# 2EI4

# Project 1- DC Power Supply

By, Vansh Dhodi, 400517436

Due, February 9, 2025

# Summary:

The purpose of this project is to design a DC power supply. This task entails designing a transformer, rectifier, filter and regulator for a DC power supply which can deliver 10 mA at 3V ± 0.1V from a source that is 120V (rms) at 1 kHz.


# Design:

## Transformer:

The transformer in a DC power supply is typically used during the AC to DC process where the AC voltage is changed to a different level whether it being increased or decreased before it is rectified. For the project a transformer did not need to be physically used but rather only designed to determine the turns ratio necessary for the input voltage. The design I chose consists of a center tapped transformer which is crucial to my rectifier design as I plan on using a center tapped full wave. The center tapped transformer allows for the voltage to be split to have both negative and positive AC, both of equal value going into my rectifier.

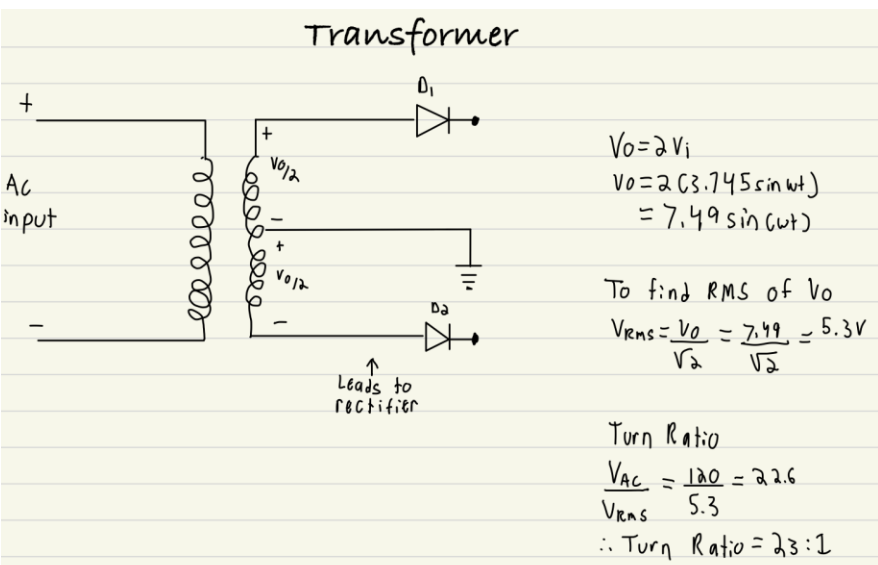


Figure 1: Transformer Calculations

# Rectifier:

The rectifier I chose for my design is the Center Tapped Full Wave Rectifier. The Center Tapped Full Wave Rectifier utilizes both AC half cycles that the Center Tapped Transformer provides thus doubling the efficiency of the rectifier compared to a half wave rectifier. It also has a higher ripple frequency as it is able to double the AC frequency. This allows for the filtering process to happen with ease and makes the DC output smoother. The use of only two diodes for a Center Tapped Rectifier compared to a bridge rectifier's 4 also allows for there to be a smaller voltage drop between components as the center tapped only utilizes one diode at a time. The peak voltage for the rectifier is 3.1V with a ripple voltage of 0.1V. The two voltage sources for the rectifier will have one positive source and the other will be negative or specifically 180 degrees out of phase. This is a property of the Center Tapped Full Wave Rectifier as it has two AC half ways where both are equal value with one being negative. The diode that the voltage goes through has a drop of 0.72V.

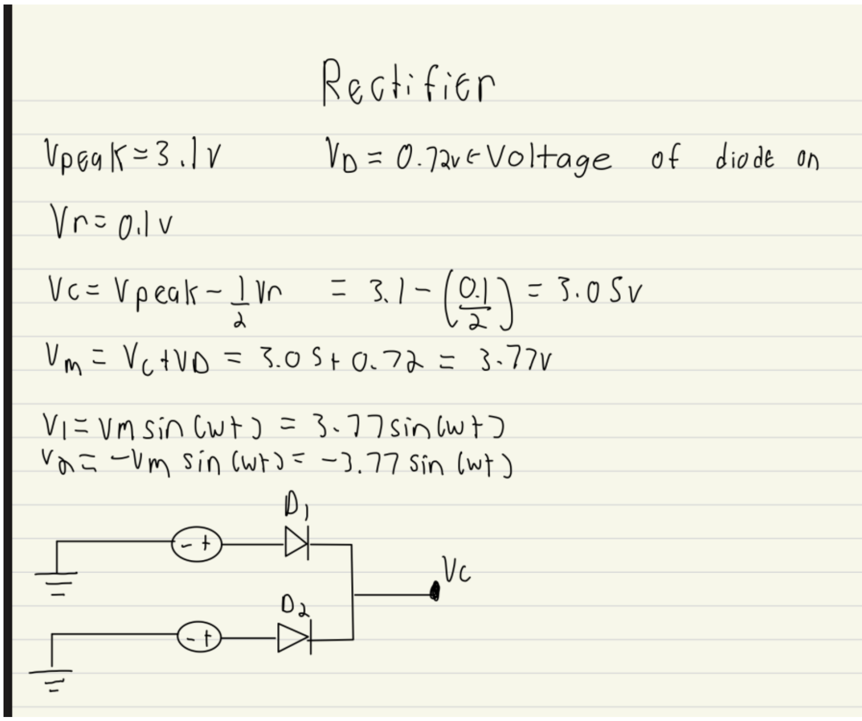


Figure 2: Rectifier Calculations

## Filter:

Due to Design limitations the filter I chose to pursue is the RC filter. The counterpart to this would be an LC filter but this requires an inductor which isn't available in our kit. Now the tradeoff with this filter is that it dissipates more heat compared to the LC filter. However, due to component limitations an RC filter had to be utilized. In terms of the design, I will be using 0.1V as my ripple voltage and from doing the calculations found in the image below it can be seen that I will need a capacitor with 50microfarads or higher. The only suitable capacitor I have available is the 100Microfarads capacitor I will be using in my design. Another aspect I needed to find was what load resistor to use which was calculated by dividing the load voltage by the current. This produced a 300-ohm resistor.

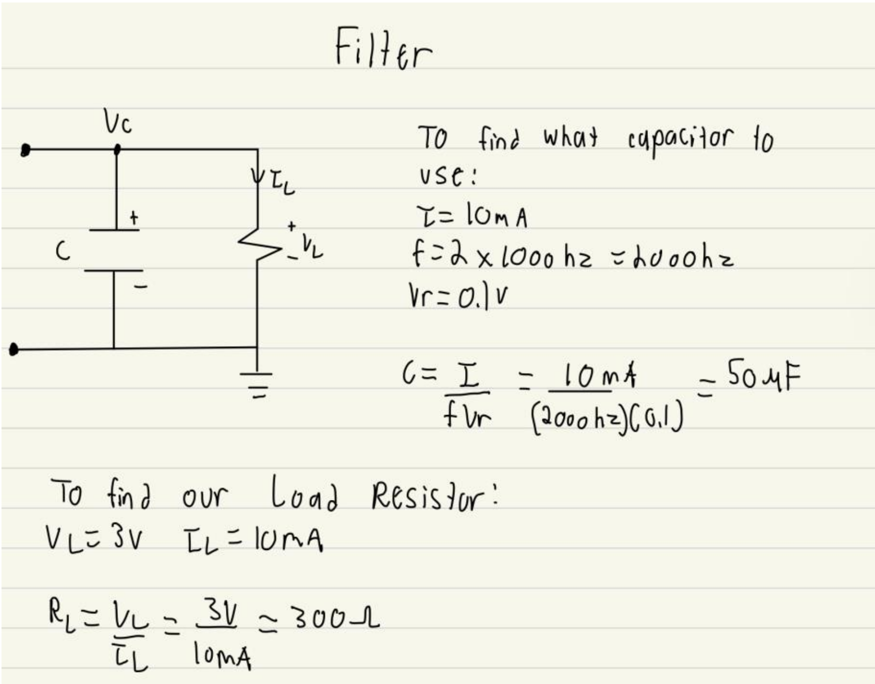


Figure 3: Capacitor and Load Resistor Calculations

## Regulator:

My design will not be utilizing a regulator.

## My Schematic:

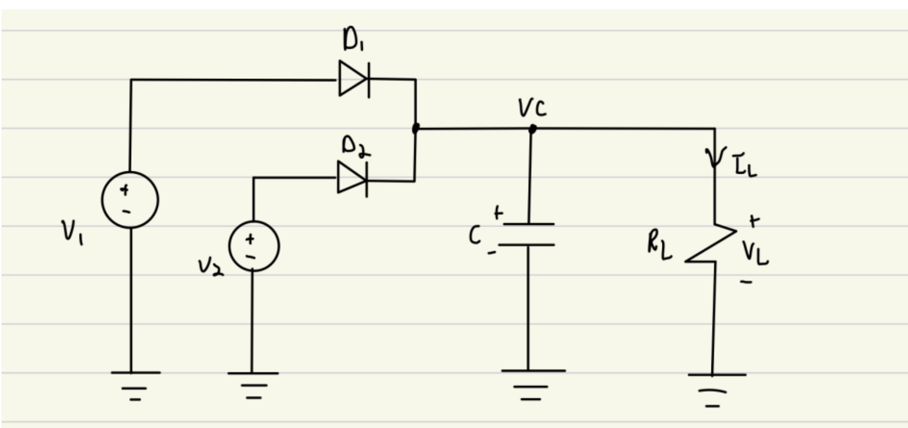


Figure 4: My final design Schematic

## Measurement and Analysis:

Due to some component limitations my physical circuit had to utilize a 330ohm resistor rather than the 300ohm one calculated and used in the simulation and due to the only capacitor close to 50microfarads being 100microfarads I had to improvise and utilize that.

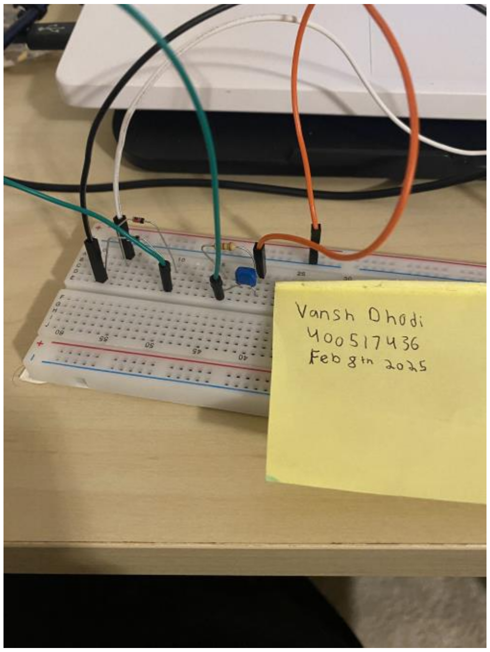


Figure 5: Physical Circuit

The wave obtained from implementing the designed circuit to a breadboard and measuring the voltage across the load resistor matches the intended values. As can be seen the wave oscillates from 2.92V up to 3.08V, this matches the condition of a 3V ± 0.1V output. Utilizing V=IR it can be found that the current ranges from 8.85mA to 9.33mA. Now this is the current calculated using the 330ohm resistor which is what had to be used for the physical design. If a 300ohm resistor was used the values would be much closer to the expected 10mA output

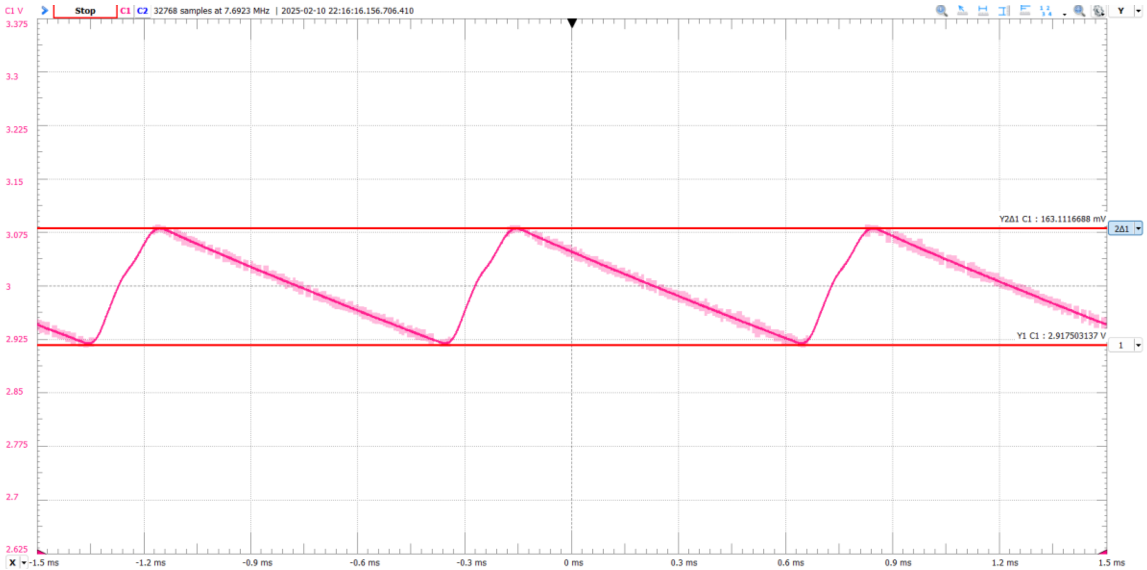


Figure 6: Waveforms Simulation

The image below showcases the AC input as we can see the two waves are out of phase by 180 degrees which is necessary for the rectifier to correctly work as one AC source will go to one diode and the negatice AC source (phase shifted by 180 degrees) will go to the other diode.

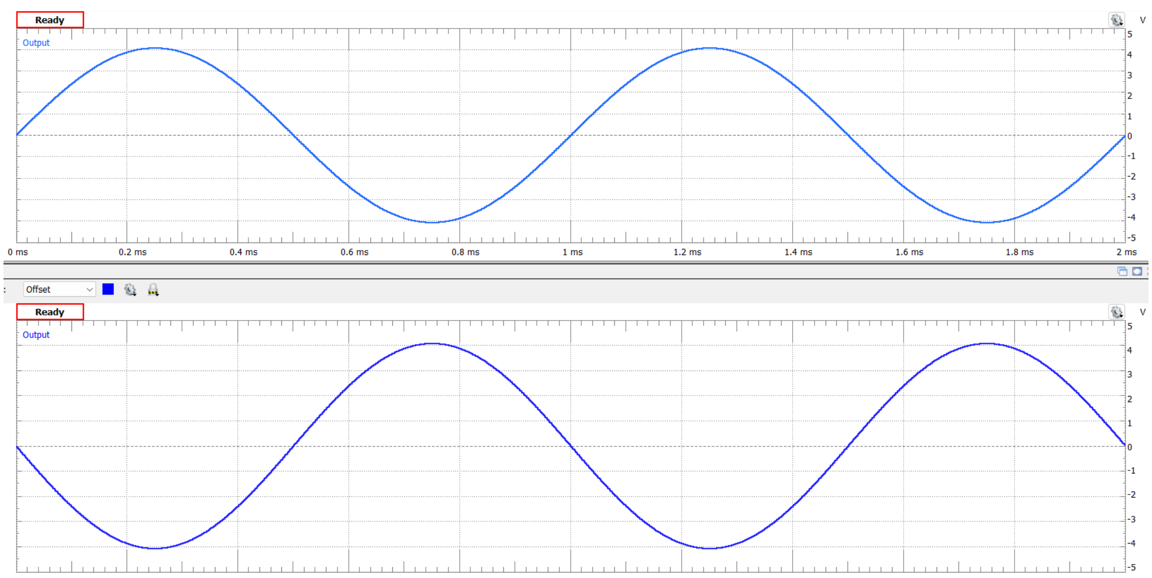


Figure 7: AC input

## Simulation:

Before the physical circuit could be made the theoretical values calculated for both the resistor, capacitor and Vsources needed to be tested. This is a necessary step in terms of safety to make sure no harm comes, or any issues arise when the physical circuit is made. Using Pspice my final design was created as a schematic and later ran in a simulation to show the results of the voltage over the load resistor and the current.

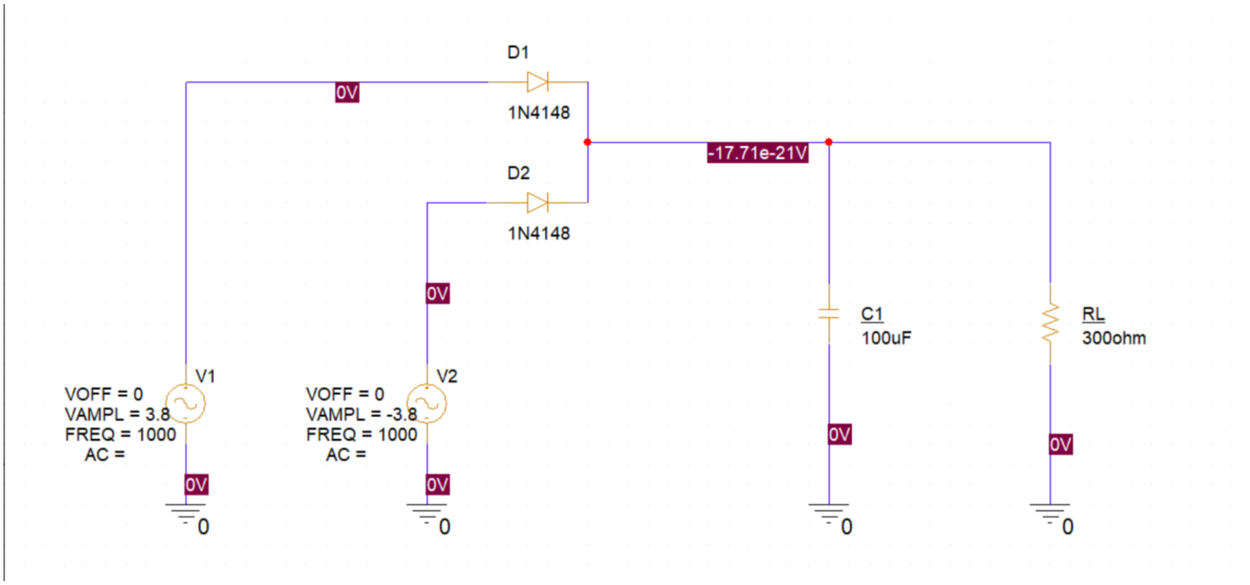


Figure 8: Pspice circuit

In the image below the simulation was ran where the time limit was set for 7ms, and it began simulating from 2.5ms. With the voltage being set to 3V with a maximum tolerance of ± 0.1V. It can be seen that this condition was successfully achieved in the top graph (yellow line). That graph measures the value of the voltage across the load resistor and the displacement of the values from 3V can be seen to go down to 2.99V and then up to 3.03V which is well within the tolerance range. This proves that the circuit was successful in the simulation. Now if we look at the bottom graph that is measuring the current going into the load resistor, it follows a similar pattern to the voltage graph, and this would make sense since the two are proportional.

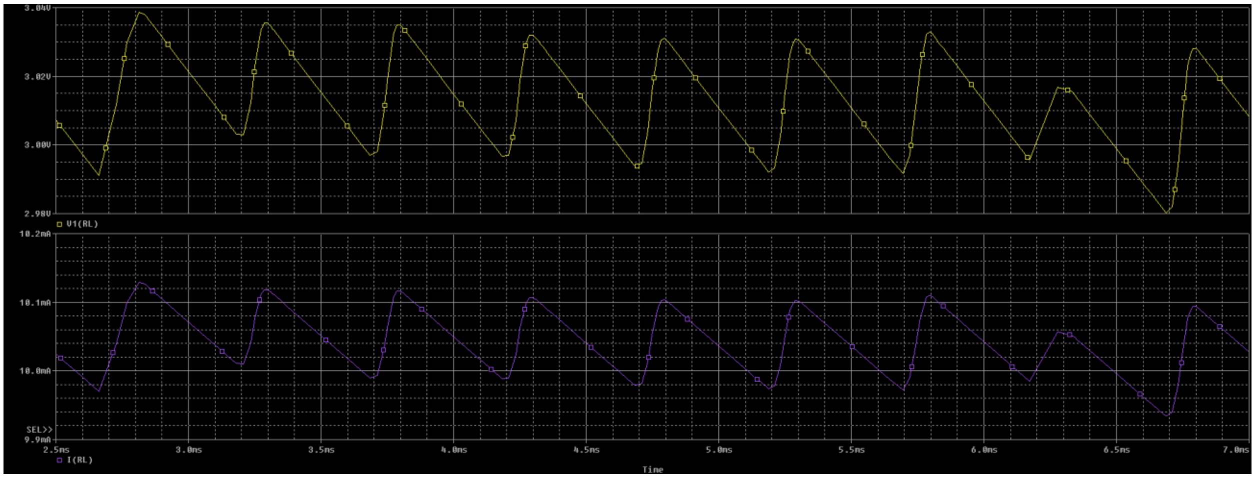

Figure 9: Pspice Simulation

```
1: * source 2CJ LAB 1
2: V_V2          N01463 0
3: +SIN 0 -3.8 1000 0 0 0
4: C_C1          0 N01415  100uF  TC=0,0
5: R_RL          0 N01415  300ohm TC=0,0
6: X_D1          N01405 N01415 awb1n4148 PARAMS:
7: X_D2          N01463 N01415 awb1n4148 PARAMS:
8: V_V1          N01405 0
9: +SIN 0 3.8 1000 0 0 0
```

Figure 10: Netlist

## Discussion:

When comparing the three activities done in this project it can be seen that various different results arise. When its came to designing and theoretical calculations values were found which would make the circuit deliver a perfect output of 10mA and 3V. Moving onto the simulation part, the design worked well, it was able to match the conditions that were set and showcased how theoretically speaking the design performs the way it's intended. Now when it came to physically implementing it some challenges arose. First the design was to use 50microfarads or above and well the only capacitor close to that was 100microfarads capacitors, thats also one of the reasons I used 100microfarads in my Pspice simulation too as I wanted to be able to compare the results of those two directly. Alongside that a 330ohm resistor had to be used which was different than the intended 300ohms. This played a major role in the discrepancies between the simulation and the Waveforms data as the current from the waveforms data was actually quite a bit off from the expected 10mA and that's simply because to solve for current the voltage had to be divided by the 330ohm resistor which was used. So, it can be seen that the results from the simulation and physically implemented design can be different, of course many factors come into play when actually testing the circuit. However, the results were still adequate to be called a success.

Limitations come into play when the physical circuit had to be created, there was the limitation of parts and then there's the inaccuracy that can be caused from resistors, wires, voltage supply and the waveforms software itself as to measure the wave you need to pause the program and depending when you pause it there can be little differences in values. A problem arose when implementing the circuit and this came from the AD3. The calculated value for amplitude was 3.77V (rounded to 3.8) but when this was used on the waveforms software the voltage output was a bit lower than expected. However, when the amplitude was increased to 4V the expected output was reached. Now this is where the issues arise with the physical circuit as the components will have their own resistance and possible leakage from the breadboard connections. These factors likely caused the output voltage to be lower thus the input voltage had to be adjusted.

## References:

**[1]** Electronics Tutorials, "Full Wave Rectifier," *Electronics Tutorials*, Available: https://www.electronics-tutorials.ws/diode/diode_6.html. [Accessed: Feb. 7, 2025].

**[2]** A. S. Sedra, K. C. Smith, T. C. Carusone, and V. Gaudet, *Microelectronic Circuits*, 8th ed. New York, NY: Oxford University Press, 2019.

**[3]** Diodes Incorporated, "AP7331: 300mA Low Dropout Voltage Regulator," *Diodes Incorporated*, Available: https://www.diodes.com/assets/Datasheets/ds12019.pdf. [Accessed: Feb. 7, 2025].