Project 1 - Report

Edison He (hee6)

ELECENG 2EI4 - Electronic Devices and Circuits 1

February 9th, 2025

Summary

This project revolves around design and creating a DC power supply which generates a final output of 10 mA at 3V ± 0.1V from an input of 120V at 1kHz AC. It includes components such as a transformer mimicked by the AD3, a rectifier which removes the negative cycles of the sinewave, a filter which smooths out the input voltage, and a regulator which maintains the target voltage at the desired output.

Design

Transformer

A transformer is used to scale the incoming voltage to the desired voltage of the device. In this project, the transformer will not be used, but rather simulated by the AD3. However, the input voltage from the voltage is still essential to the power supply circuit, and therefore still needs to be calculated. The turn ratio of the transformer is calculated at around 32 with an input sine voltage of 3.8V given a 120V source. If a transformer was used in the circuit, a center-tapped transformer would have been selected.

Rectifier

A rectifier is used to block the negative flow of current. This is done through the use of a center tapped full wave rectifier which converts the negative portion of the wave to a positive wave. The center-tapped rectifier was chosen over the bridge rectifier because it required 2 diodes rather than 4 diodes in a bridge rectifier. Because of manufacturing tolerances, using less components may yield a greater measurement accuracy. The rectifier makes use of 2 1N4148 Diodes. A forward voltage drop of 0.7V will be used in calculations.

Filter

A filter is used in this circuit to smooth out the incoming AC input to an AC ripple. The filter in this circuit is a 100uF capacitor in parallel to a 300Ω resistor. This can be done with 3x100Ω resistors. However, for simplicity, a single 330Ω resistor was used.

Regulator

Similar to a filter, a regulator is used to further smooth out the AC ripple to a consistent DC voltage. It is seen in most AC to DC power supplies, but is not required. The circuit designed in this project does not make use of a regular as without it, the target DC voltage range was obtained.

Circuit Schematic

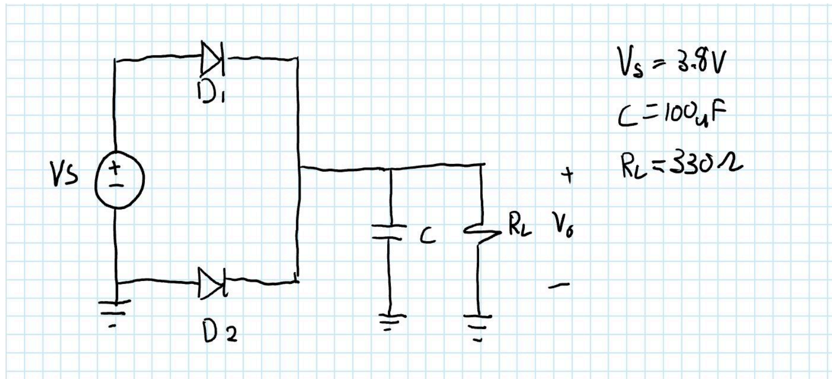


Figure 1: Circuit Diagram

Calculations

Required Input Voltage

$V_s - V_{o_1} - V_r = V_0$

$V_s - 0.7 - 0.1 = 3$

$V_s = 3.8V$

Turn Ratio

$\frac{N_1}{N_2} = \frac{V_s}{V_0} = \frac{120}{3.8} = 31.58 \approx 32$

Required Capacitance

$V_r = \frac{V_p}{fCR}$

$0.1 = \frac{3}{(1000)(330)C}$

$0.1 = \frac{3}{(1000)(330)C}$

$C = 9.09 \times 10^{-5}\mathrm{F}$

$C \approx 10\times 10^{-4}F$

$C \approx 100uF$

Expected Performance

An input of 3.7V at a 1kHz frequency is used for this project. This will be done by generating two sine waves, one of which is an inverse of the other. The rectifier has two diodes in parallel which remove the negative half of the wave and transform them into 2 positive amplitudes every cycle, or a full wave rectifier. As only one rectifier turns on each time and considering the constant drop model, only 0.7V needs to be taken into account rather than 1.4V in addition to the 3V source. Finally, the filter, or the capacitor reduces the incoming wave further to achieve the ± 0.1 voltage range.

Trade Offs

A tradeoff that I have made in the circuit is the use of 330Ω resistor rather than 100Ω resistors in series. As a 300Ω resistor was not given in the kit, I had the choice of using a single 330Ω resistor or 3x100Ω resistors in series. For simplicity, I decided to use the single 330Ω resistor. In addition, I believe this would have led to a more accurate final reading at the cost of a current drop. This is because of manufacturing tolerances, as there is a higher inconsistency with using more components. I decided that using the least amount of comments would net me the most accurate results. This is also the reason I used the center tapped full wave rectifier rather than a bridge rectifier as it uses less diodes.

Another issue that needs to be considered is the ratings of the components used as I am building the circuit in real life. Each component in the circuit (ex. capacitors, resistors, diodes) have maximum voltage and amperage ratings that must not be exceeded. If they are exceeded, the component may lose functionality or even explode. This is not only harmful to other components in the circuit but also the user. The calculations for each component can be seen above. They do not exceed the maximum ratings in this project.

Measurement and Analysis

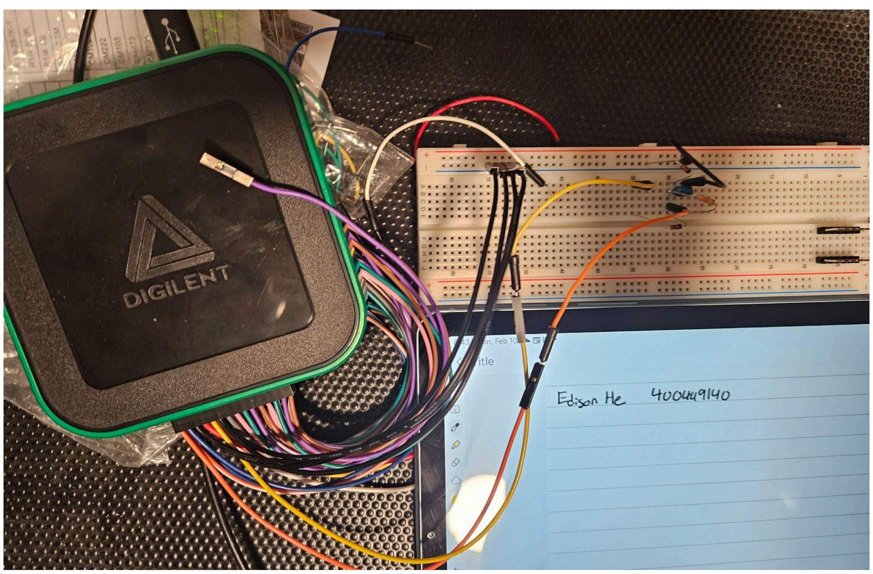


Figure 2: Completed Circuit

As seen in figure 4 below, the oscilloscope function of the AD3 was used to reveal the output of the built DC supply. To measure the performance of the supply, channel 1 was placed on the load resistor and the current was measured through the math function. The formula for the current channel is a simple Ohm's law equation of V=IR rearranged for I. In this case, the equation was I = C1/330Ω, where C1 was the measured voltage. The resulting average voltage and currents were calculated by the sum of the peaks divided by 2. This resulted in a voltage of around 2.76V and a current of 8.36mA.

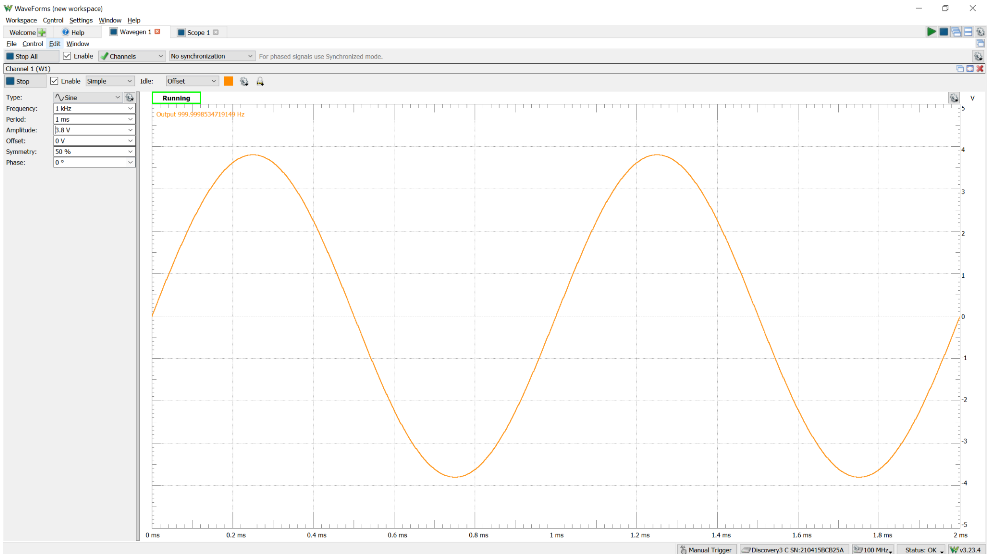


Figure 3: Wavegen Output

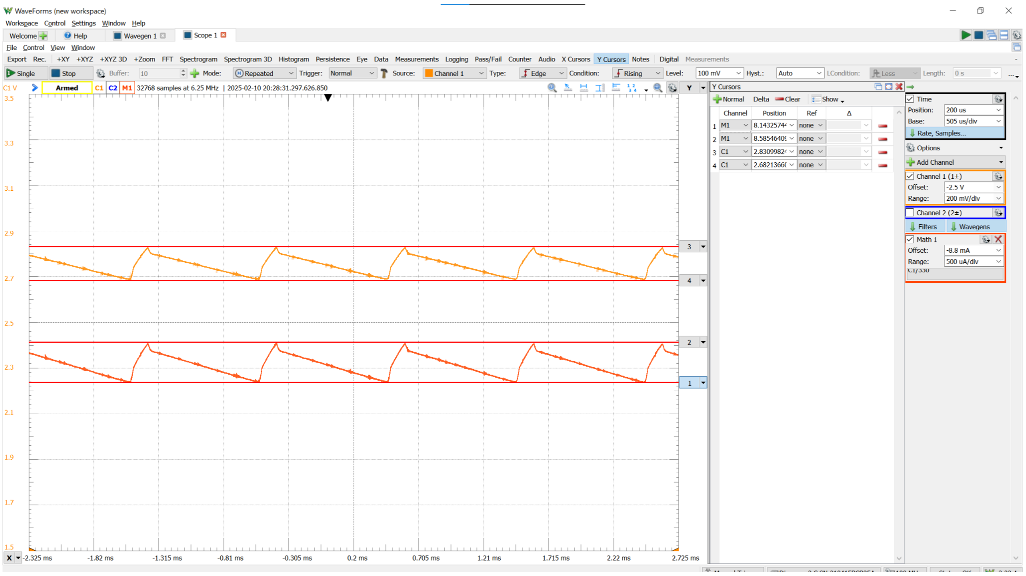


Figure 4:Final Oscilloscope Output

Results

Once again, the output voltage and current of the supply was around 2.76V and 8.36mA respectively. Although these parameters are close to the target specifications, increasing the amplitude of the input voltage to 3.9V netted closer results at 2.83V and 8.65mA, which can be seen in figure 5. This reveals that the supply that was built was not ideal and has other factors to consider. Factors include the tolerances of the components, the unknown real diode forward drop voltage, internal resistance in the components, and the load resistor. The load resistor does not affect the voltage, but rather the current. As the resistor value is 330Ω rather than 300Ω, the current value will be lower.

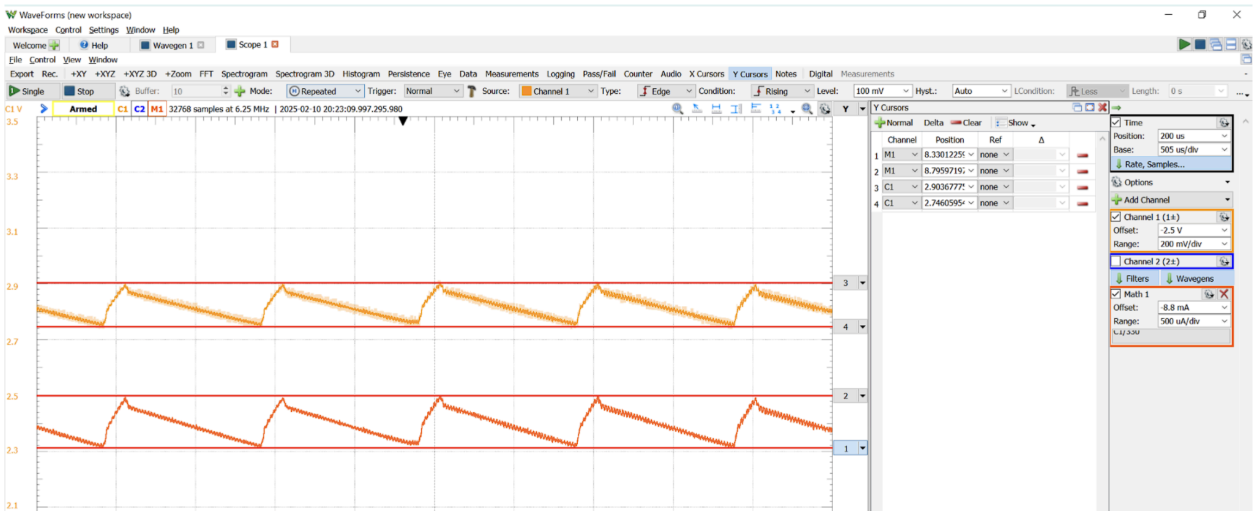


Figure 5: Voltage Amplitude at 4V

Simulation

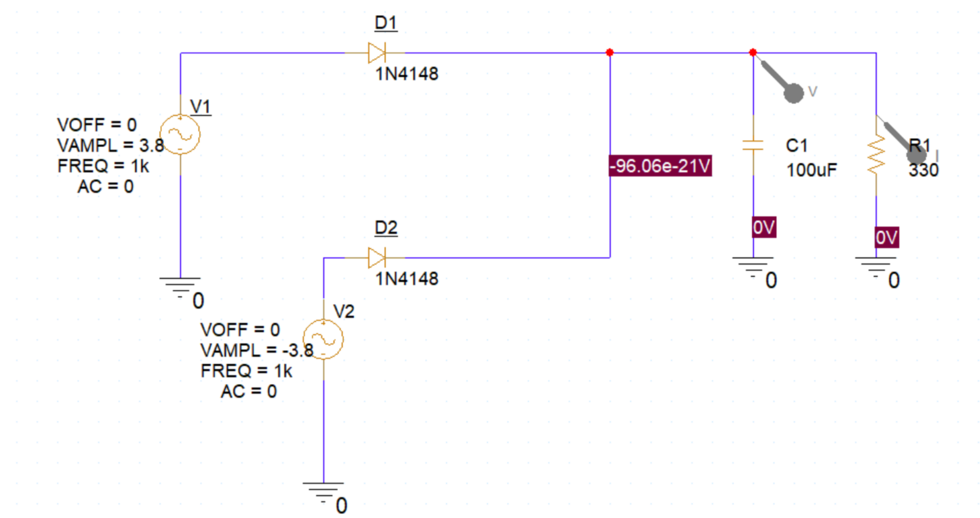
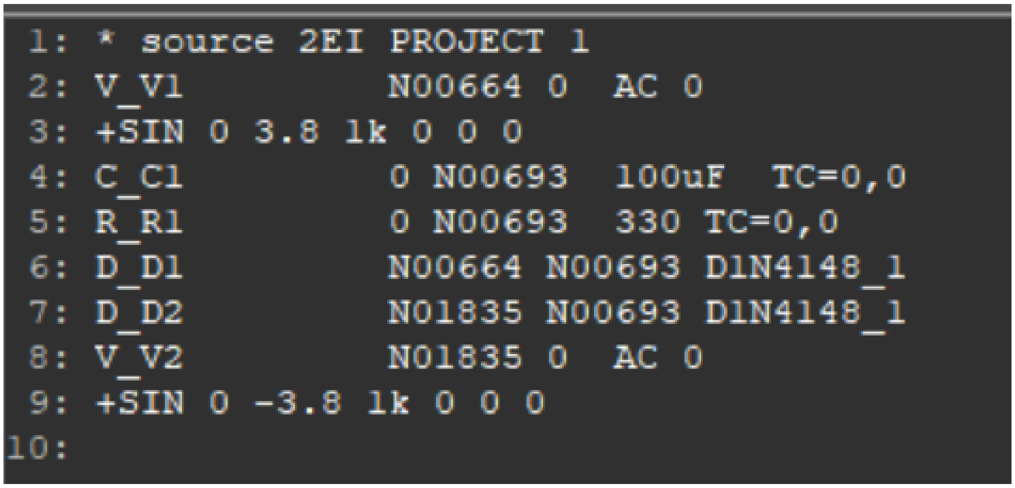


Figure 6: PSpice Circuit Schematic



```
* source 2EI PROJECT 1
V_V1         N00664 0  AC 0
+SIN 0 3.8 1k 0 0 0
C_C1          0 N00693  100uF  TC=0,0
R_R1          0 N00693  330 TC=0,0
D_D1         N00664 N00693 D1N4148_1
D_D2         N01835 N00693 D1N4148_1
V_V2         N01835 0  AC 0
+SIN 0 -3.8 1k 0 0 0
```

Figure 7: Netlit

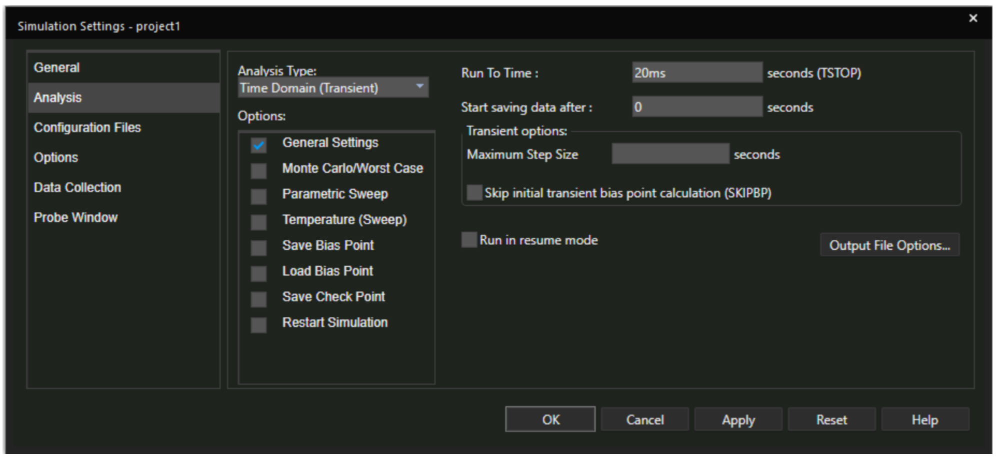


Figure 8: Simulation Conditions

Figure 9: Simulation Output

In figure 9, the results of the simulation can be seen. When looking at the voltage and current values, they line up with the target output parameters. The average voltage and current parameters are at around 2.93V and 8.87 mA respectively. Because I used a 330Ω resistor rather than a 300Ω, it is expected that the current value is lower than the target value.

Discussion

Comparison

Comparing my measured values with my simulation and theoretical values, there were minor differences throughout the project. Each step of the process yielded slightly lower values. For example, as mentioned above, the average voltage and current parameters are at around 2.76V and 8.36mA in the built circuit and 2.93V and 8.87 mA in the simulation. However, I was able to obtain slightly better values when bumping up the voltage by 0.1V each step in the built circuit. This netted values of 2.83V and 8.65mA at 3.9V. This means the real life input voltage needed is slightly higher than what is needed in the theoretical and simulation values. This is because of factors that are not accounted for in reality.

Design Limitations

The most major factors that affected my built circuit were the load resistor value in addition to non-ideal components. I used a load resistor value of 330Ω instead of 300Ω. By Ohm's Law, at a constant voltage value, resistance and current are inversely proportional, meaning increasing the resistance decreases the voltage. In addition, tolerances and internal resistances cause voltage drops. The resistor used is not exactly 330Ω due to the manufacturing process and the actual value can be greater or lower. The diode's true forward voltage was also not considered in the calculator and could have been greater or lower than 0.7V from the constant drop model used in my calculations. In addition, the internal resistances of the wires and diodes will take away from the source voltage. All these reasons result in the increase in input voltage which makes the final result more accurate.

Problems Encountered

The biggest problem that I have encountered are lower voltage and current values than the target due to the design limitations. To solve this problem, I realized that incrementing the input voltage amplitude by 0.1V until the target voltage is reached solved this issue. After experimenting with voltages, an input voltage amplitude of around 4.3V in addition to a 300Ω resistor fixed this issue.

Another issue that I had was that my input signal was incredibly noisy. From past experiences, this has many causes including using the incorrect grounds or faulty connections. After checking my circuit I realized that one of my grounds was inserted halfway which did result in a cleaner signal. However, the signal is still slightly messy and I was unable to find a way to have a 100% clean signal. This may be due to factors that I am unable to control.