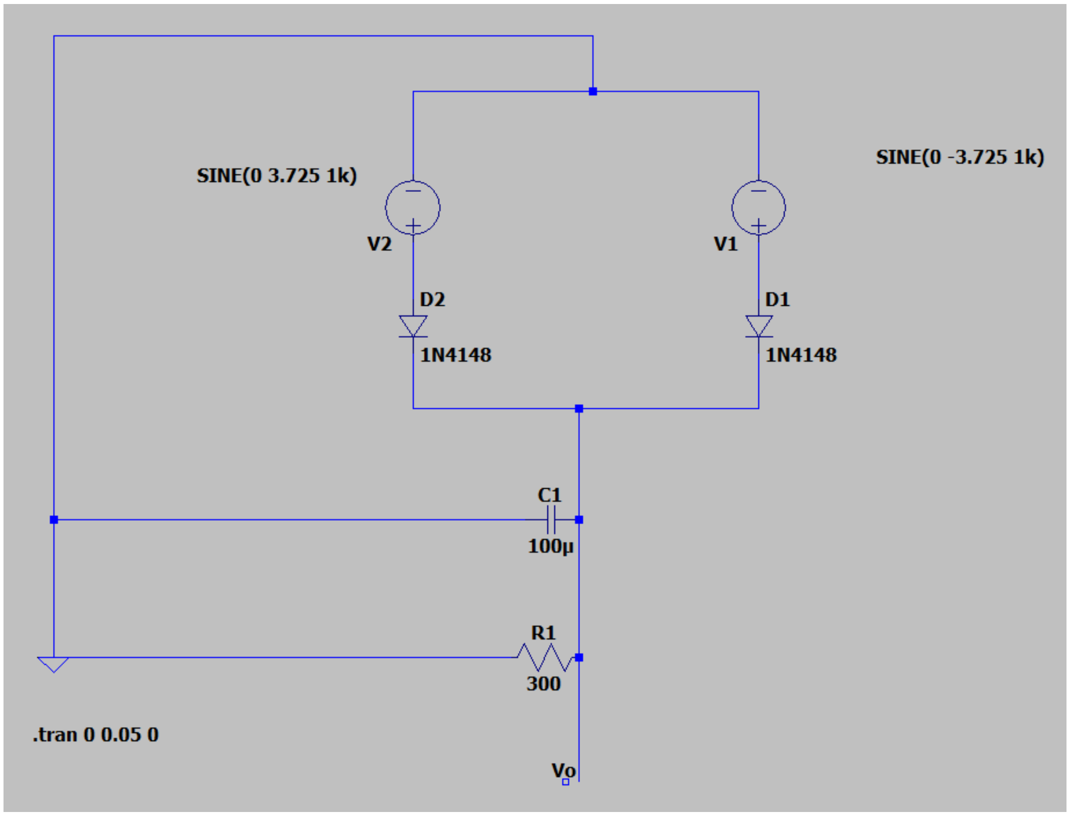

Figure 6: Simulated circuit schematic

Simulation Conditions:

Transient circuit simulation ran for 50ms, using two 1N4148 diodes.

Netlist:

* C:\Users\aaron\Documents\School\ELECENG2EI4\LTspice\project1.asc
V1 N002 0 SINE(0 -3.725 1k)
V2 N001 0 SINE(0 3.725 1k)
C1 N003 0 100µ
R1 N003 0 300
D1 N002 N003 1N4148
D2 N001 N003 1N4148
.model D D
.lib C:\Users\aaron\AppData\Local\LTspice\lib\cmp\standard.dio
.tran 0 0.05 0
.backanno
.end

Simulation Results:

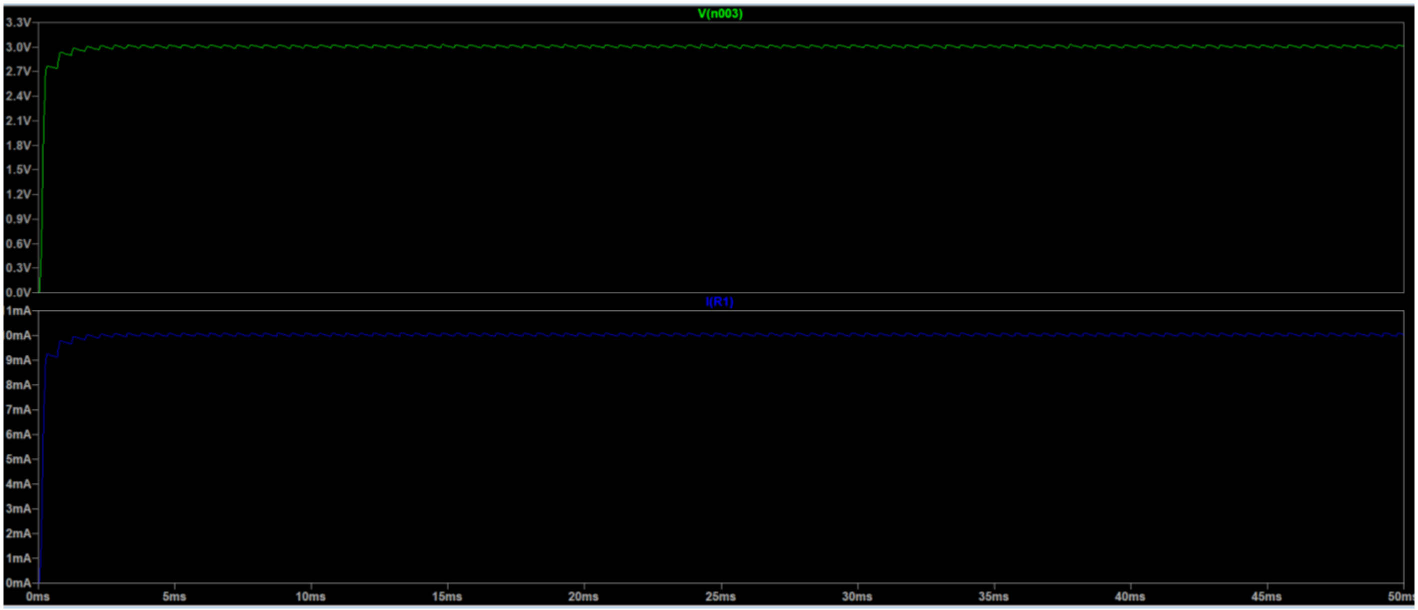

Figure 7: LTspice simulation results

The output voltage values range from 2.98V - 3.03V while the load current values range from 10.1mA - 9.97mA. This fits exactly within the requirements of the circuit specifications.

Discussion:

Comparisons:

This project was all about designing a DC power supply that converts a 120V RMS AC input into a 3V ± 0.1V DC output with a 10mA load. The design was broken down into multiple stages, starting with research, creating schematic diagrams, LTspice simulation testing, and prototyping a physical circuit for implementation. Breaking down the entire project into little chunks. It made every step much more manageable and ensured a smooth transition from theory to practice. This procedure took into account for all the various calculations involved, adjustments for all the numbers and circuit design choices, and valued operating safely. Researching how to develop a circuit from ground-up built the foundation, allowing for the understanding to create schematics and guide circuits development. The LTspice simulations helped to eliminate any potential errors down the line by identifying problems early, allowing a solution to be found and tested before building the physical circuit. The physical implementation accounted for real-world factors such as component tolerances and slight inaccuracies. Overall, the observed ~83% efficiency between the simulated and physical results demonstrate the effects.

Limitations:

One of the most challenging aspects of this project was determining the appropriate input voltage to achieve the required output voltage. The measured physical values had slight differences than the calculated ones and required adjustments to meet the specifications. While the issue was manageable for this specific application, it highlights the importance of precision when designing circuits, as even small errors can have serious consequences in real-world applications as safety considerations must always be prioritized. A key design limitation was the absence of a regulator, mostly due to the Zener diode's tolerance of 3V ± 0.15V, this exceeded the required power supply tolerance of 3V ± 0.1V and because the circuit already met the required specifications, adding a regulator for additional voltage stability was unnecessary. Instead of introducing a regulator, it was easier to adjust the transformer values. This would have been a more practical and cost-effective solution, reducing complexity while also meeting the requirements. Another limitation was not including a transformer in the physical design. This left the AD3 function generator to simulate that role. The trade-off about this was that it introduced uncertainties that can impact the rectification process, altering overall circuit performance and accuracy. Despite these factors, the procedure outlined earlier ensured that the entire design process continued smoothly, even with multiple iterations and trial-and-error, that was expected. LTspice simulations played a crucial role in validating design choices, analyzing performance, and understanding the interaction between components, ensuring a reliable final prototype.

References:

[1] "Half-wave rectifier," Analog Devices, https://www.analog.com/en/resources/glossary/half-wave-rectifier.html (accessed Feb. 5, 2025).

[2] A. S. Sedra, K. C. Smith, T. C. Carusone, and V. Gaudet, Microelectronic Circuits, 8th ed. New York, NY, USA: Oxford Univ. Press, 2019.

[3] Diotec Semiconductor AG, 1N4148, 1N4150, 1N4151, 1N4448 Small Signal Switching Diodes Data Sheet, Aug. 2017. [Online]. Available: http://www.diotec.com/.

[4] "Project 1", Avenue to Learn, February 5, 2025

[5] "Project 1 guidelines", Avenue to Learn, February 5, 2025