ELECENG 2EI4

Design Project 1

Instructor: Dr.Haddara

Suchir Ladda - laddas - 400517569

Date: 2/9/2025

As a future member of the engineering profession, 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 and the Code of Conduct of the Professional Engineers of Ontario. Submitted by [**Suchir Ladda, laddas, 400517569**]

Summary

In this project, the objective is to build a DC power supply that delivers 10mA at 3V ± 0.1V from a source that is 120V (rms) at 1kHz. The normal DC power supply that converts AC to DC consists of a transformer, rectifier, a filter, and a regulator. For safety reasons, this project will not use an actual transformer, instead the AD3 will be used to simulate the output of a transformer. For this project a function generator will be used to simulate transformer output, a center-tapped full wave rectifier will convert AC to DC, and a filter will be used to decrease the ripple voltage leading to a smooth waveform.

Design

i) Transformer

As mentioned earlier, the project does not incorporate a transformer into this design. Despite the absence of the transformer, its turns ratio must be calculated as if it was connected to a 120V (rms) source. In order to ensure the output voltage falls between the range of 2.9V and 3.1V, the following calculations were done:

$$Turn\ Ratio\ = \frac{120V\ (rms)}{3.8V} = \frac{120\sqrt{2}}{3.8V} = \frac{169.705}{3.8} \simeq 45\ turns$$

The 120V (rms) source was multiplied by $\sqrt{2}$ to get an approximate value of the amplitude (169.705). Furthermore, since the expected power supply output had an upper bound of 3.1V, the forward bias of the diode (0.7V) was added onto it to reduce error. To conclude, if an actual transformer were to be used, it would have a turns ratio of approximately 45.

ii) Rectifier

In this project, a full-wave center-tapped rectifier was chosen as the topology. Compared to the other designs, the center-tapped design is by far the most simple. Furthermore, a full wave design is better as it gives an accurate output as both halves of the AC wave are included. When compared to the bridge rectifier that uses four diodes, the center-tapped rectifier uses only two diodes which reduces the chances of error greatly skewing the results. Since the terminals of the transformer are attached to each diode separately the input AC voltage goes through only one diode in forward bias. For the physical implementation of the circuit, 1N4148 diodes are being used and the AD3 is being used as the input for the rectifier.

iii) Filter

For this project, a filter was used to filter out the waves and get a steady DC voltage. This was achieved by using an RC filter in which a resistor and capacitor are placed in parallel. Using the Vrpp formula and constraint values given the capacitance was calculated as follows:

$$Vrpp = \frac{I_L}{FC} \text{, Therefore } C = \frac{I_L}{\Delta VF} = \frac{10\times10^{-3}}{0.1(1000)} = 100\mu F$$

Vrpp/ΔV is the ripple voltage, I is the deserted output current, and F is the wave frequency. Using these values it is determined that a 100µF capacitor is to be used in the filter.

In addition to the capacitor, a resistor is needed in order to filter out the noise.

iv) Regulator

A regulator is used to lessen and smooth out the voltage ripples. In the simulation discussed below, it can be seen that the output voltage is already in the desired range thus implementing a regulator would be a waste of resources.

v) Circuit Schematic

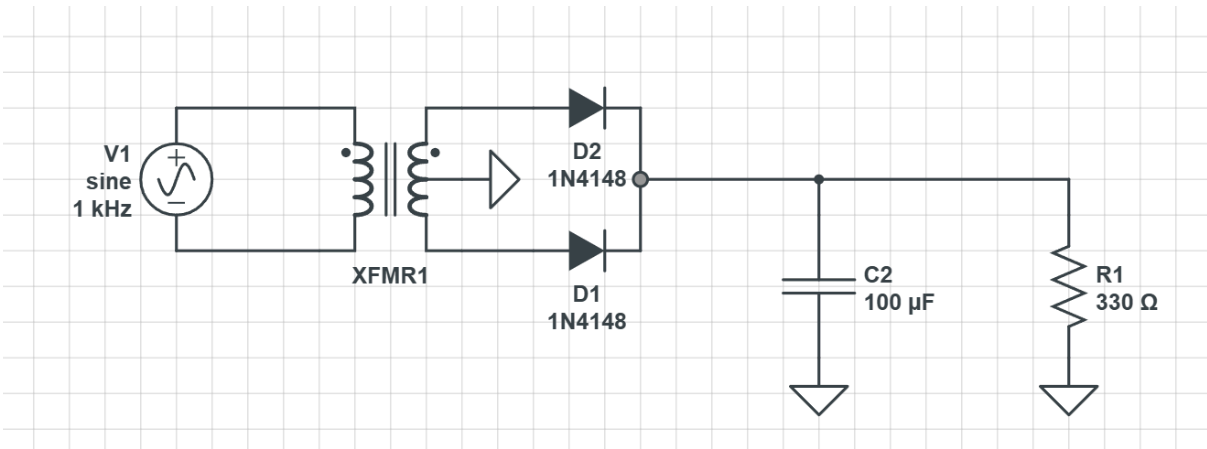


Figure 1: Circuit Schematic

vi) Calculations

AC input from Oscilloscope:

$$V_{DC} + 3V = \frac{2}{\pi}V_{MAX} + 3V = 0.900V_{RMS} + 3V = 3.9V$$

However due to possible experimental error, this value was adjusted to 3.85V in the LT Spice simulation.

i) Transformer

$$Turn\ Ratio\ =\ \frac{120V\ (rms)}{3.8V}\ = \frac{120\sqrt{2}}{3.8V}\ =\ \frac{169.705}{3.8}\ \simeq 45\ turns$$

The above calculation is for the ratio of turns which is equal to the ratio of primary and secondary voltage.

ii) Filter

$$Vrpp\ = \frac{I_L}{FC}\ ,\ \text{Therefore } C\ = \frac{I_L}{\Delta VF}\ = \frac{10\times10^{-3}}{0.1(1000)}\ =\ 100\mu F$$

The calculations indicate the using a $100\mu F$ capacitor is the best choice as it will result in the smoothest waveform.

Load Resistance calculation:

$$V\ =\ IR_L,\quad R_L\ =\ \frac{3V}{10mA}\ ,\ R_L\ =\ 300\ ohms$$

The calculations above indicate that using a 300 ohm resistor is the best choice. However, after speaking to Dr.Haddara, he suggested using a singular 330 ohm resistor in the physical circuit implementation rather than two, 150 ohm resistors in series.

vii) Expected Performance

After running the LTSpice simulation and gathering data, I believe that my design will produce the desired outcome of a DC voltage that is 3V ± 0.1V with a current of ~10mA.

viii) Design Tradeoffs and Margins

There were a few tradeoffs that were made in the construction of the DC power supply. The biggest one was choosing to use a 330 ohm resistor instead of combining two 150 ohm resistors. This was done because of simplicity and possible component interference which could lead to skewed values.

Another tradeoff that was made was choosing the center-tapped rectifier instead of the bridge rectifier. This decision was made because the center-tapped design only uses two diodes instead of four (like the bridge does). Though both designs had full-wave rectification, the center-tapped rectifier was the clear choice as it would minimise the amount of voltage drops.

Furthermore, in order to achieve a smooth waveform I opted to use a single $100\mu F$ capacitor. This worked out very well as I had an extremely small ripple voltage. I also chose to use this capacitor as I didn't need to combine any more in series or parallel to achieve the desired capacitance.

Measurement and Analysis

i) Circuit Photograph

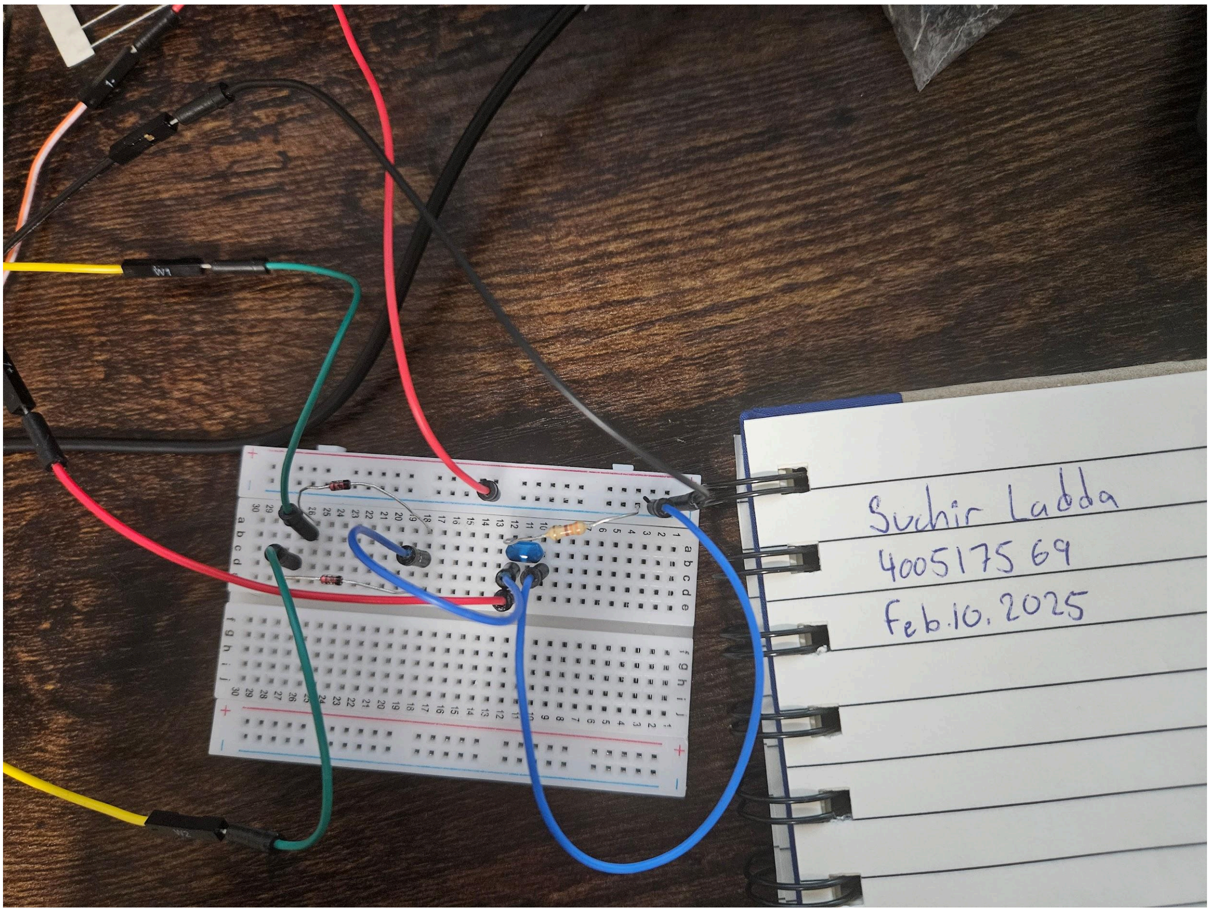


Figure 2: Physical Circuit Implementation

ii) Measurement Procedure

In order to determine that the desired output voltage and current is being reached, I had to take measurements using the AD3s built in oscilloscope. I measured the voltage by connecting the scope to the ends of the resistor. In addition to that, I will use the wavegen to send two waves which are 180 degrees out of phase and have an amplitude of 3.85V in order to mimic the transformer. Doing this should result in a voltage waveform between 2.9V and 3.1V.

iii) Key Measurement Results

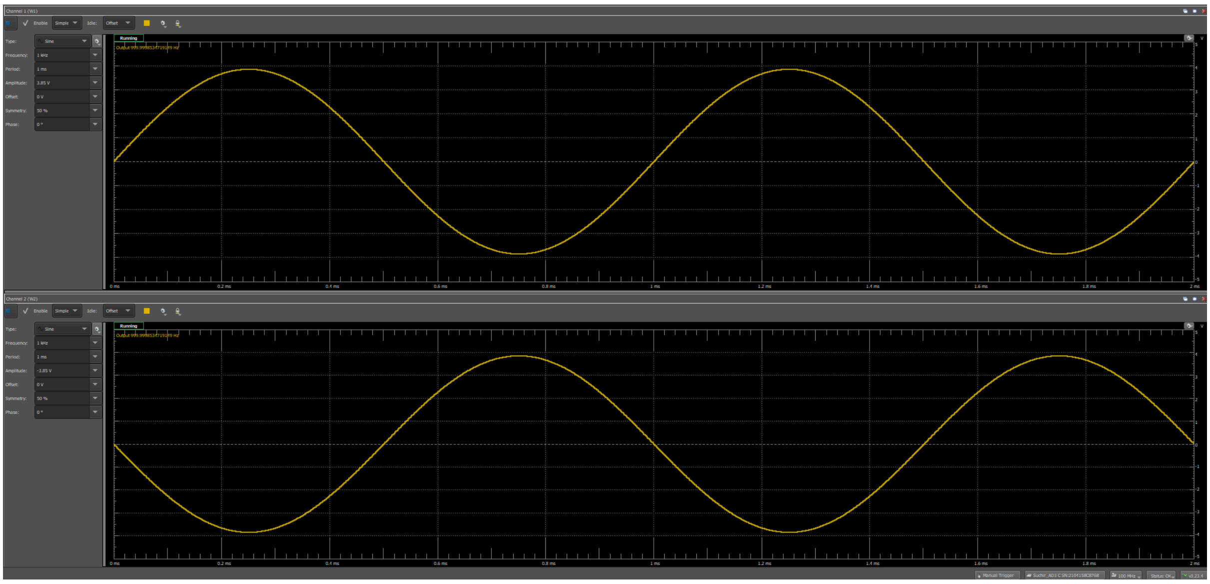


Figure 3: AC Input Sine Wave

As shown in the oscilloscope output below, the measured voltage falls in the desired range. Doing the math for the current (3V/300 ohms) gives you 10mA of current ensuring both values meet the requirements.

An important thing to note is that at the start I was not getting in the 2.9 to 3.1 range. In order to resolve the issue I had to increase my amplitude to 4V to generate the wave seen in the oscilloscope output.

iv) Oscilloscope Output

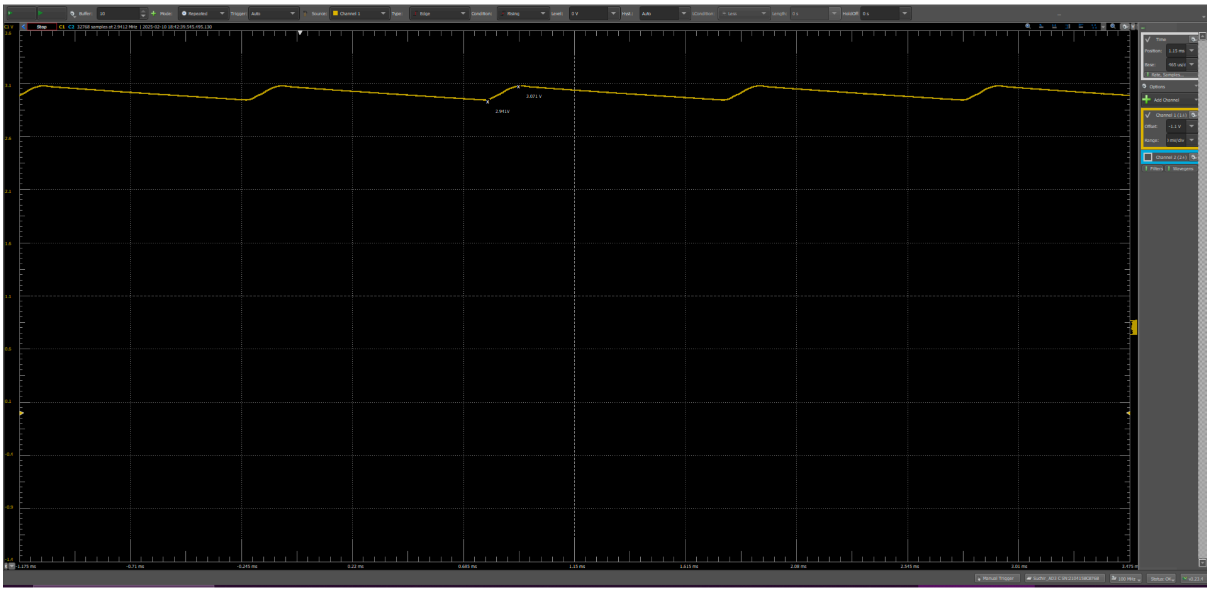


Figure 4: Output Voltage Waveform

Simulation

i) Circuit Schematic

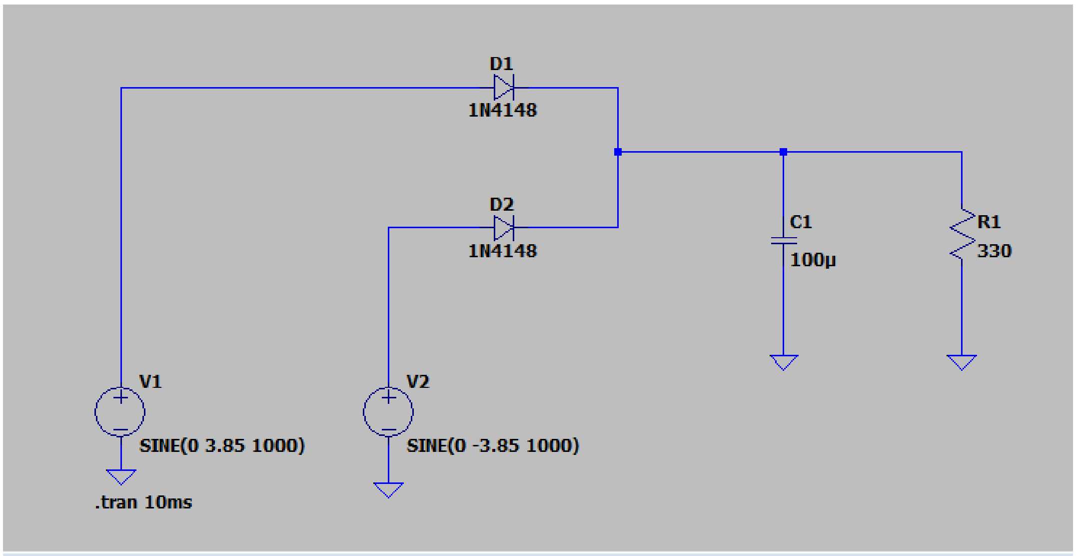


Figure 5: LTspice Circuit Schematic

ii) Netlist

ExpressPCB Netlist: C:\Users\suchi\Documents\LTspice\Draft2.net

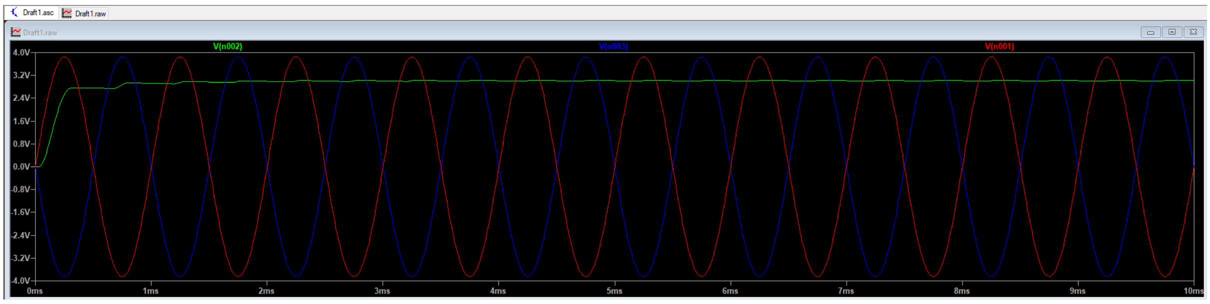
```
"ExpressPCB Netlist"
"LTspice"
1
0
0
""
""
""
"Part IDs Table"
"D1" "1N4148" ""
"D2" "1N4148" ""
"V1" "SINE(0 3.85 1000)" ""
"V2" "SINE(0 -3.85 1000)" ""
"R1" "330" ""
"C1" "100µ" ""

"Net Names Table"
"N001" 1
"N002" 3
"N003" 7
"0" 9

"Net Connections Table"
1 1 1 2
1 3 1 0
2 1 2 4
2 2 2 5
2 5 1 6
2 6 1 0
3 2 1 8
3 4 1 0
4 3 2 10
4 4 2 11
4 5 2 12
4 6 2 0
```

Figure 6: LTspice Circuit Netlist

iii) Simulation Conditions

Transient simulation with data being recorded from 0 to 10 ms and max timestep of 0.001 ms.

iv) Simulation Output



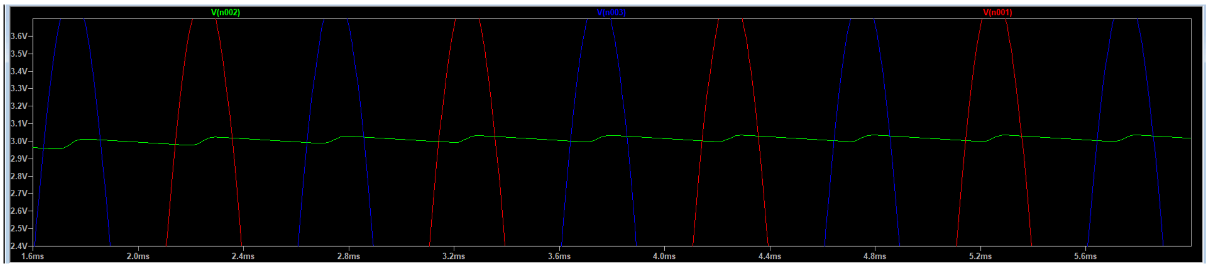


Figure 7: LTspice Simulation Results

The simulation outputs clearly indicate that my proposed schematic will give the desired results.

Discussion

After comparing the calculations I made during the design phase to the values obtained in measurement, I noticed a few small discrepancies. In my original design, I assumed a standard diode voltage drop of 0.7 volts. Based on this, I initially raised the voltage from 3 to 3.85 volts. However, after assembling the circuit on the board, I found that the expected output voltage wasn't achieved with an input of 3.85 volts. To address this, I had to further increase the input voltage to 4 volts. Upon investigation, I realized this issue stemmed from various factors, such as resistor tolerances, defects in components like wires, diodes, and capacitors, as well as environmental conditions like temperature. Moreover, real-world circuit behavior often deviates from theoretical models, particularly with capacitors, which may not function exactly as simulations predict.

A major design limitation I faced was that my AD3 was not capable of measuring current. Though it would have been nice to show the waveform I ended up using Ohm's law to prove that I am getting the right amount of current output. I didn’t face too many problems in this project. The only hard part about this was the research phase as this was my first time interacting with a circuit simulation software like LTspice. I also had to spend a lot of time learning about rectifiers and understanding transformers so I could give an in depth description of how they work and how they were used.

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.