Electronic Devices and Circuits I [ELECENG 2EI4] Project #1

Instructor: Dr. Haddara

Zayeed Ghori – L06 – ghoriz1- 400398943

Summary

The purpose of this project is to design a 3 +/- 0.1V DC voltage source from a 120V AC source at 1kHz frequency. The design simulates the use of a step-down transformer to convert the 120V AC to a suitable voltage to have 3V at the output using an AC low voltage source in the AD2. A rectifier should be used to convert the AC to DC and then a filter used to make the DC voltage more stable, and then an optional regulator to cap the voltage.

Design

i.   **Transformer: you will not use a transformer. Specify the input voltage(s) required for your design then determine the turns ratio of the transformer that would be needed if this input were to be generated from a 120V ac source. In your actual circuits and simulations, you will use the AD2 to provide the input(s) to your circuit.**

The input for my design was found to be 7.68V which was split into two mirrored 3.68V waves.

$$Vi = 3.84V - (-3.84V) = 7.68V$$
$$Vintrans = 120V$$
$$Vm = 120\sqrt{2} = 170V$$
$$a = \frac{170V}{7.68V} \approx 22 \; turns$$
$$22:1 \; ratio$$

ii. **Rectifier: identify and justify the rectifier topology used and identify the relevant diode parameters.**

The rectifier used is a centre tapped full wave rectifier which is identified by the two opposite polarity voltage sources. It is a full wave rectifier so the other half of the wave is not wasted as heat. It uses 2 diodes instead of a bridge rectifier's 4 diodes, also means a lower voltage drop at each cycle, leading to a higher efficiency.

Diode Specs:
$$Vbreakdown \; = \; 100V$$
$$Vf \; = \; 1V \; max, typ \; .84V \; @ \; 10mA$$
$$Ir \; = \; 25 \; uA$$

iii. **Filter: specify the filter used (including component values).**

The filter that was used is a single 100uF capacitor due to the formula for Vripple pp, the larger the capacitor value, the more stable the DC output voltage will be. In this case, the max acceptable voltage 3 +/- 0.1 V.

iv. **Regulator: a regulator is optional. State whether you used a regulator. If so, explain the design, including the values of components used.**

Regulator was not used.

Figure 1: Hand drawn circuit schematic
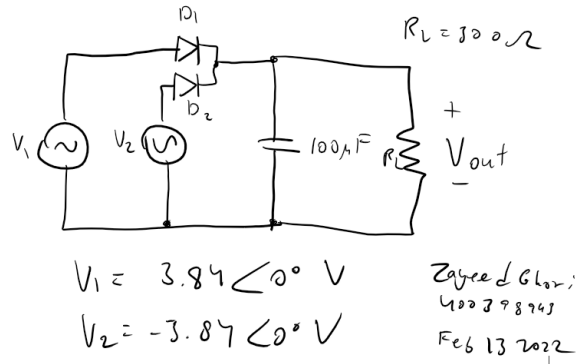
vi. **Show the calculations that led to the choice of all component values.**

The resistance of the resistor is found using the required voltage and current across it.

$$RL \ = \ 3V \ / \ 10mA \ = \ \text{300 Ohms}$$

The capacitance that was used was the only capacitor that I had that met the Vripple pp threshold (0.1V – (-0.1V)) = 0.2 V

$$V \ ripple \ pp \ \leq 0.2 \ V$$

$$C \geq \ 10 \frac{mA}{1KHz \ * \ 0.2V} \geq 50uF$$

$$\text{100uF} \geq \text{50uF}$$

vii. **Specify the expected performance according to your design calculations.**

$$\eta \ = \ Pdc/Pac$$

$$\eta \ = [\ (2 \ * \ \frac{10mA}{\pi})^2 \ * \ 300 \ ohms] \ / \ [(\ 10mA \ / \ \sqrt{2})^2 \ * \ (.84/10mA \ + \ 300 \ ohms)]$$

$$= \ .812 \ / \ (1 \ + \ .84 \ V/0.01mA/300 \ Ohms)$$

$$= \ 63.4\%$$

viii. **Discuss design tradeoffs, design margins, component ratings, safety, and other issues that you considered in your design?**

Instead of using a full bridge rectifier which used 4 diodes, opting for a 2 diode design reduced the number of voltage losses across different diodes, this means that only 1 diode would have a drop across of it at a time. But this would mean that two sources would be used. Design margins would include the capacitor value required for a valid Vripple and the output

limit of the AD2 board of 5V on each channel. The diodes had a rated current over 10mA and the capacitor had a rated voltage over 3V.

Measurement & Analysis

i.   **Include a photograph of your actual circuit. The photo must include a paper with your name, student number, and the date.**
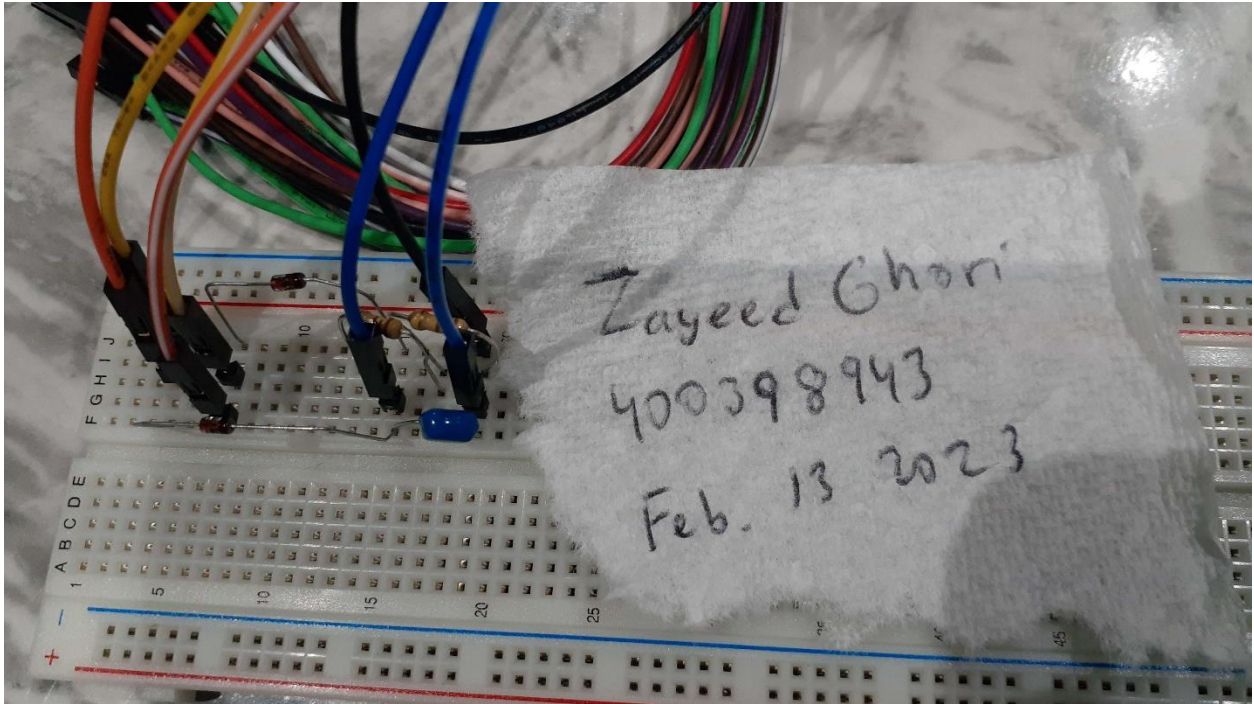

ii.   **Explain the measurement procedure: how did you determine the performance?**

I measured the input peak and output voltages and calculated the performance. The main factor was that the output dc voltage did not fluctuate more than the +/- 0.1V tolerance from 3V.
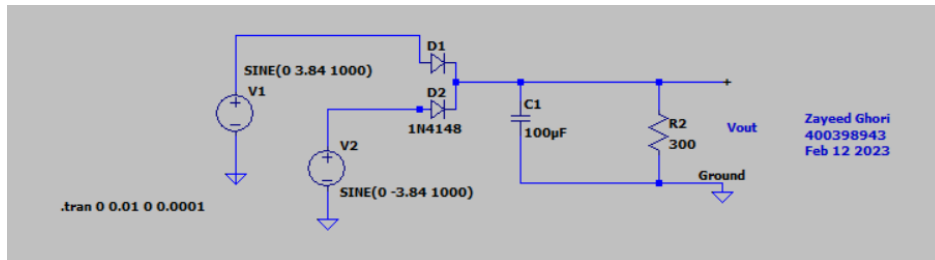
iii.   **Provide the key measurement results.**

$$Vdc = 2.95 \ V \ +/- \ 0.7$$
$$I = 9.84 \ mA$$

iv.   **Include screenshots of your oscilloscope output, clearly marked to show the performance obtained.**
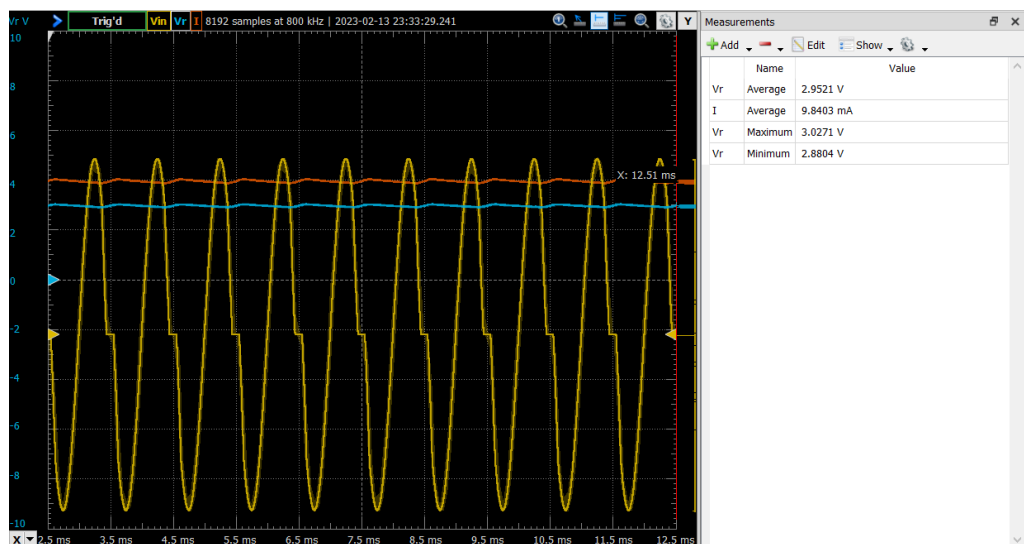
Simulation

i. Circuit schematic from the simulator.

ii. Netlist.

```
* D:\Downloads\Project1.asc
D1 N001 + 1N4148
V1 N001 0 SINE(0 3.84 1000)
C1 + 0 100µF
R2 + 0 300
V2 N002 0 SINE(0 -3.84 1000)
D2 N002 + 1N4148
.model D D
.lib C:\Users\zayee\AppData\Local\LTspice\lib\cmp\standard.dio
.tran 0 0.01 0 0.0001
* Vout
* Zayeed Ghori\n400398943\nFeb 12 2023
.backanno
.end
```

iii. Simulation conditions. This includes type of simulation (operating point, transient, ac sweep, etc.) and the simulation parameters (e.g. simulation time).

The simulation was a transient sweep, and the simulation time was from 0 to 0.01s.

The voltage was measured from the top of the resistor to ground and current was measured through the resistor.

iv. Simulation output. Choose the output figures and numbers required to specify the performance of the circuit.

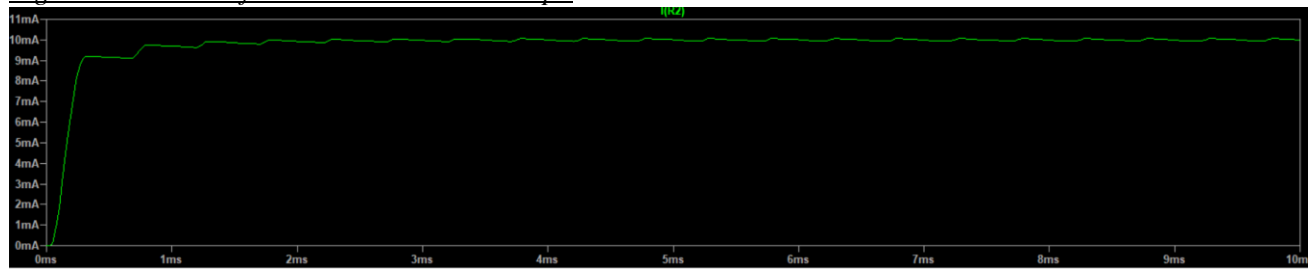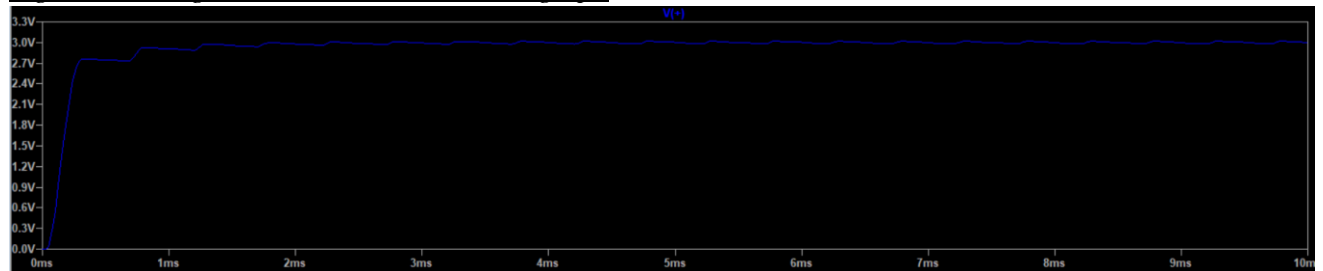*Figure 2: Current of Resistor Simulation Graph*

Figure 3: Voltage across resistor simulation graph

Discussion

i.   Compare the results from design, simulation, and measurement.

The outputs of my circuit design, simulation, and measurement were very similar despite variations in accuracy levels.

	Vout (V)	Iout (mA)
Simulation	3.00 +/- 0.01	10
Measurement	3.05 +/- 0.7	9.84

The error in the simulation was: (3V – 2.99)/3V = 0.33%
The error in the measurement was: (3V – 3.75)/3V = 25%

The simulation managed to keep a very steady 3V while the real circuit barely managed to keep the voltage within spec.

ii.   Discuss any discrepancies observed.

The existence of noise. Numerous things can cause noise, including electromagnetic interference and thermal noise. Such noise can cause errors or inconsistencies in the circuit's functioning as well as inaccurate voltage and current readings. Strategies like shielding, filtering, and correct grounding can be used, among others, to lessen the effects of noise. The accuracy and dependability of the circuit can be greatly improved by carefully analysing and addressing the various sources of noise.

The lack of a regulator means that if a load drew too much current, the voltage would start dropping below what was desired. Adding one may have made it more versatile, adding a shutoff if the voltage dropped below the 3 – 0.1V threshold.

iii.   Discuss the limitations of the design and the limitations of the measurements you performed.

The main limitation was the requirement to limit the source voltage to below 5V due to the max voltage output of the AD2 which was limited to +/- 5V. Also the input waveforms were not synced to 0 degree phase, so the input voltage did not look right when measured.

iv.   Describe any problems encountered in measurements and the troubleshooting steps you took

I encountered incorrect voltage drops across the resistor, around 1.9V when the design was supposed to output 3V. I rebuilt the circuit simpler with less wires, also to reduce any losses through wires, and figured out that the diodes were connected in the wrong direction. I also was confused about the AD2 showing out of phase waveforms when I measured them, even though the phase was set to 0. I realized that the phase shift did not affect the output voltage to a great extent, but it made the input voltage look less than it was.