Electronic Devices and Circuits I 2EI4
Design Project #1 - DC Power Supply

Rahim Aziz - C01

Instructor: Dr. Yaser M. Haddara

Department of Electrical and Computer Engineering, McMaster University

Due Date: February 12th 2023

Summary

A DC power supply that converts an AC source to a DC power source is composed of a transformer, rectifier, a filter, and a regulator. (See figure 1 below) [1]. The purpose of this project was to determine the specifications for the aforementioned parts, given a list of input and output specifications. The DC power supply was expected to receive 120V (rms) at 1kHz and to deliver 10mA at 3V ± 0.1V to its load.

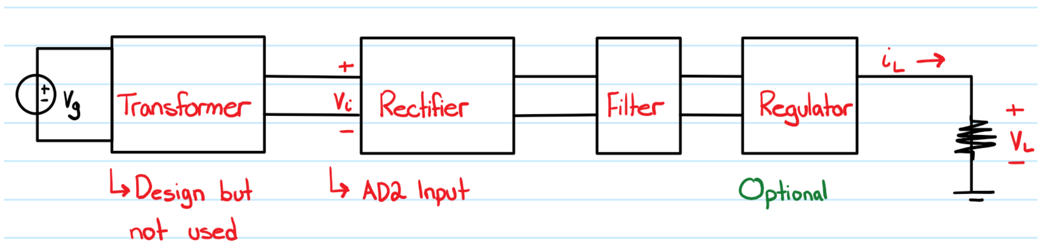


Figure 1: Block Diagram

Design

The design process entailed an iterative process involving two subsequent designs. The first was a center tapped full wave rectifier design with a regulator. The final design ended up being a center tapped full wave rectifier without a regulator. The reason for this choice will be expanded upon later in this report.

Figure 2 below shows a preliminary schematic design for the final design that was hand drawn prior to completing any simulations. This hand drawn diagram was purely for manual calculation purposes.

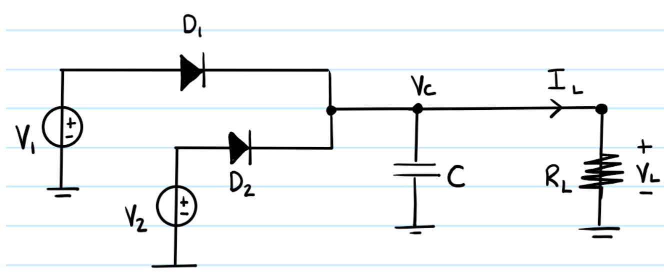


Figure 2: Hand Drawn Schematic

The first component that was determined was the load resistor using the calculations below (Figure 3). Using ohm's law, the value of the resistance load was determined to be 300 ohms. For the physical circuit, a 330 ohm Abra resistor was used as it is the closest value resistor in the provided course kits.

$$R_L = \frac{V_L}{I_L} = \frac{3V}{10mA} = \frac{3V}{0.01A} = 300\Omega$$

Figure 3: Resistor Calculations

The second component that was determined was the filter type. The filter type of choice was the parallel RC circuit. The alternate filter designs were an LC filter, and other variations involving inductors. While the RC filter dissipates more energy in the form of heat, the components involved in an RC filter were more readily available [2]. The calculations in (Figure 4) were to determine the minimum capacitance value given the current constraint on the load, as well as the desired ripple voltage (peak to peak). For the filter circuit, a 100 microfarad Abra capacitor (CM107) was used, as it is available in the provided courses kits.

For a capacitor filter in parallel with R_L we know…

- $V_C = V_L$
- For a small ripple voltage at V_L, $CR >> T$
- The ripple voltage can be expressed as $V_r = \frac{I_L}{fC}$
- For a center tapped full wave rectifier $f_{output} = 2f_{input}$

We would like to keep our ripple voltage below 0.1V so we will aim for a ripple of $V_r = 0.05$ (Peak-to-peak)

$$V_r \geq \frac{I_L}{f_{output} \cdot C}$$

$$0.05V \geq \frac{10mA}{(2kHz) \cdot C}$$

$$C \geq 100uF$$

Figure 4: Capacitance Calculations

The third component that was determined was the rectifier. For this design, the full wave rectifier topology was chosen. Firstly, after further investigation into the various topologies, it was determined that the full wave rectifier was the optimal choice. The full wave rectifier was selected over the half-wave rectifier as it has smaller ripple factor (0.482), higher efficiency (81.2%), and overall a higher transform utilization factor [3]. Additionally, there are two more categories of full wave rectifiers. The center tapped the full wave rectifier and the bridge rectifier. The center tapped full wave rectifier was selected over the bridge rectifier as it is more cost-effective and requires fewer diodes (2 instead of 4). The calculations in (Figure 5) demonstrate what the voltage inputs to the rectifier should be, given the peak capacitor voltage and the target ripple voltage. For the physical circuit being built, we are using the Analog Discovery 2 (USB Oscilloscope) as inputs to the rectifier. For the physical rectifier circuit the 1N4148 Abra diodes are being used as it is also available in the provided course kits.

We know that…

- $V_{pc} = Peak\ Capacitor\ Voltage = 3.05V$
- $V_C = V_{pc} - \frac{1}{2}V_r$
- By KVL, $V_1 = V_{D_1} + V_C$
- The corresponding data sheet indicates that this diode has a forward voltage of 0.72V at 20mA. The data sheet also indicates a voltage breakdown of 100V.

$V_C = V_{pc} - \frac{1}{2}V_r = 3.05V - \frac{1}{2}(0.05V) = 3.025V$

$V_1 = V_{D_1} + V_C = 0.72V + 3.025V = 3.745V$

For the full rectifier circuit input

$V_1 = V_m sin(wt) = 3.745 sin(wt)$

$V_2 = - V_m sin(wt) = - 3.745 sin(wt)$ → Is required to be 180 degrees out of phase

Figure 5: AC voltage input from Oscilloscope

The final component that was determined was the transformer design. This project did not require a physical implementation of the transformer, only a turns ratio calculation. The center tapped transformer in conjunction with the full wave rectifier was chosen because it makes use of both positive and negative half cycles of the AC input signal. This allows for a more efficient voltage transformation. The calculations for the turns ratio is given below in (Figure 6)

$$V_0 = 2V_1$$
$$V_0 = 2(3.745sin(wt)) = 7.49sin(wt)$$

Convert V_0 to an RMS value

$$V_{0RMS} = \frac{V_0}{\sqrt{2}} = \frac{7.49}{\sqrt{2}} = 5.30V\ (RMS)$$

Turns Ratio

$$a = \frac{N_1}{N_2} = \frac{V_S(RMS)}{V_0(RMS)} = \frac{120V}{5.3V} \approx 22{:}1 \text{ Turns Ratio}$$

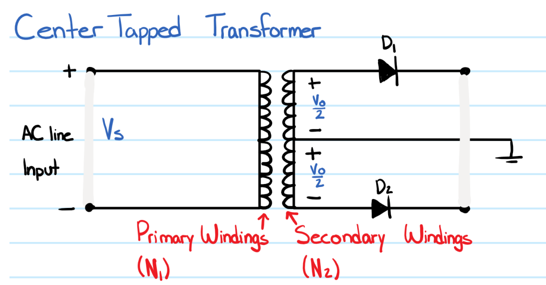


Figure 6: Transformer Calculations

First Iteration Circuit

In the first iteration of the DC power supply design, a regulator in parallel with a 30 Ω resistor was added to reduce ripple voltage and create a smoother DC output signal. The first iteration circuit calculations followed a slightly different process to determine the ideal component values. The regulator was composed of the 1N5225B Abra zener diode in parallel with a 30Ω resistor. The corresponding data sheet indicated a forward voltage of 3.0V at an operating current of 20-50mA. Despite various successful simulations and calculations, the physical circuit failed to meet specifications. A smooth DC signal would oscillate between 2.4 and 2.5V. After further investigation, it was determined that maintaining the operating zener diode current in conjunction with maintaining the voltage across the capacitor created current spikes across the diodes. These current spikes exceeded the rated output current that the AD2 wave generator could output. The AD2's wave generator current output is rated for 10-50mA [4]. In the simulation graph for the first iteration circuit below (Figure 7) you can see the current coming out of the initial diodes exceeds 200mA.

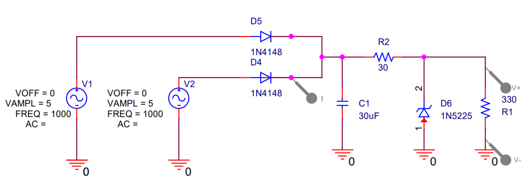
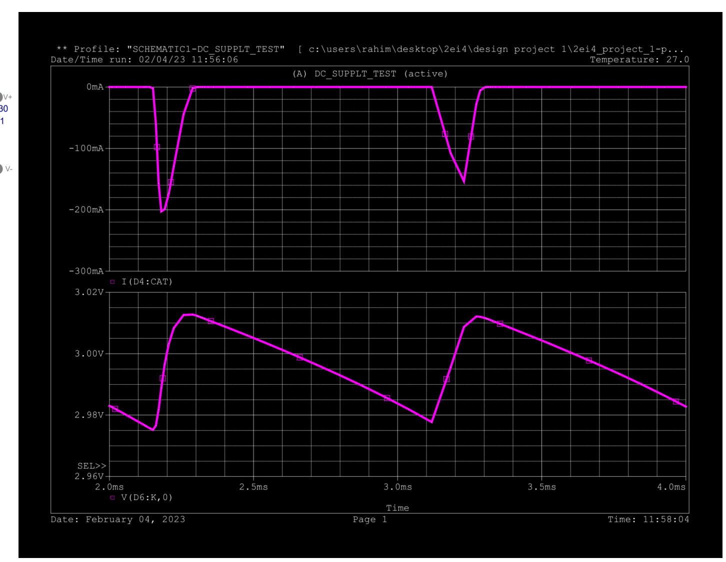


Figure 7: First Iteration Circuit



Another design flaw with the first iteration is related to the component rating of the zener diode. According to the datasheet of the zener diode, it has a voltage tolerance of $\pm$ 5%. This means if there is a 3V drop across the diode we can see it fluctuate anywhere between 2.85V and 3.15V which exceeds our design specifications.

Nevertheless, after proceeding with the final iteration design, we expect it to perform within the voltage range of 2.95V - 3.05V. Additionally, the current is expected to be below 10mA due to the fact a slightly larger resistor is being used. Furthermore, after all the calculations, it is expected none of the currents or voltages will exceed any of the component ratings that were found in the corresponding datasheets. Finally, throughout this process, there were a couple design tradeoffs that were made. Firstly, the filter choice (RC) is considered by far the least efficient filter that results in a less smooth DC output. Secondly, the alternate full wave bridge rectifier has a higher transformer utilization factor of 81.2% while the center tapped full wave rectifier only has a transformer utilization factor for 57.32%.

Below are a list of the data sheets that were consulted

- 1N5225B Zener Diode → LINK
- 1N4148 Diode → LINK

Simulation

Prior to physically building the circuit, a simulation in OrCAD PSPice was created to ensure that the currents and voltages did not exceed component ratings. The schematic for the design is shown below in (Figure 7). The net list is given in (Figure 8) and the simulated output waveform is given in (Figure 9). The diode components used were taken from the PSPice library such that they had the exact same parameters as the ones provided in the course kits. The simulation analysis type was in the Time Domain (Transient) with a runtime of 2000us to 4000us and a maximum step size of 10s. From the simulated output graph you can see the output voltage oscillates between 2.98V and 3.04V, which is within specification. The current oscillates between 9.05 mA and 9.15mA. This is because in the schematic, the circuit uses a larger 330 ohm resistor that is available in the provided course kit, which is expected to reduce the current.

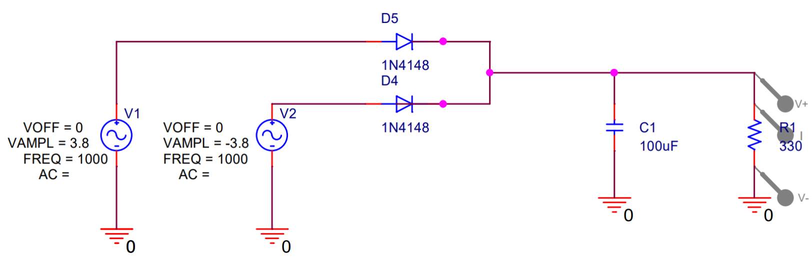


Figure 8: OrCAD PSPice schematic design

```
* source 2EI4_PROJECT_1
V_V1          N00232 0
+SIN 0 3.8 1000 0 0 0
V_V2          N00336 0
+SIN 0 -3.8 1000 0 0 0
X_D4          N00336 N00340 awb1n4148 PARAMS:
X_D5          N00232 N00340 awb1n4148 PARAMS:
C_C1          0 N00340  100uF  TC=0,0
R_R1          0 N00340  330 TC=0,0
```

Figure 9: Netlist

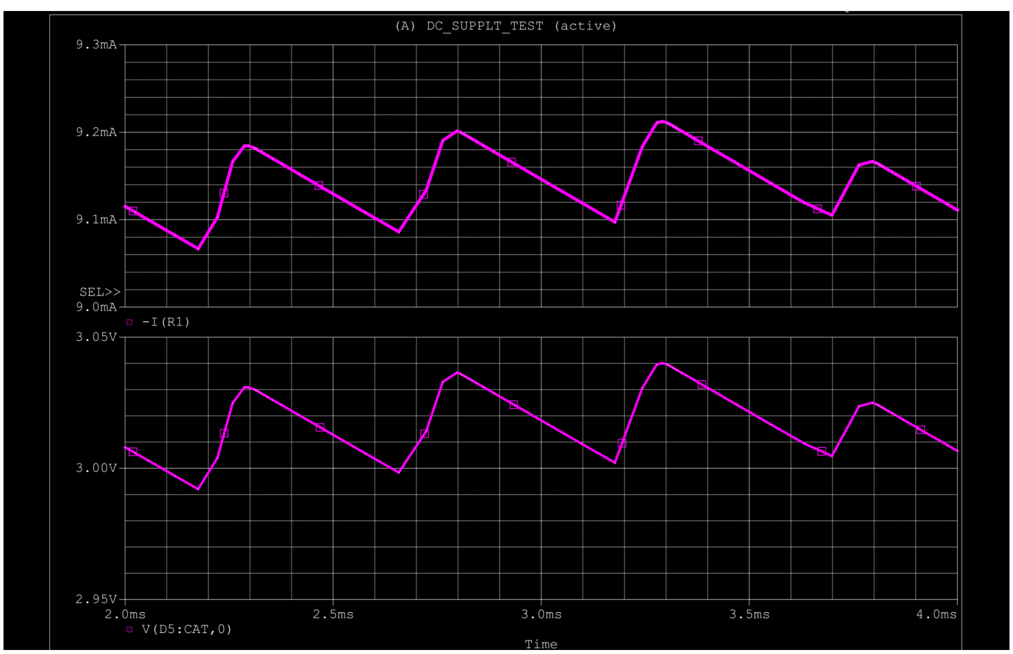


Figure 10: Simulation Waveform

Measurement and analysis

To test the accuracy of our calculations and simulations, a circuit was built (As seen in Figure 11). To test the circuit two wave generators from the AD2 were put at the input diodes of the circuit and two voltage probes were placed across the load resistor. The wave generators used two sine waves with a voltage of 3.8V and with one with a phase offset of 180 degrees. There was no vertical offset and the frequency was 1kHz. With this input voltage the measured voltage across the load was slightly below specification, oscillating between 2.7V-2.8V. The input voltage was then raised to 4.2V to obtain an output voltage within specification. The resulting voltage across the load resistor was plotted on the graph in (Figure 13). From the graph we can see that the voltage oscillates between 2.94V and 3.095V which is within the design project specification. The current can be found by dividing this voltage range by the resistance of the load, which produces a current range of, 8.9mA to 9.3mA. This variation is expected as a 330 ohm resistor was used as opposed to the calculated 300 ohm resistor

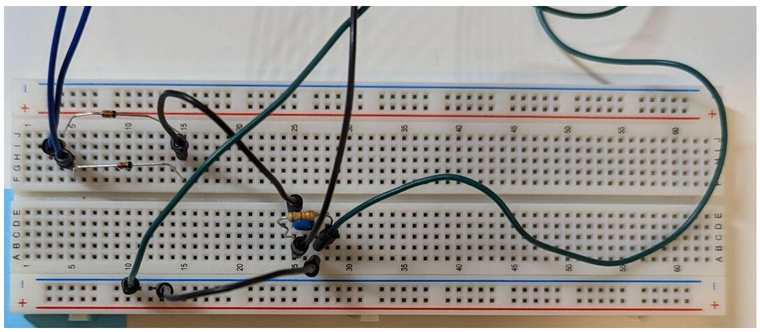


Figure 11: Physical Circuit

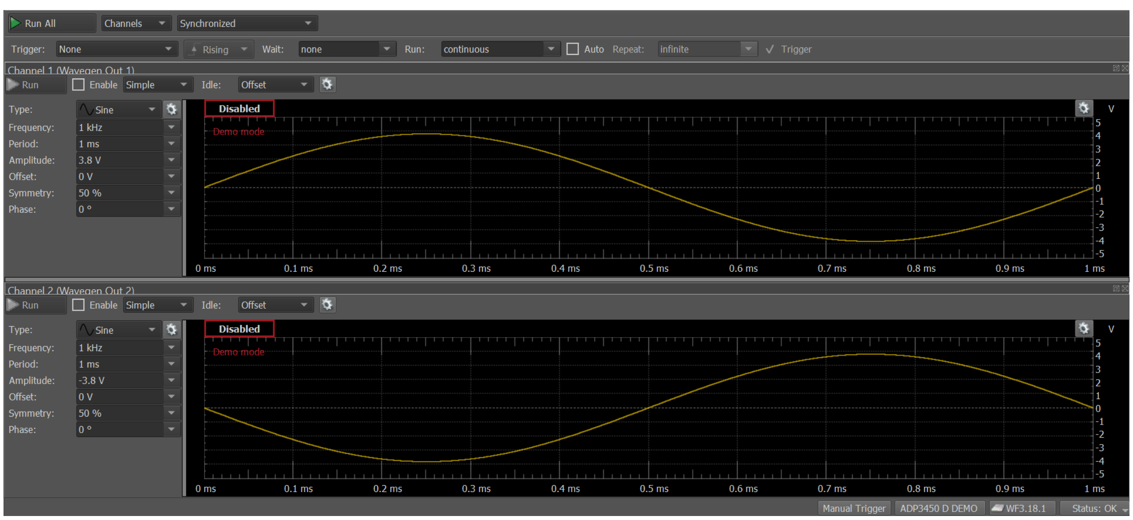


Figure 12: Function generator sine wave input

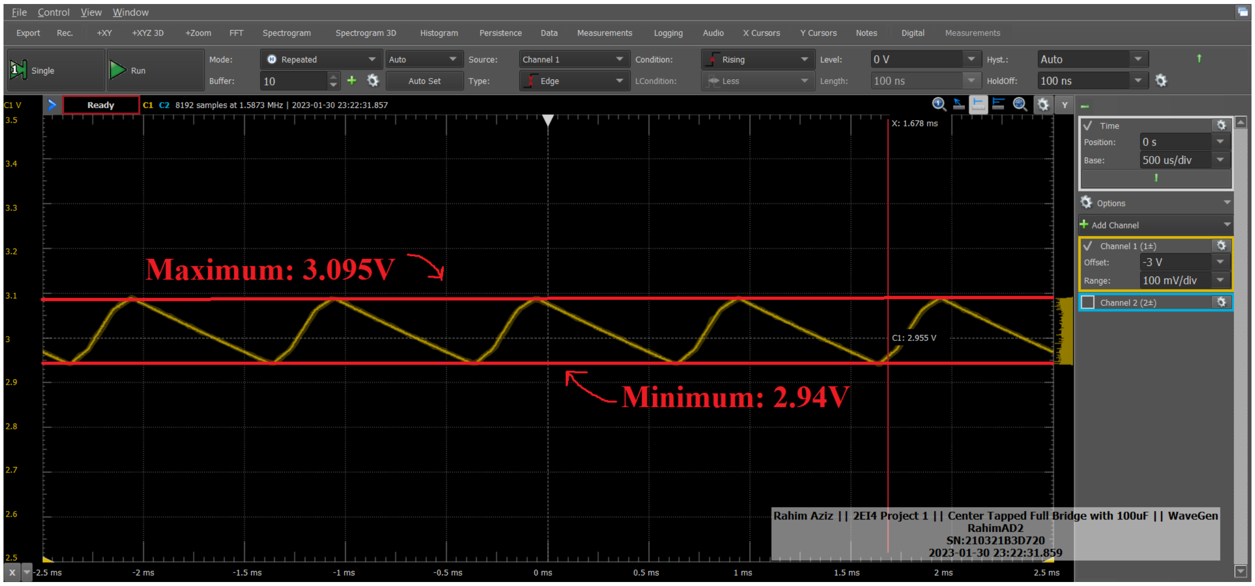


Figure 13: AD2 Voltage Load Output

Discussion

After further investigation of the design calculations, simulations, and measurements, we can note varying discrepancies, limitations, and issues that were encountered during the process. Firstly, the design simulation matches very closely to the calculations with the exception of the change in resistance. The simulation demonstrated a ripple voltage between 2.995V and 3.04V. Additionally, the simulated circuit showed that the expected current through the load resistor would be approximately 9.05mA to 9.15mA. However, when moving over to the physically designed circuits and its measurements, we can see a discrepancy in the output voltage when using the calculated/simulated input voltage (3.8V). The result was an output not within specification. As a result, the voltage input amplitude had to be increased to about 4.2V. Additionally, the current was calculated to be between 8.9mA to 9.3mA. These discrepancies can be attributed to potential voltage and current tolerances of the components that may have affected the final output graph. All the components used in the design have some very small tolerance, but it can add up quickly. For example, in the first iteration as previously mentioned, the voltage tolerance across the zener diode ($\pm$ 5%) used in the regulator, caused the output voltage to fluctuate outside the given specifications. While designing and implementing the circuit, the limitations present were especially evident in the first iteration of the circuit. As explained earlier, with the regulator in place, the current spikes exceeded the maximum current output of the AD2. Therefore, the resulting load voltage was not within specification. Another limitation was the lack of inductor components. LC circuits are far more efficient and with the correct inductor values a more effective and efficient filter could have been built. Finally, throughout the measurement process, consistent results below specification were obtained. To troubleshoot this, measurements of each component (Measured the resistors and diodes using a voltmeter) were taken to ensure they weren't damaged, various sections of the circuit were probed to ensure the flow of current, and lastly wires were swapped out to ensure no faulty wires were used. When the circuit was initially setup with a sine wave input of 4.2V the output was not with specification. This was due to faulty wires being used, and therefore swapping them out resolved the measurement problems.

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] Pani, “Power supply filters,” Electronics Tutorials, Jun. 18, 2018. https://electronicspani.com/power-supply-filters/ (accessed Jan. 31, 2023).

[3] R. Visintini, ‘Rectifiers’, 08 2006, pp. 133–183.

[4] “Analog Discovery 2 - Digilent Reference,” digilent.com. https://digilent.com/reference/test-and-measurement/analog-discovery-2/start?redirect=1 (accessed Jan. 31, 2023).