EE 2EI4 - Electronic Devices and Circuit I

Project 1

Stefan Tosti - 400367761

Summary

We are tasked with Designing and building a DC power supply that delivers 10mA at 3V ± 0.1V that is 120V ac (rms) at 1kHz. This circuit design will not use an actual transformer, and will instead mimic the output of a transformer using our AD2. This design should rectify all negative cycles of an AC signal, and then generate a smooth ripple using a simple capacitor filter, as well as a regulator to smooth out the final signal.

Design

Part I - The Transformer

Although this design does not incorporate the use of an actual transformer, we simulate the use of a step down transformer through the input of our circuit. Using a transformer with an approximate turn ratio of $a \simeq 39$ we are able to achieve a sinusoidal input with 4.4V peaks, corresponding to 3.1V rms. This 4.4V input was simulated, using the AD2's integrated DC power supply. The hypothetical transformer used in this project is a center tapped transformer. The design choice will be explain in more depth in the following subsection in this project (Part II - The Rectifier).

Part II - The Rectifier

The rectifier that I have chosen to use for my circuit design is a center tapped full wave rectifier. I decided to go with this design because of simplicity. The center tapped rectifier uses only 2 diodes, whereas the bridge rectifier uses 4 diodes, which would likely cause greater experimental error in my final measurements. The use of a center tapped rectifier also limits us to a center tapped transformer with respect to the transformer topology. The use of a center tapped transformer means that at any given moment, we will only be getting half of the output voltage, compared to if we were to use a conventional transformer. Each branch of the transformer output will be attached to a single diode. Since the input AC voltage only has to go through one diode in forward bias, the output waveform is unipolar, and experiences a voltage from of $V_d = 0.7V$.

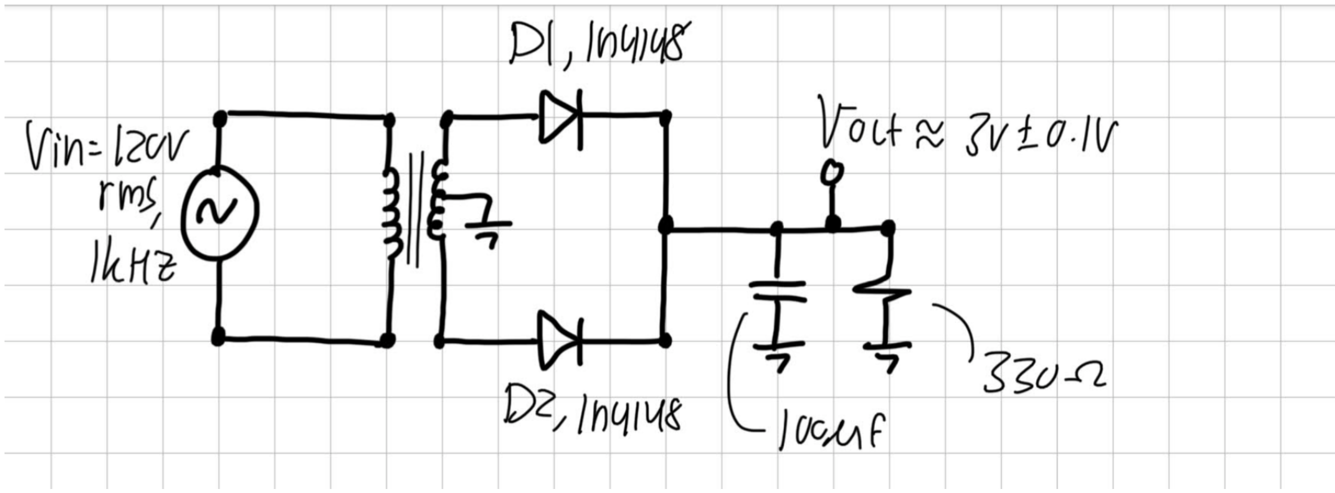
Part III - The Filter

The simplest type of filter is a capacitor in parallel with the load. From equation 4.27(b) in our course textbook, we can see that $V_{ripple} = \frac{I_{out}}{fC}$ which we can rearrange for the desired value of capacitance, giving us $C_{min} = \frac{I_{out}}{f \cdot V_{ripple}}$. By plugging in our respective values, we get a minimum capacitance requirement of $50\mu F$. More on this calculation can be read in the 'Calculations' section of my report.

Part IV - Regulator

A regulator is used to smooth out particularly noisy peak to peak voltage ripples. The simplest form of regulator is a zener diode. In my particular circuit, a zener diode regulator was not required.

Part V - Circuit Schematic



Part VI - Calculations

	Theoretical Calculation
AC Wavegen	$V_o = 3V$ $V_D = 0.7V$ $V_i - V_D = V_o \Rightarrow V_i = V_o + V_D = 3V + 0.7V = 3.7V$ … Since we actually want our peak voltage to be at $\pm$ 0.1V, we should actually be using an input amplitude of $3.8V$
Transformer Turn Ratio	$Turn\ Ratio = \frac{120V\ (rms)}{3.1V\ (rms)} = \frac{120\sqrt{2}}{3.1\sqrt{2}} \simeq 39\ turns$ The above turn ratio calculation is based on the actual input voltage of 4.4V which is used to achieve the desired DC output voltage
Diodes (#: 1N4148)	$V_D = 0.7V$ (assuming constant drop model)
Capacitor (#: CM107)	$V_{ripple} = 3.1V - 2.9V = 0.2V$ $I_{out} = 10mA = 0.01A$ $f = 1\ kHz = 1000\ Hz$ $C_{min} = \frac{I_{out}}{f \cdot V_{ripple}} = \frac{0.01A}{1000Hz \cdot 0.2V} \simeq 50\mu F$ My calculations above suggest that a $50\mu F$ capacitor (at minimum) would need to be used to produce a suitable ripple. I will use one with the largest possible capacitance, as a larger capacitance will result in a smoother output
Resistor (Ideal: 300 Ω) (Actual: 330 Ω)	$R = \frac{V}{I} = \frac{3 \pm 0.1V}{0.01A} = [290,\ 310]\Omega$ The above calculations suggest that a resistor within the range of 290 to 310 ohms will be suitable for this design. Despite this, the completed circuit will

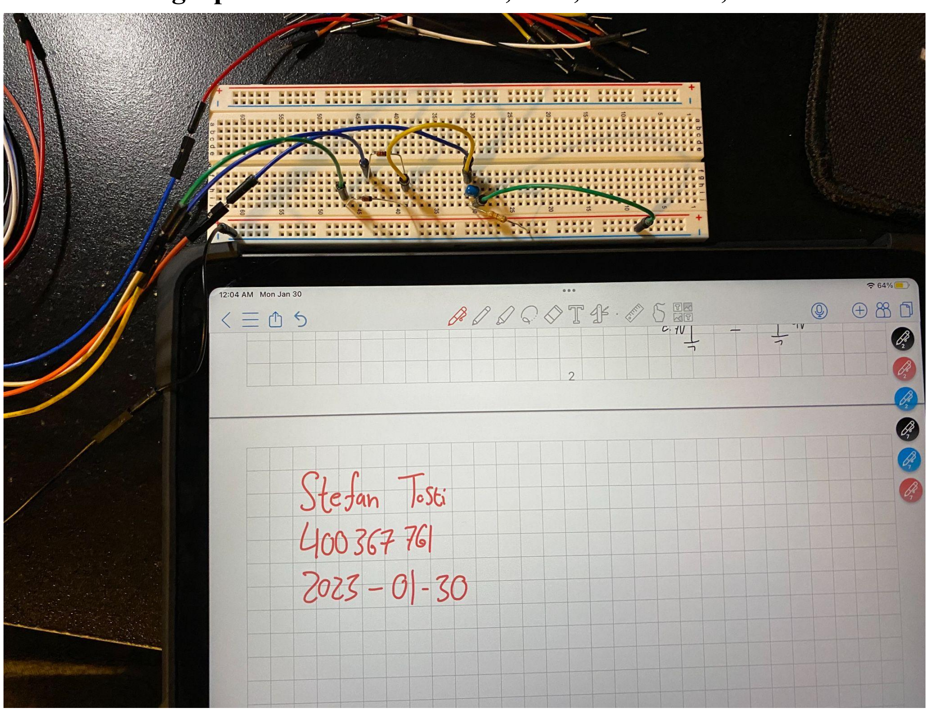
	use a 330Ω resistor, as per Dr. Haddara's suggestion

Part VII - Expected Performance
According to the above calculations, we should be using an input signal of 3.7V amplitude, and 1KHz frequency. Two of these waveforms should be generated, each supplied to one of the two diodes being used. Due to the topology chosen, these input waves should be the negative of each other, meaning that when one is on its positive cycle, the other should be on its negative cycle. Because of this, when D1 is conducting, D2 will be non-conducting, and when D1 is non-conducting, D2 is conducting. This will, in turn, produce a fully rectified wave. Since the waveform will only be going through one diode at a time, I have taken into account a 0.7V drop when choosing the amplitude of my input wave, which is why the amplitude is 3.7V, rather than the desired 3.0V. The capacitor in use should produce a smooth ripple, that limits the output waveform to ± 0.1 V.

Part VIII - Trade Offs
There were several trade offs made during the construction of this circuit. The first of such tradeoffs was the use of a 330Ω resistor, rather than the calculated 300Ω. Our component kits only had 330Ω resistors, and thus this was the closest option. A decision had to be made whether or not to use a single 330Ω resistor, or use a series combination of 3 × 100Ω resistors. I made the decision to go with the single 330Ω resistor out of simplicity, and consistency. In terms of simplicity,

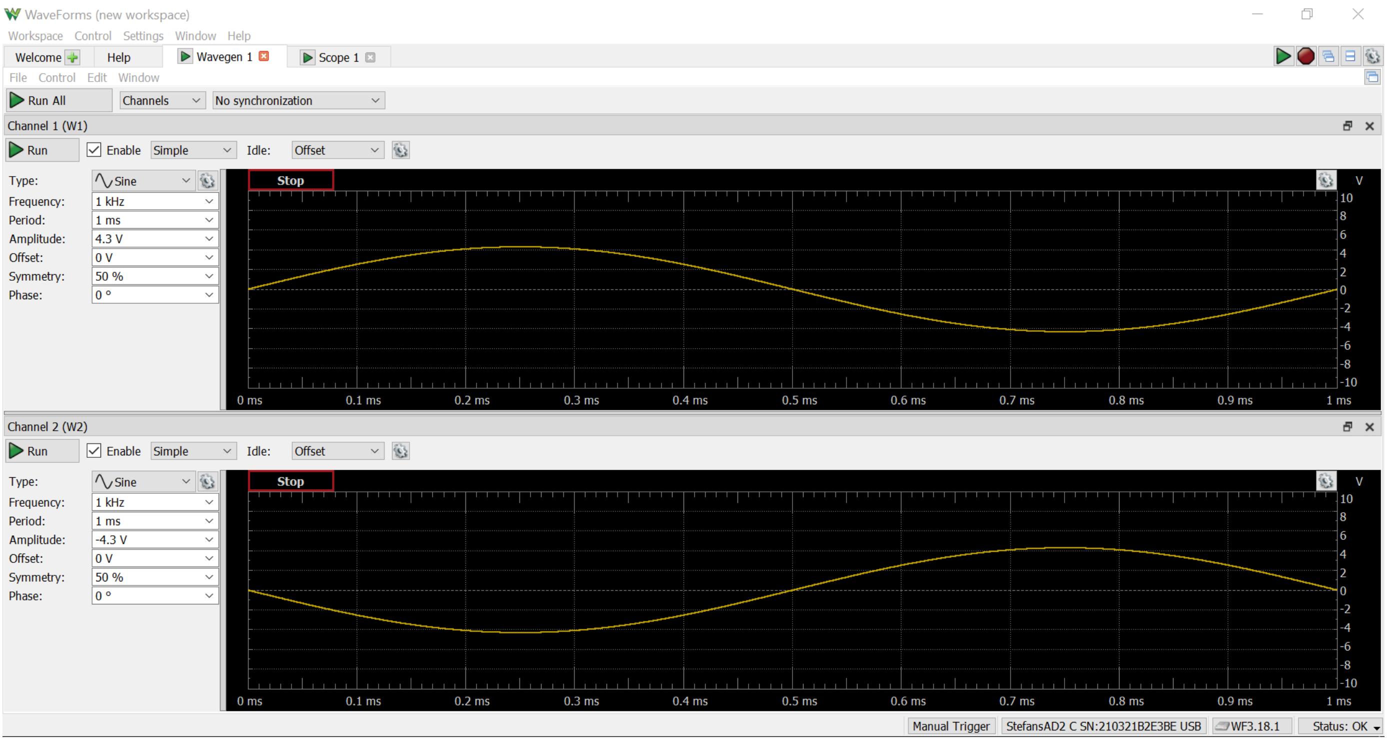
Measurement and Analysis

Part I - Photograph of circuit with name, date, student ID, student Number, Time



Part II - Measurement procedure

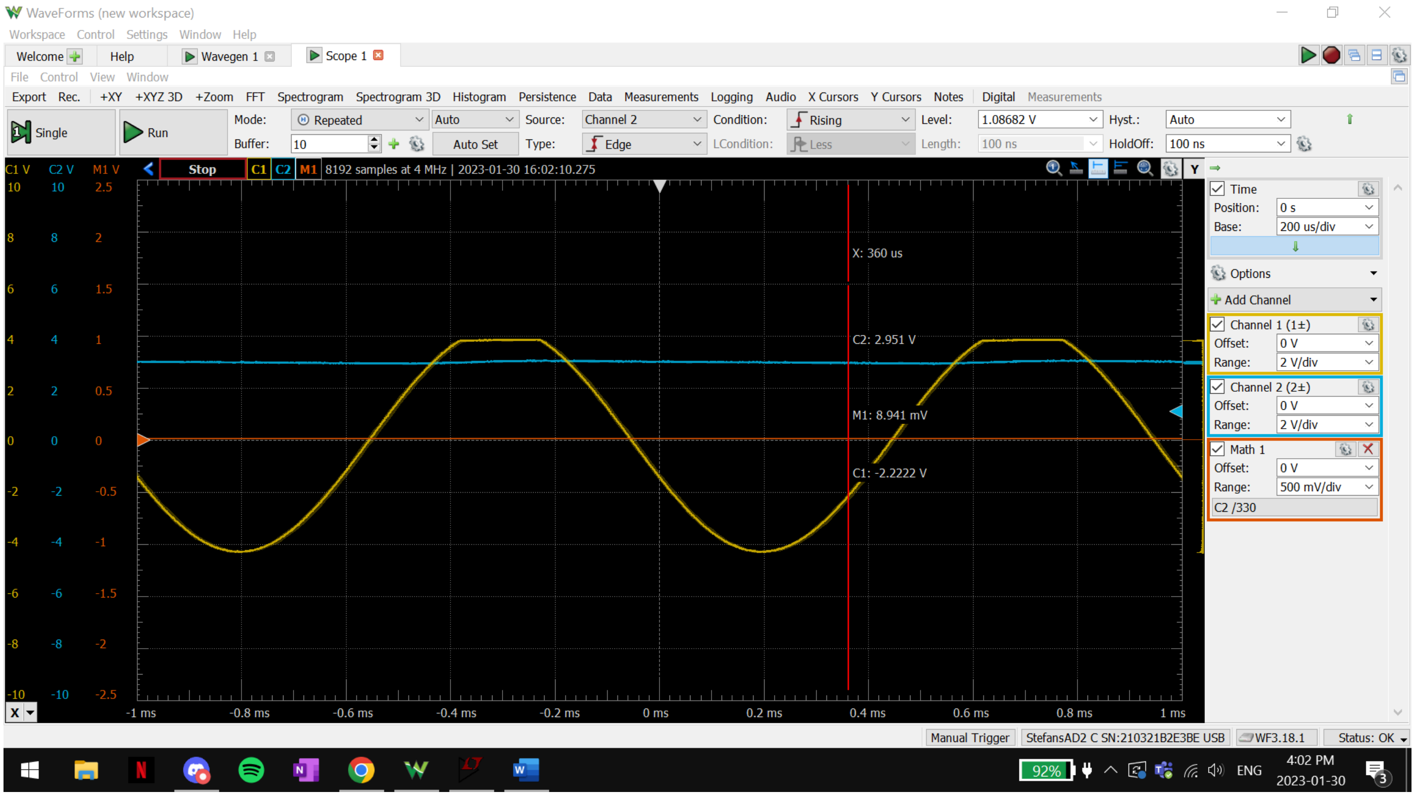
To determine the performance of the circuit, the oscilloscope was used to show the output of the circuit itself (the final ripple produced by the circuit) as well as the rectified wave, after going through the center tapped rectifier. I measured the voltage at these two points in the circuit with the 2 waveform inputs on the AD2. Additionally, since the AD2 does not have a means of measuring current, I implemented my own math function, which takes the output voltage on channel 2, and divides it by 330 to find the current.



Part III - Measurement Results

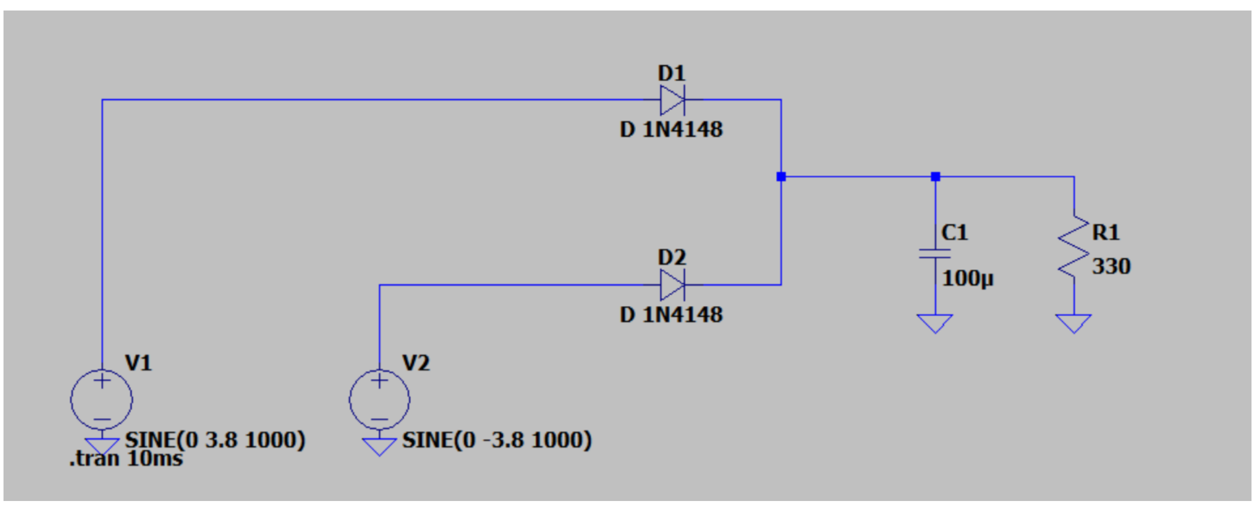
As can be seen in the “Oscilloscope output” section of my report, the circuit was able to meet specifications, but I had to increase the amplitude of the input waveform in order to make up for unaccounted for errors. The oscilloscope output displays an output voltage of about 2.951 volts, and an output current of 8.941 mA. The reason that the input amplitude had to be increased was because there were many unaccounted for errors in the design of the circuit. I considered the voltage drop of a diode to be 0.7V, however the 1n4148 diodes are rated for 1V. There was also errors that were out of my control, such as resistance in wires, and variability in resistance performance.

Part IV - Oscilloscope output



Simulation

Part I - Circuit Schematic



Part II - Netlist

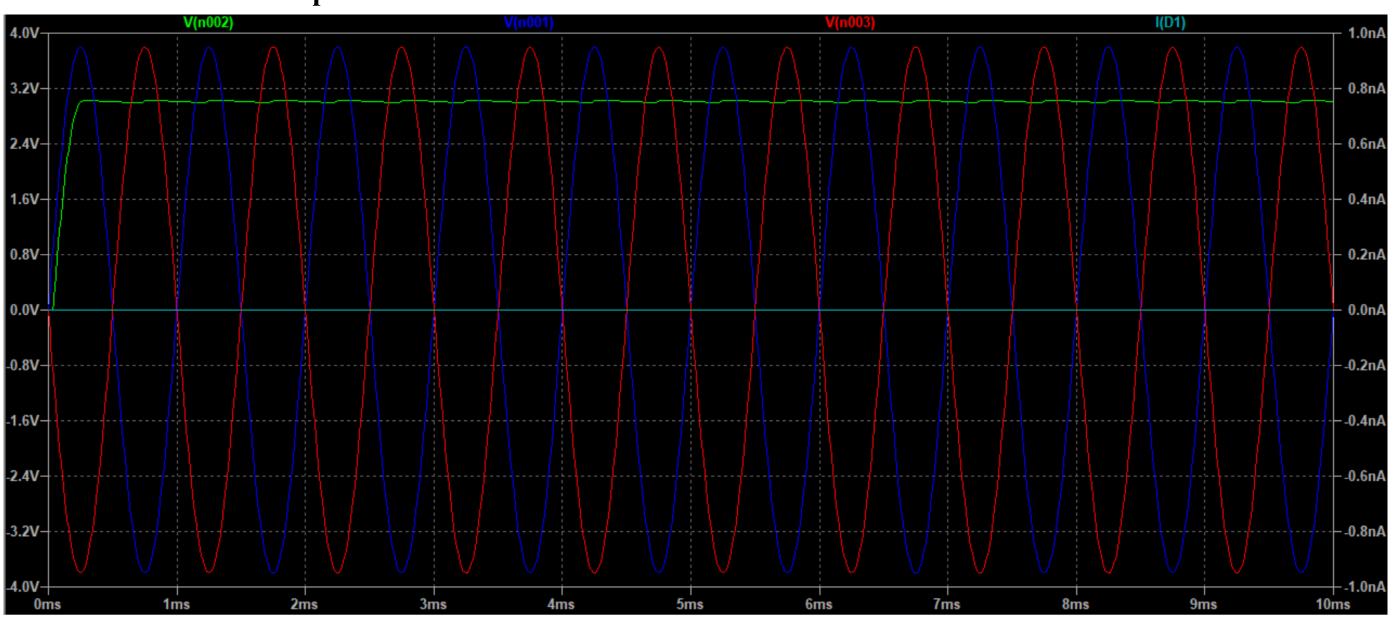
SPICE Netlist: C:\Users\tosti\AppData\Local\LTspice\Draft1.net

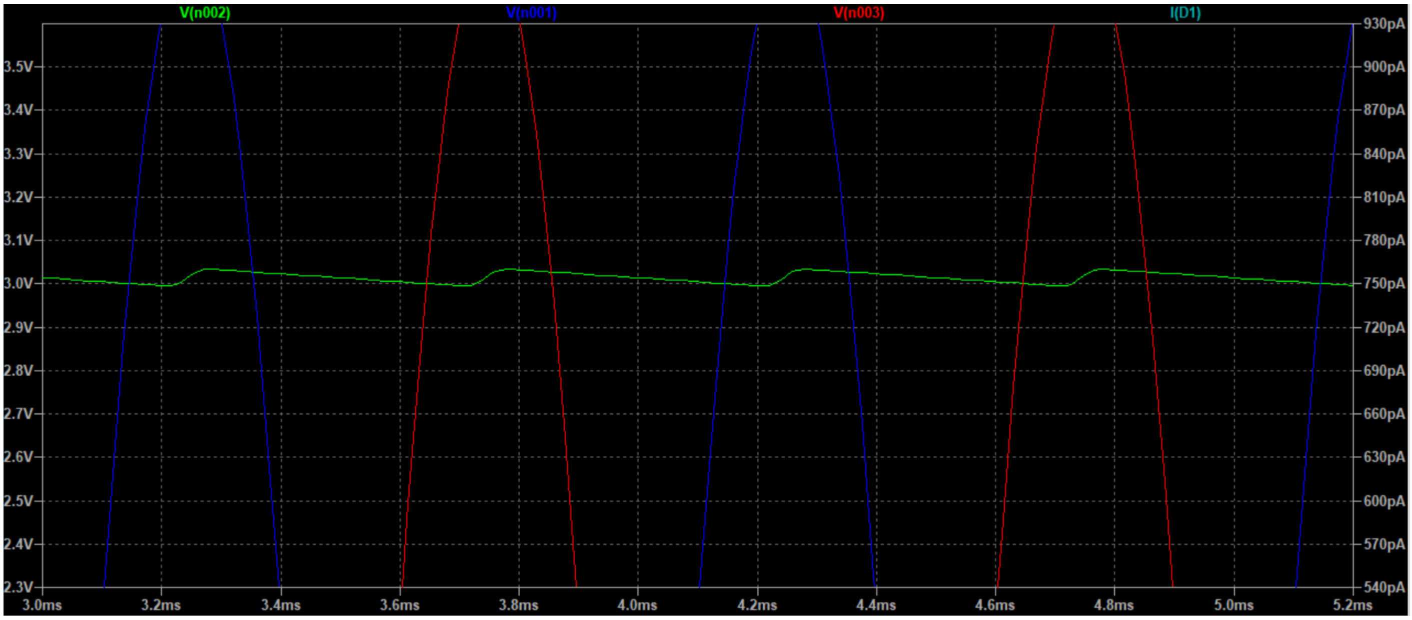
```
* C:\Users\tosti\AppData\Local\LTspice\Draft1.asc
V1 N001 0 SINE(0 3.8 1000)
V2 N003 0 SINE(0 -3.8 1000)
D1 N001 N002 D 1N4148
D2 N003 N002 D 1N4148
R1 N002 0 330
C1 N002 0 100µ
.model D D
.lib C:\Users\tosti\AppData\Local\LTspice\lib\cmp\standard.dio
.tran 10ms
.backanno
.end
```

Part III - Simulation Conditions

My simulation was performed as a transient simulation, and data was set to record from 0s to 10ms

Part IV - Simulation Output





As can be seen from the pictures above, our output waveform is perfectly centered on 3.0V, and is clearly within the range of ± 0.1V. My first trial run had an input wave of amplitude ± 3.7V, as per my calculations above. This resulted in an output waveform that was just slightly too low, around 2.9 ± 0.1V. As a result, I decided to increase my input to 3.8V in amplitude, which yielded the above graphs.

Discussion

Part I & II - Result comparison & Discrepancies

Comparing my results from my theoretical calculations, my physical circuit measurements, and my circuit simulation shows that there were many slight differences in the values obtained for each step of the design process.

In the calculations, I obtained values that suggested I use an input waveform of ± 3.7V, a resistor of 300Ω, and a capacitance value of greater than or equal to 50 μF. These values needed to be altered very sleightly to cooperate with my physical means. In real life, I only had access to a 330Ω resistor, as well as a 100μF capacitor. These component values were what was used in my real life measurement, as well as my simulation, to keep consistency.

In my simulation, I had initially tested my device with the calculated input signal having a 3.7V amplitude, a 100μF capacitor, and a 330Ω resistor. This produced a waveform that was slightly out of the desired range, and was within 2.87V to 2.92V. As a result, I slightly increased the amplitude of the input signal to 3.9V, which resulted in an output waveform that was perfectly aligned with the desired 3 ± 0.1V output.

There was also a very large discrepancy noticed between the simulation output and the results achieved by the AD2 implementation. The amplitude of the waveform required when using the AD2 was substantially higher than expected. I believe that this is a discrepancy with an intrinsic part of the AD2.

The AD2 limits current when the only supply is a computer, as it caps out at around 400mA. This means that the very large 100µF capacitor does not have time to charge and discharge.

Part III - Design Limitations

The two most impactful limitations on my design were the resistors and diodes that I had to use. As previously mentioned, my design was constrained to a 330Ω resistor, instead of the calculated 300Ω resistor. This was likely a cause for some errors in the experimental and simulation analysis, and could have been a reason that the 3.7V amplitude was not suitable in the ideal simulation environment. In addition to this, the diodes that we had access to were 1N4148 diodes, which are rated for a forward voltage drop of 1V. This is inconsistent with my calculations, which were based on a forward voltage drop of 0.7V. There was also a limitation posed by the AD2, since the device itself cannot measure current. I had to implement my own current math function, which is likely not a true representation of the current in the system.

Part IV - Problems encountered and troubleshooting steps

The biggest problems encountered for this project was building and testing my circuit on the AD2. Using the AD2, I encountered countless problems. My first problem was that I forgot to connect the other end of my oscilloscope probes to ground, which resulted in my output waveforms being extremely noisy. Another problem that I ran into was with using new components. I was unaware of the polarity of physical diodes, so to troubleshoot I sent 3V into a diode, and measured the other end of the diode on the oscilloscope, switching the diode around between attempts.

As discussed earlier, another problem was with the expected input voltage. My calculated input waveform was not suitable for my circuit meeting spec, and thus I had to increase it to approximately 4.3V to achieve the required specification.

References

[1] Adel s. Sedra and Kenneth C. Smith, *Microelectronic Circuits Eighth Edition*, United States of America: Oxford University Press, 2020.

[2] Last Minute Engineers, *The Full-Wave Bridge Rectifier, Last Minute Engineers*, Jan. 29, 2020. https://lastminuteengineers.com/the-full-wave-bridge-rectifier/ (accessed Jan. 27, 2023).

[3] Cantho Automation*, Full wave center tapped rectifier with capacitor filter , How it works?, N.d.* https://canthoautomation.com/full-wave-center-tapped-rectifier-with-capacitor-filter/. (accessed Jan. 27, 2023)