Building an adjustable AC to DC converter

Ziyao Xiong

***Abstract*—In this project, my partner and I developed an AC to DC converter capable of converting a 5-volt, 1kHz AC signal into an adjustable DC signal ranging from 3.6V to 13.4V. Our circuit primarily comprises five parts: a push-pull amplifier that amplifies both the voltage and current of the signal from the oscilloscope to act as a 5V AC power source; a transformer that further steps up the voltage to 14.56V; a bridge consisting of four diodes, which performs full-wave rectification of the AC signal; a capacitor that smooths the voltage to an approximate DC signal; and finally, a linear regulator circuit that serves to reduce the ripple and adjust the output voltage to any desired value within the specified range.**

I. Introduction

ELECTRONIC devices typically utilize direct current (DC) for charging, as the battery chemistry necessitates a stable voltage to function safely and effectively. However, alternating current (AC) is seen as a more practical solution for long-distance transmission due to its ability to be easily stepped up or down in voltage using transformers [1]. The mains electrical power supply in the UK provides AC signals at 230 volts [2], which requires the use of an AC-to-DC converter in chargers. They are ubiquitous in modern electronics, from small power supplies for consumer electronics to large industrial machinery.

The objective of this project is to build such a converter which converts a 5-volt AC signal with frequency 1 kHz to a 5-volt DC signal, with an output current of around 100 mA. To adopt the different charging requirements of electronic devices, we made the output voltage of the converter adjustable. This allows for a wider range of devices to be charged utilizing the same supply unit, which increases the circuit's versatility and usefulness.

II. Design and Theory

The signal generator housed within the oscilloscope can be used to provide an initial voltage to the circuit, which produces an AC signal with a maximum amplitude of 2.5V. Therefore, an amplifier is required to increase its output to 5V, enabling it to be utilized as the power source for the subsequent circuit. In reality, the source voltage is a hazardous 230V AC signal, and hence an amplifier is not included in the converter. However, a low-voltage source is employed to ensure a safe and controlled testing environment for evaluating the circuit's performance.

The transformer constitutes the second part of the circuit. It is utilized typically to step down the incoming mains voltage. In our project, we adopted an opposing approach by using the transformer to raise the voltage level. This was done with the intention of generating a broader range of final output voltages.

A full-wave rectifier and a high-capacity capacitor are used subsequently to convert the AC signal to an unstable positive DC signal with ripples. To address this issue, a linear regulator circuit is employed to stabilize the voltage and facilitate the adjustment of the output voltage in the range of 3.6V to 13.4V via the rotation of the potentiometer.

A. Push-pull amplifier and transformer

Since a transformer is constructed after the amplifier in the circuit, the change in magnetic flux in the secondary coil will result in a change in the magnetic flux of the primary coil, by the means of Faraday's law [3]. Therefore, the voltage and current in the primary coil are dependent on that of the secondary coil, which may be varying. Consequently, a feedback current resulting from the magnetic induction will be generated from the primary coil, and in turn affecting the input signal. The push-pull amplifier is therefore selected by us as it can set the voltage at its output constantly at 5V regardless the changes in the magnetic field. Figure 1 shows its circuit diagram.

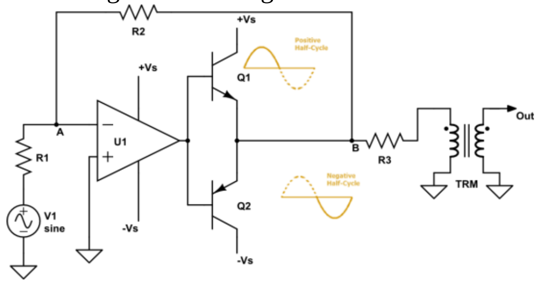

Fig 1. Configuration of a push-pull amplifier and a transformer. The output of the amplifier is obtained at point B. The inverting input of op-amp U1 is connected to an AC power supply V1 via a resistor R1 and the emitters of the NPN transistor Q1 and the PNP transistor Q2 via R2, while the non-inverting input of U1 is connected to the ground. U1 is also linked to a DC power supply, Vs. The base of Q1 and Q2 are connected to the output of U1. The collectors of Q1 and Q2 are linked to the positive and negative terminals of Vs respectively. A resistor R3 is situated between the transformer and the amplifier.

As the op-amp is modelled to have infinite input impedance, its inputs draw no current. With respect to Kirchoff's current law [4], at junction A, the sum of the current through R1 and R2 equals zero. Therefore, the relationship between the output voltage at B (V_B) and input voltage (V1) can be expressed as:

$$\frac{V1}{R1}+\frac{V_B}{R2}=0\ . \tag{1}$$

The gain of the circuit is therefore:

$$G=\frac{V_B}{V1}=\frac{-R2}{R1}\ . \tag{2}$$

Consequently, the output voltage can be amplified by a factor equivalent to the ratio of R2 to R1.

Apart from the voltage, the current needs to be amplified as well. The LT1001 op-amp's current threshold is specified as 10mA [5]. However, this current limit may prove insufficient for the signal to traverse the transformer, as a portion of the power will be dissipated. Current amplification is done the push-pull stage consisting of two transistors, one NPN Q1 and one PNP Q2. Q1 is turned on during the positive half cycle of the AC signal, while Q2 is turned on during the negative half cycle, as their emitters are biased in opposite directions. Inside the transistors, small changes in the base current can result in larger changes in the collector and emitter current. By alternating the conduction of the transistors during each half cycle of the signal, the push-pull amplifier effectively amplifies the audio signal without producing any significant distortion [6]. This signal will be used as the AC power source for the following circuit.

The resistor R3 is used to restrict the current flowing through the transistors to a manageable level provided there is an accidental short-circuit of the amplifier output.

The transformer consists of two coils of wire, known as the primary and secondary coils, wrapped around a magnetic core. When an AC voltage Vp is applied to the primary coil, a magnetic field is created, the magnetic flux lines traverse the core and the secondary coil, thereby inducing a voltage within it. The magnitude of the voltage ε in the coils is contingent upon:

$$\varepsilon=-N\frac{d\emptyset}{dt}\ , \tag{3}$$

where $\frac{d\emptyset}{dt}$ represents the rate of change of magnetic flux and N represents the number of turns of the coil. The negative sign indicates the induced emf always opposing the direction of change of magnetic flux, according to Lenz's law [7].

The cross-sectional area of the coils on both windings is equivalent, as well as the magnetic field strength, resulting in the same value of $\frac{d\emptyset}{dt}$. The magnitude of the induced voltage thus only depends on the number of turns of coils, by means of:

$$\frac{Vs}{Vp} = \frac{Ns}{Np}, \tag{4}$$

where Vp and Vs are the voltage of the primary coil and secondary coil respectively; Np and Ns are the turns in the primary coil and secondary coil respectively.

B. Full-wave rectifier

The intended output is a positive DC signal, however, what passes through the transformer is AC with both positive and negative components. To rectify this, a bridge configuration comprised of four diodes is implemented to allow current to flow in a manner that ensures the output voltage is consistently positive with respect to the ground, irrespective of the input voltage polarity. Diode arrangement is depicted in Figure 2.

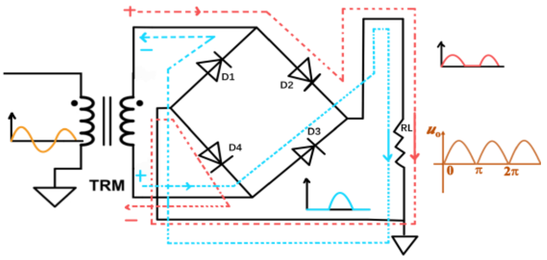

Fig 2. The bridge consists of four diodes D1, D2, D3, and D4. During the positive half-cycle of the AC signal, D2 and D4 in the bridge that are oriented in the forward direction conduct, while the other two diodes remain off. As a result, the current flows through the load resistor RL in the same direction as the input voltage, producing a positive half-cycle in the output waveform. During the negative half-cycle of the AC signal, D1 and D3 that were previously off now conduct, this causes the current to flow through RL in the opposite direction of the input voltage, so the negative half-cycle is rectified and added to the positive half-cycle in the output waveform.

Since the voltage across each diode is approximately 0.7v, there is a total voltage drop of 1.4v. Thus, U_0, the amplitude of the rectified waveform can be found by:

$$U_0 = Vs - 1.4. \tag{5}$$

C. Smoothing capacitor

The installation of a high-capacity capacitor in parallel with a load resistor can generate an approximate DC output, as its effect is to smooth out the variations in the voltage waveform. The voltage pattern across the capacitor is shown in Figure 3.

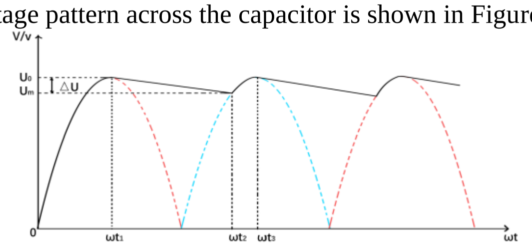

Fig 3. Assuming an initially uncharged capacitor, when the circuit is activated at t=0, the diode becomes forward-biased and the capacitor starts to charge where its output voltage equals the source voltage. After t_1, the capacitor discharges into the load resistor as the source voltage decreases ($\omega t_1=\pi/2$). The source voltage falls below the output voltage, causing the diode to become reverse-biased, thereby disconnecting the load from the source [8]. The output voltage decays exponentially with an RC time constant from t_1 to t_2. When the source voltage increases in the next cycle, the diode becomes forward-biased again and the output voltage matches the source voltage (t_2 to t_3). The difference between the peak voltage U_0 and the minimum voltage U_m that the capacitor discharges to is the ripple voltage ΔU.

As shown in Figure 3, the output voltage exhibits a positive sinusoidal waveform only when one of the paired diodes is in a state of conduction. In converse, the voltage decays exponentially over time. The output voltage U_c is given by:

$$U_c = \begin{cases} |U_0 \sin(\omega t)| & one\ paired\ diode\ on (t_0 - t_1) \\ U_0 e^{\frac{-(\omega t - \frac{\pi}{2})}{\omega RC}} & diodes\ off (t_1 \sim t_2) \end{cases}, \tag{6}$$

where ω is the angular frequency of the source voltage, R is the resistance of the load resistor, and C is the capacitance of the capacitor. When the value of RC is increased, the rate of decay decreases, resulting in the voltage decaying less amount within a given time frame. Additionally, as the frequency of the source voltage increases, the distance between wave peaks decreases, resulting in less time available for the capacitor to discharge. Thus, the ripple voltage ΔU is inversely proportional to the value of RC and the source voltage frequency f, such that:

$$\Delta U = \frac{U_0}{2fRC}. \tag{7}$$

D. Linear voltage regulator with adjustable voltage

In the final phase of our design, we incorporated a circuit to minimize ripple and enable adjustment of the output voltage. The rationale behind this decision was the observation that in order to attain a precise output voltage across the capacitor, it is imperative to adjust Vs to a corresponding specific value. However, controlling Vs is challenging as it is contingent on the efficiency of the transformer. The inclusion of the adjustable circuit guarantees the production of the desired output voltage regardless of the value of Vs. Furthermore, a reduced level of ripple results in an output voltage that more closely resembles a DC current, thereby expanding the applicability of the converter.

We therefore decided to build a linear voltage regulator circuit to stabilize the output voltage, where a Zener diode is involved as an essential part. Unlike a regular diode, it conducts current in the reverse direction when a breakdown voltage V_{BE} is applied to it, where the electric field across its depletion region increases, leading to the generation of electron-hole pairs by the impact ionization process [9]. This causes the diode to conduct current in the reverse direction, allowing the voltage across it to remain constant at V_{BE}, regardless of the current. This makes the Zener diode useful as a voltage regulator as it can maintain a constant voltage level even when the current or input voltage varies. The configuration of the circuit is shown in Figure 4.

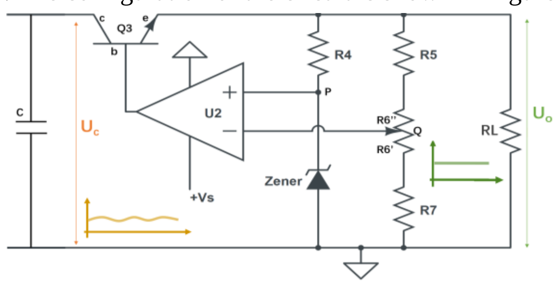

Fig 4. Unregulated voltage Uc from the filter capacitor is applied to the circuit, where the output Uo is a stabler DC voltage with an adjustable range. The op-amp U2 has its non-inverting input connected to a Zener diode that is reverse-biased and series-connected with a protective resistor R4. U2's inverting input is connected to a potentiometer R6, which is in series with resistors R5 and R7. U2's output is connected to the base of transistor Q3, whose emitter is integrated with R4, R5, and a load RL. A positive DC source Vs supplies power to U2.

The op-amp U2 has its two inputs at approximately the same level, thus the voltage at Q should be equal to the Zener potential at P, V_{BE}. This is done by the feedback loop, where U2 amplifies the difference in its input voltages and the output is fed back to the inverting input through the transistor Q3. Since R5, R6, and R7 are connected in series, according to Kirchoff's voltage law [4], the voltage at Q can be calculated by:

$$V_Q = \frac{R6' + R7}{R5 + R6 + R7} \cdot Uo = V_{BE}, \tag{8}$$

where R6 denotes the entire range of the potentiometer, and R6' signifies the resistance of its section that links to R7. The transistor acts as a variable resistor that can be controlled by the feedback circuit [10]. Since V_Q decreases as the output voltage Uo goes down, its increased voltage difference with V_{BE} will be amplified via the op-amp. This raises the voltage at base of Q3, pushing the transistor to allow more current to flow through its emitter, which increases Uo. Conversely, if Uo is too high, the transistor limits the current through its emitter and lowers the output. Thus, any tendencies toward fluctuations in Uo will be counteracted by inverse changes, resulting in a stabilized output.

By means of equation (10), the range of output that the circuit can generate can be computed, with R6' varying from 0 to R6:

$$Uo_{min} = (1 + \frac{R5}{R6+R7}) \cdot V_{BE} \ , \tag{9}$$

$$Uo_{max} = (1 + \frac{R5+R6}{R7}) \cdot V_{BE} \ . \tag{10}$$

It is important to note that the voltage output Uo cannot surpass the input voltage Uc. This is since the transistor and load are connected in series, and hence the voltage across the load and the transistor's collector-emitter are summed up to the input Uc. Achieving proper functionality of the circuit needs forward-biasing of the transistor, causing a positive voltage across it. Therefore, to constrain Uo below Uc, it may be necessary to restrict R6' from reaching its minimum value of 0.

III. Simulation and Circuit Building

Consolidating all the previously mentioned designs into one circuit, we constructed a circuit in LTspice and simulated it utilizing spice models of the components employed in the laboratory. Its configuration can be observed in Figure 5.

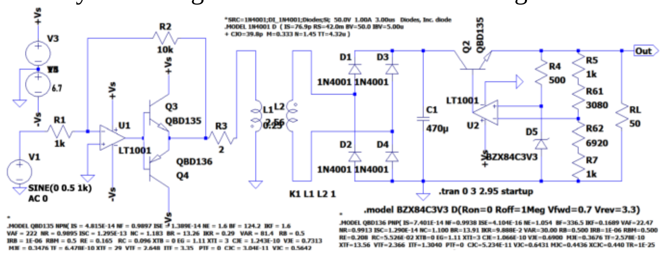

Fig 5. The circuit to be simulated in LTspice. Spice models for the LT1001 op-amp [5], 1N4001 diode [11], BD135 NPN transistor [12], BD 136 PNP transistor [13], and BZX84C3V3 Zener diode are imported [14]. In practical R6' and R6'' are replaced by a potentiometer. The value of R6' in this particular simulation is selected such that the output is at 5V and 100mA.

Upon successful simulation of the circuit in LTspice, which generates an output of 5V and 100mA as shown in Figure 6a, we proceeded to the physical construction of the circuit. The photo of our final circuit is illustrated in Figure 6b.

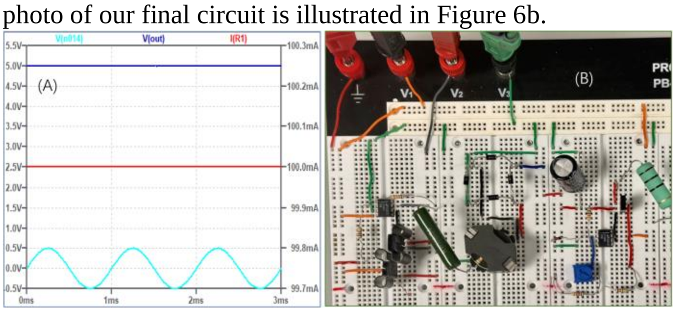

Fig 6a. The outcomes of the LTspice simulation. The cyan trace represents the input 0.5-volt AC signal from the oscilloscope, the blue trace represents the output 5-volt DC signal, and the red trace represents the current flowing through the load resistor. Hence, it can be inferred that the circuit will operate effectively. Fig 6b. The photograph of the constructed circuit, wherein the green wire serves as the grounding connection. The red and black wire functions as the connection to the positive and negative terminals of the AC power source respectively. The orange and grey wires respectively serve as the connections to the positive and negative terminals of a 6.7-volt DC power supply.

We paid meticulous attention to the color coding throughout the circuit construction to facilitate a clear identification of wires. We also maintained a tidy circuit layout to prevent the onset of parasitic capacitance and parasitic inductance of wires. The coupling of electric and magnetic fields between wires situated in close proximity can give rise to unintended inductance and capacitance, which can impede circuit performance and lead to delays, noise, and other disruptive phenomena [15].

A. *Push-pull amplifier*

As previously indicated, the objective is to achieve a voltage output amplitude of 5V from the amplifier. To minimize the energy waste and prevent excessive power dissipation in the amplifier's feedback loop, its current is typically restricted to ~0.1mA [5]. Hence, we selected R1 and R2 with resistance of 1kΩ and 10kΩ respectively. The convenience of selecting them with a 10-fold difference lies in the simplification of calculations, which means the gain can be easily calculated from equation (2) to be -10. Therefore, we generated a 1kHz, 0.5V sinusoidal wave from the oscilloscope and expected to obtain a 5V signal as simulated, showing in Figure 7a. The LT1001 is specified over a range of DC supply from ±3V to ±18V, where its maximum output is equal to the voltage supplied to it [5]. It is worth mentioning that the transistors generate a 0.7V voltage drop, therefore, to guarantee a 5V output, magnitude of Vs should be at least 5.7v. The actual outcome of waveforms when Vs<5.7V and Vs>5.7V is shown in Figure 7b and Figure 7c.

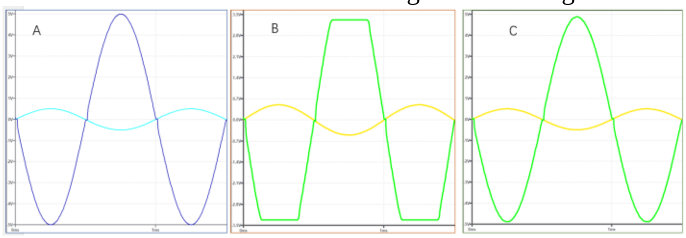

Figure 7a. The simulated pattern in LTspice obtained by setting the values of R1 and R2 at 1kΩ and 10kΩ respectively. The cyan trace corresponds to the input 0.5v signal, while the blue trace represents the amplified output signal of 10V. As expected, the output signal is flipped due to the negative gain. Figure 7b. The waveforms captured by the oscilloscope with an input voltage of 4V. The yellow trace depicts the input signal, and the green trace depicts the output signal, which is capped to approximately 3.3V. Figure 7c. The waveforms captured by the oscilloscope with an input voltage of 6.7V. The output signal is entirely displayed with an amplitude of 4.95V.

The measured output value is 0.05V lower than the expected value. This discrepancy may be attributed to either the 5% tolerance in resistance of the resistors used [16], or the fact that the input resistance of the op-amp is not ideally infinite [5].

The protective resistor R3 was connected in series with the amplifier and the transformer. To ensure that the transformer receives a significant portion of the voltage, the resistor should possess small resistance. Furthermore, since the power-supply current flows through the resistor, it is imperative to select a resistor with a sufficiently high power-rating. As such, we have opted for a 2Ω resistor with a power rating of 8w.

B. *Transformer and Bridge*

In order to achieve a relatively large output voltage range, a step-up transformation of approximately 3 times the voltage from the amplifier was deemed necessary. This necessitates the secondary coil of the transformer to have three times the number of turns in the primary coil, as per equation (4). However, due to some of the magnetic flux generated by the primary winding failing to link with the secondary winding and instead leaking into the surrounding air or non-magnetic materials, the transformer's efficiency is reduced. This phenomenon is referred to as flux leakage and is difficult to determine its size with precision. Our laboratory utilized an RM8 transformer with a toroidal ferrite core having a relative permeability of 2500 to air [17], leading us to assume that the level of flux leakage would be relatively small. To compensate, we wound the secondary and primary coils by 50 and 160 turns, which was a slightly higher ratio than 3:1. The relatively large number of turns helped reduce flux leakage. The transformer was represented by two inductors corresponding to the primary and secondary coils in LTspice. Since the inductance of a coil is directly proportional to square of its number of turns [18], the simulated coils' inductance ratio should be 25:256. The simulated output voltage, following the passage of the signal through the bridge and transformer, is depicted in Figure 8a, with the transformer's efficiency modelled to be 100%. There is a 1.4V voltage drop due to the bridge, and the output exhibits an amplitude of 14.6V, which is consistent as calculated using equations (4) and (5).

During the winding process, we made two attempts. Initially, the secondary coil was wound directly on top of the primary coil without any insulating methods. However, this resulted in a transformer with a significantly reduced efficiency, generating only a voltage of 9.42V as shown in Figure 8b. The distorted shape of the waveform indicates the presence of large parasitic capacitance and inductance in coils, negatively impacting the transformer's functionality. In the second attempt, we wound the secondary winding over the primary winding on the core former, with an insulating layer of tape separating them. Another layer of primary winding was then added on top, again with insulation. The core former was carefully clamped with the ferrite core, and insulating washers were placed between them [17]. The resulting waveform after passing the transformer and bridge measured by the oscilloscope is presented in Figure 8c, with the output voltage amplitude measured to be 14.56V, indicating the transformer has a remarkably high efficiency of 99.7%.

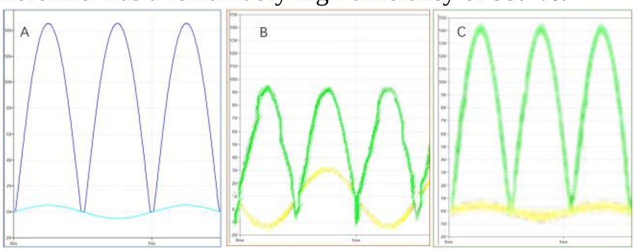

Fig 8a. Simulation outcomes when efficiency of transformer is 100%. The blue trace illustrates the waveform after the bridge, where the negative part is rectified, and the positive part is in phase with the 0.5V signal input (cyan trace). This outcome is attributed to the negative gain of both the transformer and amplifier, resulting in an overall positive gain. Fig 8b. The waveform captured by an oscilloscope when the first winding method was employed. The output signal (green trace) after passing through the bridge is not a perfect rectified sinusoidal wave. The amplitude of the output signal is 9.42V. Note that the input channel (yellow trace) has a different scale compared to the output channel, with an actual input of 0.5V. Fig 8c. The output waveform recorded by the oscilloscope displays an amplitude of 14.56V with almost no shape distortion, where the input and output channels have the same scale.

C. *Capacitor and linear regulator circuit*

Based on Equation (7), an increase in capacitance of the capacitor leads to smaller ripple. Therefore, we chose a 470μF capacitor instead of the 220μF and 22μF capacitors available in the laboratory. To achieve an output voltage of 5V and a load current of 100mA, a load resistor of 50Ω was selected. Consequently, the ripple in the absence of the linear regulator was calculated to be 309mV, a result consistent with our simulation using LTspice, where the ripple was found to be 294mV (Figure 9c). In practice, the measured ripple was 480mV (Figure 9a), where its deviation from the theoretical value may be attributed to the limitations of the spice models in accurately describing component behavior and the impact of noise.

Upon passing the signal through the linear regulator circuit, a significant reduction in the ripple was observed. The oscilloscope displayed a nearly straight line, with ripple of 24mV detected upon closer inspection (Figure 9b). However, LTspice did not simulate any ripples, even when the minimum division of 1nV was reached. This discrepancy is due to the oversimplification of LTspice. For instance, the voltage across the Zener diode may vary slightly as the current changes, and the input impedance of the op-amp is not infinitely high. These factors affect the performance of the voltage regulator, nevertheless, compared to a circuit that solely relies on a capacitor for smoothing, the output is significantly better.

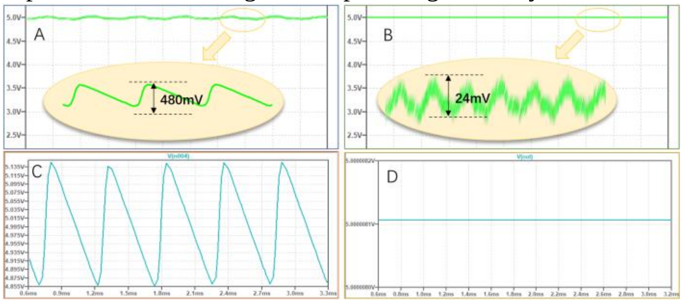

Fig 9a/9b. Output recorded without and with the linear regulator connected in the circuit respectively. The ripple in the former case was measured to be 480mV while it reduced to 24mV in the latter. A magnified image of the ripple waveform when the scaled was zoomed in is also presented. Fig 9c/9d. LTspice simulation performed without and with the connection linear regulator circuit respectively. The output ranges from 4.843V to 5.137V when the regulator circuit is not connected, accompanied by a ripple of 294mV, whereas the ripple is simulated to be smaller than 1nV when it is connected.

Equations (9) and (10) reveal that the size of the Zener diode has an impact on the range of output that can be produced by the circuit. A smaller breakdown voltage results in a smaller minimum output. Therefore, we opted to use a Zener diode with a breakdown voltage of 3.3V, rather than 10V or 12V, which were also available in the lab. The BZX84C3V3 Zener diode selected by us has a peak working current of 250mA [13]. To ensure that this current would not be exceeded, we chose R4 with a resistance of 500Ω. The potentiometer we used can provide resistance ranging from 0 to 10kΩ. We selected R5 and R7 with a resistance of 1kΩ, enabling a theoretical output range of 3.60V to 14.56V, as per equations (11) and (12). We were able to generate the lower bound output of 3.6V successfully, however, the maximum output we obtained in practical was 13.4V, possibly due to power dissipation and voltage drop across the pass transistor. Hence, our final DC output range is 3.6V to 13.4V, with current ranges from 72mA to 268mA.

IV. CONCLUSION

In conclusion, our project was successful in fulfilling its objective of converting a 1kHz, 5V AC signal into a 5V, 100mA DC signal. On top of that, our circuit was also able to generate an adjustable output DC voltage ranging from 3.6V to 13.4V, with a remarkably low ripple of only 24mV. We generally followed three steps to keep the project organized. Firstly, we calculated the theoretical parameters required for the circuit and selected appropriate components. Next, we used LTspice to simulate the circuit and ensure its functionality. Finally, we constructed the physical circuit based on simulation results.

While LTspice provided valuable insights into the circuit's behavior, it is essential to acknowledge that its simulations do not always reflect practical scenarios accurately, such as noise, shape distortion, parasitic capacitance and inductance, tolerance of parameters, and inefficient energy transfer. Furthermore, we observed that the simulation results were not always consistent, as multiple attempts yielded varying ripple voltage values. Even with identical circuits, my partner and I obtained slightly different outcomes. As such, we recognized the need to consider practical solutions and to conduct a more in-depth analysis of the circuit's behavior, rather than relying solely on the software.

References

[1] Grigsby, L. L., et al. The Electric Power Engineering Handbook. USA: CRC Press. (2001). ISBN 0-8493-8578-4

[2] UK energy security - GOV.UK. Accessed April 28, 2023. https://www.gov.uk/business/uk-energy-security.

[3] Clerk Maxwell, James (1904). *A Treatise on Electricity and Magnetism*. Vol. 2 (3rd ed.). Oxford University Press. pp. 178–179, 189.

[4] Oldham, Kalil T. Swain (2008). *The doctrine of description: Gustav Kirchhoff, classical physics, and the "purpose of all science" in 19th-century Germany* (Ph. D.). University of California, Berkeley. p. 52. Docket 3331743.

[5] "LT1001 - Precision Operational Amplifier - Analog Devices." Linear Tec*hnology. Accessed April 27, 2023.* https://www.analog.com/media/en/technical-documentation/data-sheets/LT1001.pdf.

[6] "The Push-Pull Voltage-Amplifier Stage." Audio Power Amplifier Design, 2013, 201–30. https://doi.org/10.4324/9780240526140-8.

[7] Lenz, E. (1834), "Ueber die Bestimmung der Richtung der durch elektodynamische Vertheilung erregten galvanischen Ströme", Annalen der Physik und Chemie, 107 (31), pp. 483–494. A partial translation of the paper is available in Magie, W. M. (1963), A Source Book in Physics, Harvard: Cambridge MA, pp. 511–513.

[8] " 4.3 Controlled Full-Wave Rectifiers - Courses.minia.edu.eg." minia.edu. eg. University of Minia. Accessed April 27, 2023. https://courses.minia.edu.eg/Attach/9921Lecture04.pdf. "

[9] Saxon, Wolfgang (July 6, 1993). "Clarence M. Zener, 87, Physicist And Professor at Carnegie Mellon". The New York Times.

[10] Gaetano Palumbo; Salvatore Pennisi (2002). Feedback Amplifiers: Theory and Design. Springer Science & Business Media. ISBN 9780792376439.

[11] "Datasheet for 1N4001-E3/4 Vishay Rectifier Diodes." Octopart. Accessed April 27, 2023. https://octopart.com/datasheet/1n4001-e3%2F4-vishay-39644907?utm_source=bing&utm_medium= cpc & amp;utm_campaign=b_cpc_intl_search_dsa_english_en_usd_datasheets&utm_term=datasheet&utm_content=Intl+Datasheet+DSA.

[12] "BD135 - Plastic Medium-Power Silicon NPN Transistors." Octopart. Accessed April 27, 2023. https://www.electrokit.com/uploads/productfile/40320/BD135-D.PDF.

[13] "Datasheet for BD136 Stmicroelectronics Bjts." Octopart. Accessed April 27, 2023. https://octopart.com/datasheet/bd136-stmicroelectronics-40841680?utm_source=bing&utm_medium = cpc&utm_campaign=b_cpc_intl_search_dsa_english_en_usd_datasheets&utm_term=datasheet&utm_content=Intl+Datasheet+DSA.

[14] "BZX84-c3v3 - Voltage Regulator Diodes." Nexperia. Accessed April 27, 2023. https://www.nexperia.com/products/diodes/zener-diodes/BZX84-C3V3.html.

[15] Alley, Charles L.; Atwood, Kenneth W. (1973). Electronic Engineering, 3rd Ed. New York: John Wiley & Sons. p. 199. ISBN 0-471-02450-3.

[16] "4 Band Resistor Color Code Calculator." Resistor Color Code Calculator - 4 band, 5 band, 6 band | DigiKey Electronics. Accessed April 28, 2023. https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-resistor-color-code.

[17] "Ferrites and Accessories - TDK Electronics AG." tdk-electronics. Accessed April 28, 2023. https://www.tdk-electronics.tdk.com/inf/80/db/fer/rm_8.pdf.

[18] Fritz Langford-Smith, editor (1953). Radiotron Designer's Handbook, 4th Edition, Amalgamated Wireless Valve Company Pty., Ltd. Chapter 10, "Calculation of Inductance" (pp. 429–448)

Acknowledgement

I would like to acknowledge my lab partner Kevin Ni for his contributions and collaboration throughout this project. The results presented in this report were taken together with him, and his insights and efforts were instrumental in the completion of this project.