

Analysis of Electrical and Electronic Measurements: Simulations, Theoretical Calculations, and
Validation Using Proteus Software

OWEN MILLER

BAE SYSTEMS, UNIVERSITY OF WALES TRINITY ST DAVID

09/04/2023

Abstract

This report explores the utilization of Proteus simulation software for performing electrical and electronic measurements. Two simulations were conducted to investigate maximum power transfer and resonant frequency determination in circuits. The simulations were accompanied by theoretical calculations, and the results were compared to validate the accuracy of the simulations.

The report emphasizes the importance of integrating theoretical calculations with simulations to ensure accurate and reliable results. Simulations offer a practical approach to observe circuit behaviours, while theoretical calculations provide analytical insights and precise predictions. The combination of these methods enables a comprehensive understanding of circuit characteristics and performance.

Standard electrical laws and formulae were applied to perform calculations and analyse the relationships between voltage, current, resistance, and power in the circuits. The theoretical calculations and simulations facilitated the examination of circuit behaviour, power transfer efficiency, and resonant frequency.

Advantages of using Proteus simulation software, including its user-friendly interface, extensive component library, and visualization capabilities, were discussed. Proteus allowed for the simulation and analysis of complex circuit configurations, observation of voltage waveforms, and collection of valuable data for analysis.

Discrepancies between simulation results and theoretical calculations were acknowledged, attributing them to factors such as communication errors, assumptions, rounding, calculation errors, and simulation limitations.

Recommendations for future studies were made to minimize these errors, including improved communication, accurate data collection, detailed component models, advanced simulation methodologies, and thorough verification and error analysis.

In conclusion, the integration of theoretical calculations and Proteus simulations proved to be a valuable approach for analysing electrical and electronic circuits. This combination provided insights into circuit behaviour, power transfer efficiency, and resonant frequency determination. Understanding the strengths and limitations of simulations and theoretical calculations enables informed decision-making in circuit design, optimization, and analysis, advancing the field of electrical and electronic measurements.

I. Contents

Abstract	2
1. Introduction.....	5
2. Background information	6
3. Procedure	8
4. Results circuit 1.....	13
5. Results Circuit 2.....	17
6. Analytical improvements and further study areas	23
7. Conclusion	25
8. Bibliography	26
9. Appendix.....	27

1. Introduction

The objective of this report was to investigate the performance of electrical and electronic circuits through the use of the Proteus simulation software.

In the realm of electrical and electronic engineering, precise measurements are crucial for understanding and optimizing circuit performance. However, conducting real-world experiments to obtain accurate data can be time-consuming, costly, and sometimes impractical. To address these challenges, simulation software like Proteus provides engineers and researchers with a valuable tool to perform virtual experiments, record data, and analyse circuit behaviour.

Proteus, a widely utilized simulation software in the field, offers a comprehensive range of features and capabilities for accurate circuit modelling and simulation. Leveraging Proteus, researchers can investigate circuit behaviour under different conditions, visualize results, and make informed design decisions.

Throughout this report, the practical application of Proteus in conducting these simulations, recording data, and analysing the results will be explored. The simulations were designed to provide insights into obtaining precise measurements and validate the accuracy of the simulation software by comparing the results with theoretical calculations.

By employing Proteus, engineers and researchers gain valuable insights into circuit behaviour, optimize designs, and predict system performance without the need for extensive physical experimentation. This report aims to showcase the differences between theoretical calculation and simulated data.

2. Background information

Electrical and electronic measurements play a vital role in circuit analysis and design [1]. The combination of simulations and theoretical calculations provides a comprehensive approach to validate and verify the accuracy of the results obtained.

1. Fundamental Electrical Laws and Standard Formulae

Several fundamental electrical laws govern circuit behaviour and are essential for accurate analysis. Ohm's Law, for instance, establishes a relationship between voltage (V), current (I), and resistance (R) through the equation $V = I * R$ [2]. This law allows for the calculation of voltage, current, or resistance in a circuit, depending on the known variables.

Kirchhoff's Laws, including Kirchhoff's Current Law (KCL) and Kirchhoff's Voltage Law (KVL), are key principles in analysing complex circuits. KCL states that the sum of currents entering a node is equal to the sum of currents leaving that node. KVL, states that the sum of voltage drops in a closed loop is equal to the sum of voltage rises [3].

By applying these fundamental laws, engineers can calculate circuit parameters using standard formulae. Ohm's Law, for example, enables the determination of voltage, current, or resistance when two values are known [4].

Power calculations involve different formulas such as $P = V * I$ or $P = I^2 * R$.

The resonant frequency of a circuit, can be calculated using the formula $f = 1 / (2 * \pi * \sqrt{L * C})$.

2. Importance of Simulations and Theoretical Calculations:

Simulations allow for the evaluation of circuit performance under different conditions, identification of potential issues, and optimization of designs [5]. However, simulations have limitations, including model inaccuracies and numerical approximations.

To enhance confidence in simulation results, theoretical calculations play a crucial role. Theoretical calculations offer a theoretical perspective that enables verification of simulation accuracy. Ensuring accurate measurements and reinforcing the analysis process [6].

3. Utilizing Proteus Simulation Software (advantages):

- a. **Comprehensive Functionality:** Proteus offers a wide range of simulation capabilities, facilitating the analysis and evaluation of circuits with varying complexities [7].
- b. **Extensive Component Library:** Proteus provides an extensive component library that allows accurate modelling of real-world systems. This library ensures fidelity in simulations and supports precise analysis [8]
- c. **User-Friendly Interface:** Proteus features an intuitive and user-friendly interface that simplifies circuit design, behaviour simulation, and result analysis. Its ease of use allows researchers to focus on their study objectives without being hindered by complex software operations.
- d. **Industry Acceptance:** Proteus has gained widespread acceptance in both industry and academia. Its compatibility with industry-standard file formats and its active user community contribute to its credibility and usability [7]

3. Procedure

The procedure for any circuit builds within proteus follows the same format with variations to the specific components applied to the circuit, this format is as follows and examples shown are from simulation 1:

1. Create a new project by selecting "File" from the menu bar and choosing "New Project."

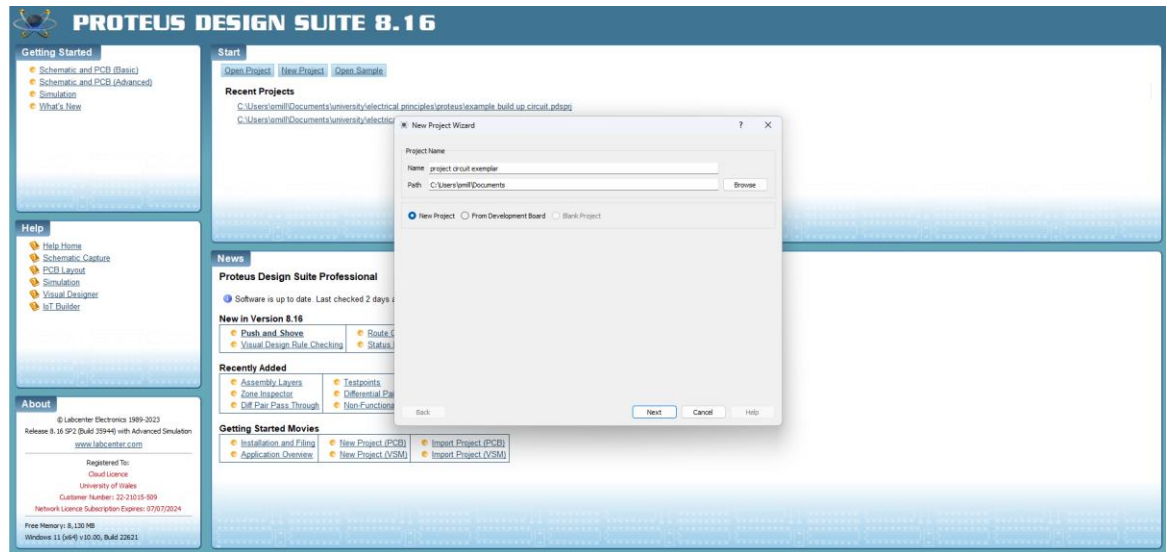


figure 1 – saving new project

2. Assign a suitable name and location for the project and save it. [as shown in Figure 1]

3. Access the "Schematic Capture" tab in the Proteus workspace.

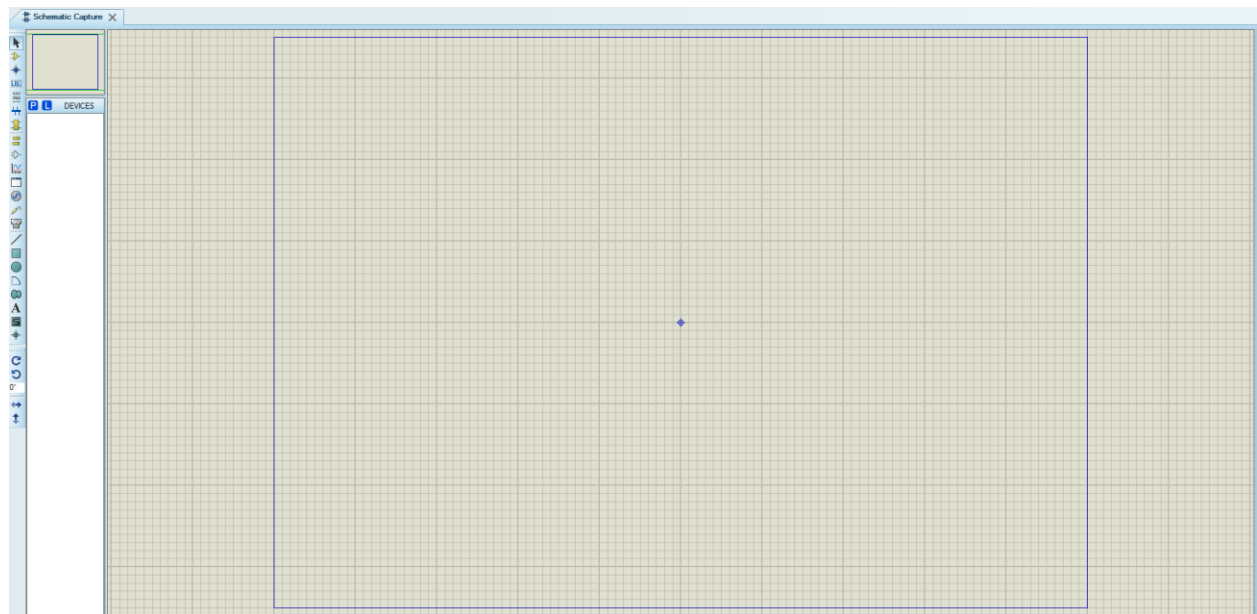


Figure 2 – Schematic capture

4. Select the required components from the component library by clicking on the "P" button or navigating through the "Library" menu. Drag and drop the components onto the workspace.

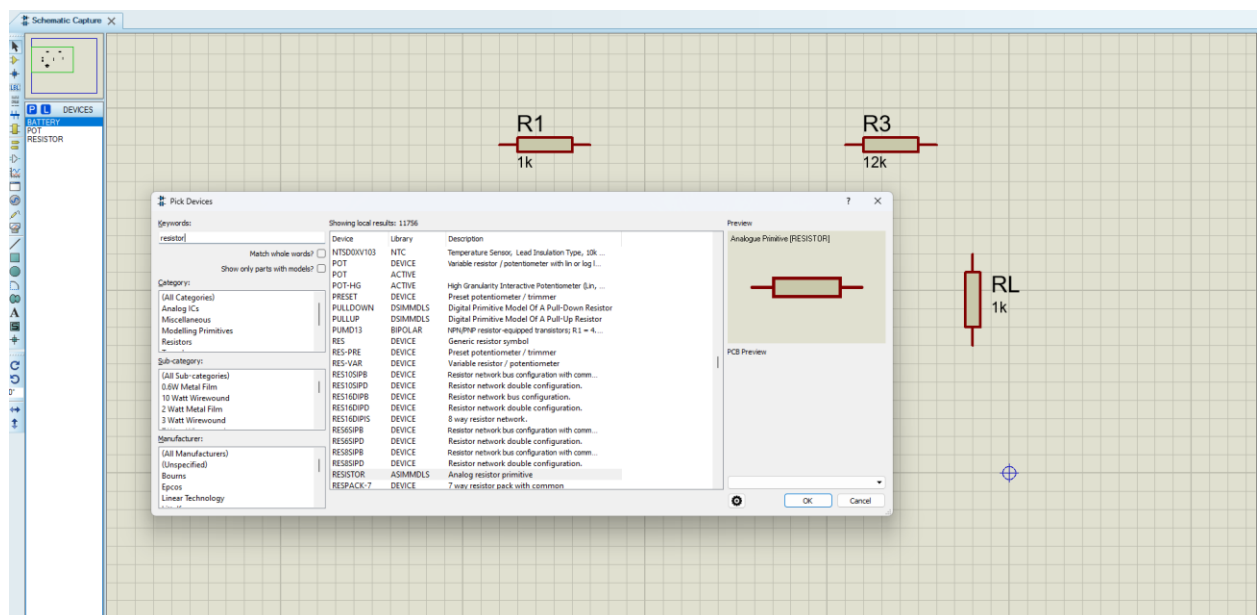


Figure 3 – component selection

5. Establish connections between the components by placing wires between their connection points. Use the "Wire" button in the toolbar to add wires.

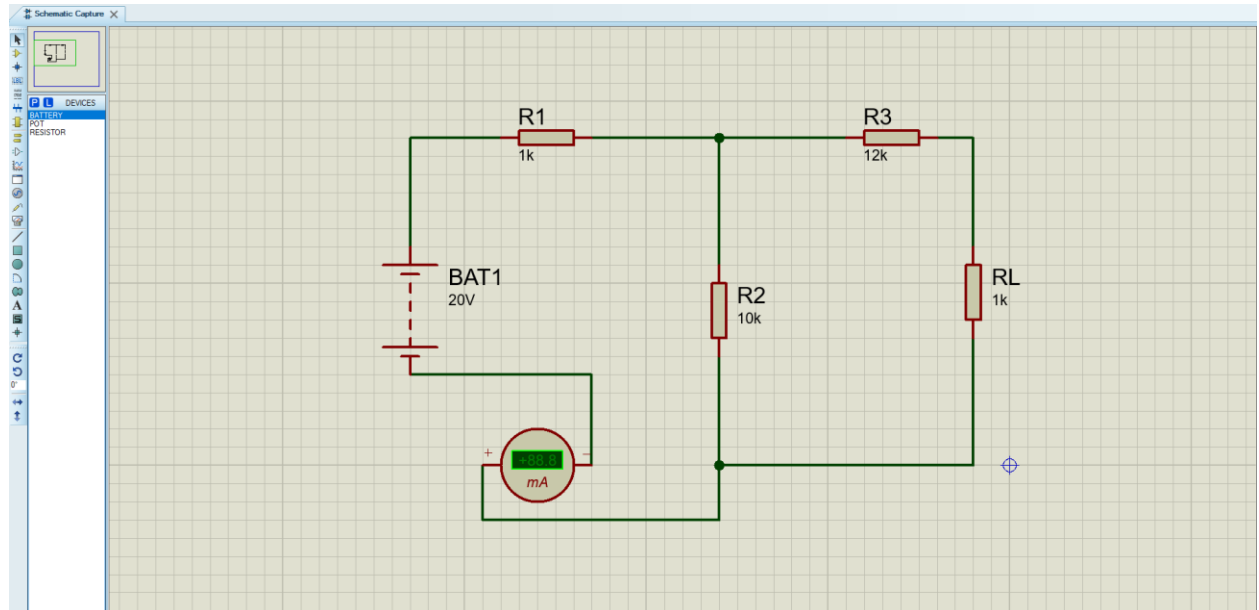


Figure 4 – connecting circuit

6. Configure the properties of each component, such as resistance values or voltage sources. Double-click on a component to open its properties window and make the necessary adjustments.

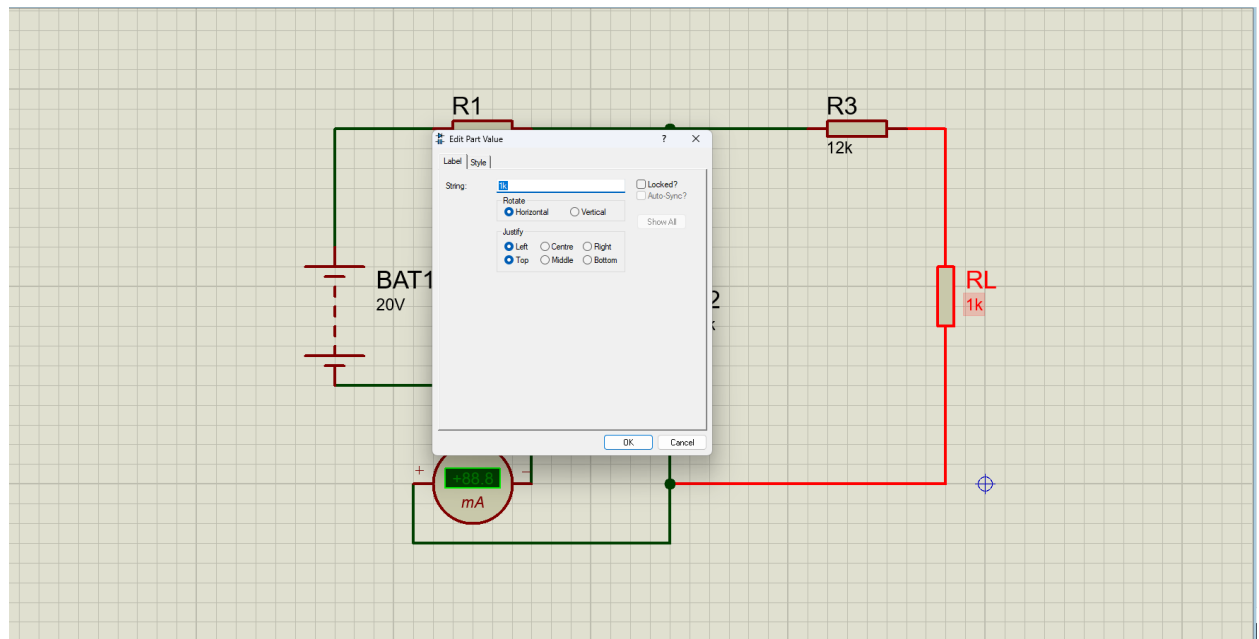


Figure 5 – configuration of components

7. Verify the connections and ensure there are no errors or incomplete connections in the circuit.
8. Save the schematic by selecting "File" from the menu bar and choosing "Save" or using the keyboard shortcut Ctrl + S.

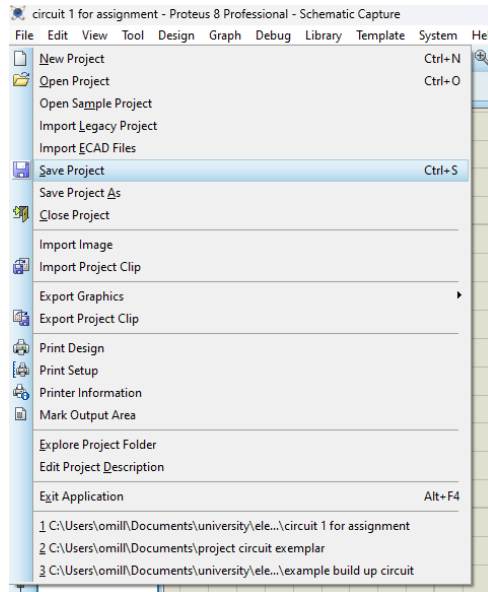


Figure 6 – saving schematic

9. Launch the simulation environment by clicking on the "Run" button in the toolbar or selecting "Simulate" from the menu bar and clicking "Run Simulation."

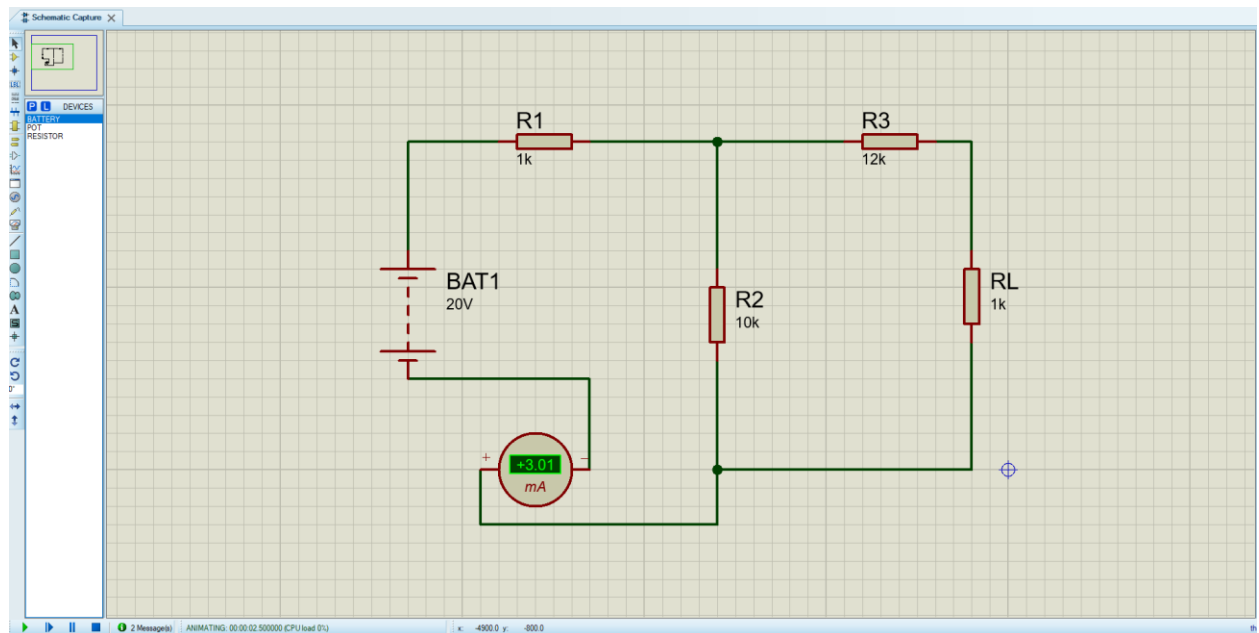


Figure 7 – simulation 1 running

10. Observe the circuit's behaviour and analyse the results presented in the simulation window. Examine voltage and current waveforms and other relevant parameters. (if applicable)
11. Modify the circuit design, component values, or simulation settings if necessary to further investigate the circuit's behaviour.
12. Save the simulation project by selecting "File" from the menu bar and choosing "Save Project." (as per figure 6)
13. Close the simulation environment and return to the Proteus workspace.

NOTE: this is a basic example and may have differences with more complex schematic environments

4. Results circuit 1

4.1. Simulation 1

In the first simulation, a variable resistor (R_L) was utilized with its resistance incrementally adjusted from 1 k Ω to 20 k Ω in steps of 1 k Ω , providing a range of resistance values for analysis. The power transfer in the circuit was then calculated using these resistance values.

The circuit configuration included a fixed supplied voltage of 20 V. Additionally, three more resistors with fixed values were present in the circuit (figure 8). The purpose of this simulation was to examine how the power transfer varied with different resistance levels.

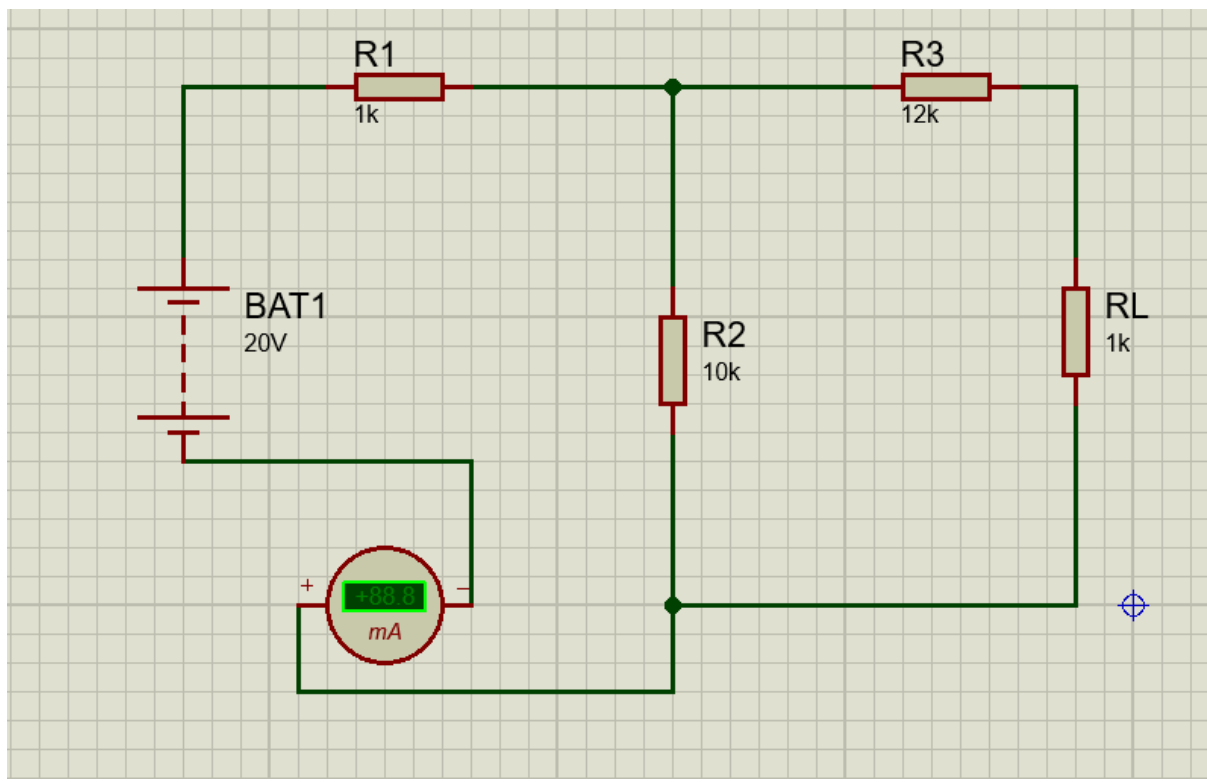


Figure 8 – circuit 1, complete

- 4.1.1. The data collected from all simulations was compiled into a table, summarizing the key parameters for each simulation.

Voltage (v)	load resistance (kΩ)	Load current (mA)	power (mW)
20	1	3.01	9.0601
20	2	2.93	17.1698
20	3	2.86	24.5388
20	4	2.8	31.36
20	5	2.74	37.538
20	6	2.69	43.4166
20	7	2.65	49.1575
20	8	2.61	54.4968
20	9	2.57	59.4441
20	10	2.54	64.516
20	11	2.51	69.3011
20	12	2.48	73.8048
20	13	2.46	78.6708
20	14	2.43	82.6686
20	15	2.41	87.1215
20	16	2.39	91.3936
20	17	2.37	95.4873
20	18	2.35	99.405
20	19	2.34	104.0364
20	20	2.32	107.648

Figure 9 – table of figures circuit 1

The load current values were measured using an ammeter during the simulations. By measuring the load current, the corresponding power values were calculated using the formula $P = I^2 * R$, where I represents the load current and R represents the load resistance.

This table provides a concise overview of the collected data, including the voltage, load resistance, load current, and power values for each simulation.

4.1.2. The values from the table were utilized to create a graph illustrating the relationship between load resistance and power.

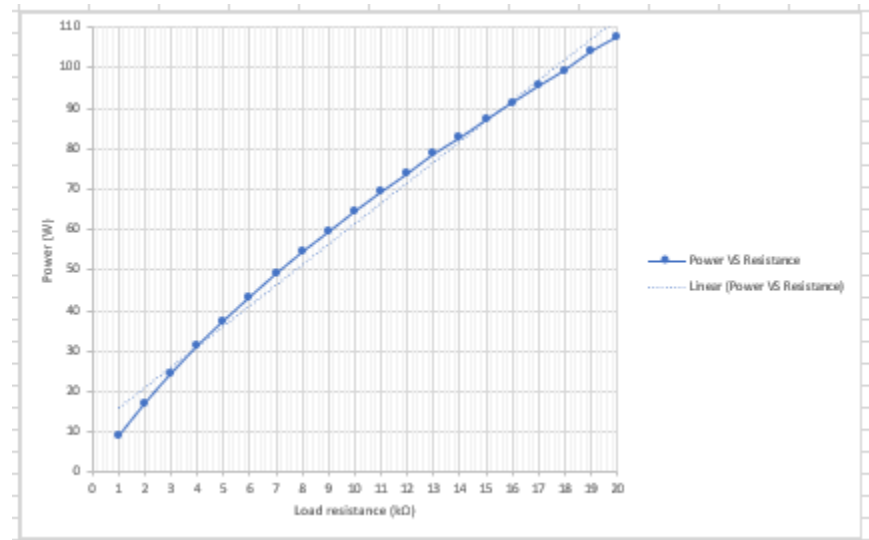


Figure 10 – resistance vs power

The graph showcases a positive correlation between load resistance and power, indicating that increasing the resistance led to a corresponding increase in power transfer. However, the rate of increase gradually diminished as the resistance continued to rise. This observation can be attributed to the relationship between power, current, and resistance.

According to the formula $P = I^2 * R$, it is evident that power is directly proportional to the square of the current and resistance. When the resistance is increased, the current flowing through the circuit decreases due to the fixed voltage supply. As a result, the power gain becomes less significant since it is influenced by the square of the current.

4.2. Theoretical calculation values

Calculating total resistance of circuit assuming $R_L = 1k\Omega$

Given:

$$R_3 = 12k\Omega \text{ (12,000 ohms)}$$

$$R_L = 1k\Omega \text{ (1,000 ohms)}$$

$$R_2 = 10k\Omega \text{ (10,000 ohms)}$$

$$R_1 = 1k\Omega \text{ (1,000 ohms)}$$

Total resistance (R_T) of R_3 and R_L in series:

$$R_T = R_3 + R_L$$

$$R_T = 12k\Omega + 1k\Omega$$

$$R_T = 13k\Omega \text{ (13,000 ohms)}$$

Equivalent resistance (R_{Eq}) of R_T and R_2 in parallel:

$$1 / R_{Eq} = 1 / R_T + 1 / R_2$$

$$1 / R_{Eq} = 1 / 13k\Omega + 1 / 10k\Omega$$

$$1 / R_{Eq} = (10k\Omega + 13k\Omega) / (13k\Omega * 10k\Omega)$$

$$1 / R_{Eq} = 23k\Omega / (130k\Omega * 10k\Omega)$$

$$1 / R_{Eq} = 23k\Omega / (1.3M\Omega)$$

$$1 / R_{Eq} = 0.0176923$$

$$R_{Eq} \approx 56.54\Omega \text{ (rounded to two decimal places)}$$

Total resistance (R_T) of R_{Eq} and R_1 in series:

$$R_{Total} = R_{Eq} + R_1$$

$$R_{Total} \approx 56.54\Omega + 1k\Omega$$

$$R_{Total} \approx 1056.54\Omega \text{ (rounded to two decimal places)}$$

Therefore, the total resistance is approximately 1056.54 Ω .

Calculating power through RL assuming $R_L = 1k\Omega$

Given:

Applied voltage: 20V

Load resistor (R_L): $1k\Omega$ (1,000 ohms)

Calculation of Load Current:

The load current (I_L) through R_L can be determined using Ohm's Law: $I_L = V / R_L$, where V is the applied voltage and R_L is the load resistor.

Substituting the given values, $I_L = 20V / 1k\Omega$.

$I_L = 0.02$ A or 20 mA.

Calculation of Power Dissipation:

The power dissipated by the load resistor can be calculated using the formula $P = I * V$, where P is the power, I is the load current, and V is the applied voltage.

Substituting the given values, $P = 0.02$ A * 20V.

$P = 0.4$ W or 400 mW.

Therefore, the load current through R_L , with an applied voltage of 20V, is 20 mA. The power dissipated by the load resistor is 0.4 W or 400 mW.

5. Results Circuit 2

5.1. Simulation 2:

The second circuit configuration included several components: a signal generator, a resistor (100 ohms), an inductor (5 mH), a capacitor (0.1 uF), and an oscilloscope. The signal generator was utilized to control the voltage, while the frequency was varied to determine the peak response of the circuit.

The fixed voltage supplied by the signal generator was set to 10V, while the frequency was adjusted to investigate the circuit's behaviour. By varying the frequency, the resonant frequency of the circuit could be determined, which corresponds to the frequency at which the circuit exhibits maximum response or impedance.

To measure and analyse the waveforms generated by the circuit, an oscilloscope was employed. This enabled the observation of amplitude, frequency, and phase characteristics, aiding in the determination of the resonant frequency at different circuit points [9].

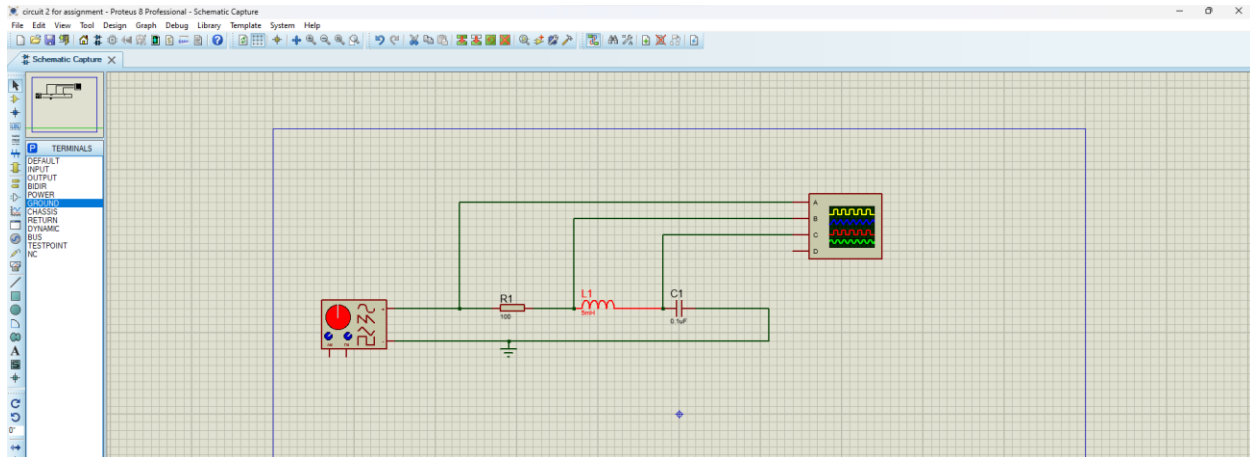


Figure 11 – circuit 2 build

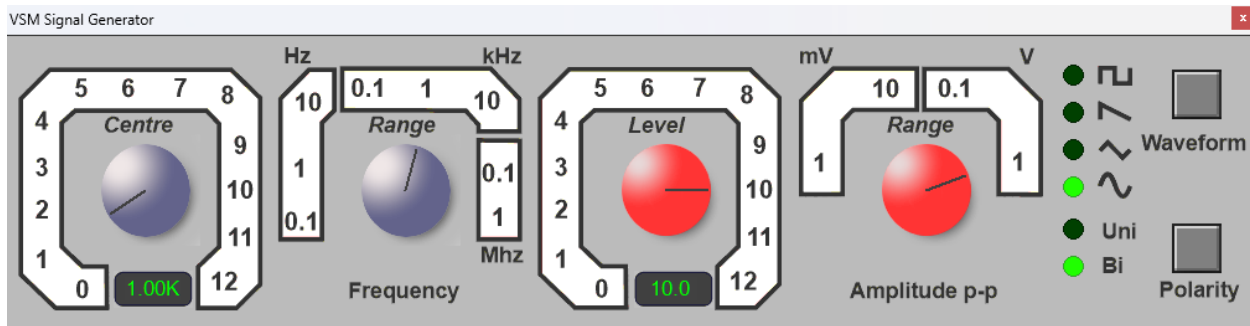


Figure 12 – oscilloscope initial settings (fixed voltage)

Specifically, the oscilloscope captured the peaks and lows of three different voltage waveforms: VR (voltage, resistor), VL (voltage, inductor), and VC (voltage, capacitor).

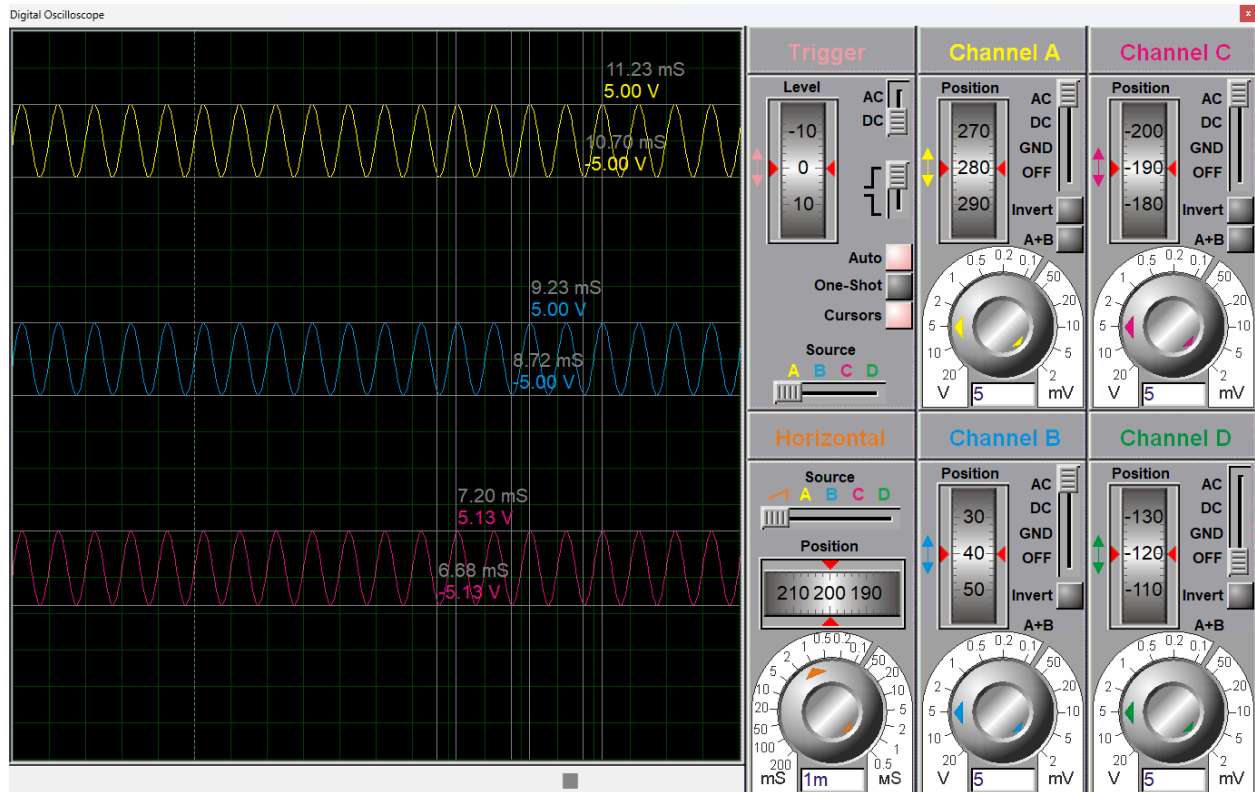


Figure 13 – reading the oscilloscope.

The data obtained from varying the frequency from 1 kHz to 20 kHz was organized and applied to a table to summarize the results. The table includes the following information:

frequency Hz	VR (V)	VL (V)	VC (V)
1000	10	10	10.26
2000	10	10	10.76
3000	10	9.76	12
4000	10	9.26	13.76
5000	10	8.26	17
6000	10	5.76	21.26
6300	10	4.26	22.16
6500	10	3.26	22.76
6800	10	1.5	22.76
7000	10	0.25	22
8000	10	5	16.76
9000	10	7.5	11.5
10000	10	8.5	8.26
11000	10	9	6.26
12000	10	9.26	4.76
13000	10	9.5	4
14000	10	9.5	3.26
15000	10	9.76	2.76
16000	10	9.76	2.26
17000	10	9.76	2
18000	10	9.76	1.75
19000	10	9.76	1.5
20000	10	9.76	1.5

Figure 14 – table of data from oscilloscope

The data obtained from the table was plotted on a semi-log graph to visualize the resonant frequency of the circuit.

The x-axis of the graph represents the frequency (kHz), ranging from 1 kHz to 20 kHz. The y-axis represents the peak voltages (V) observed across the resistor (VR), inductor (VL), and capacitor (VC).

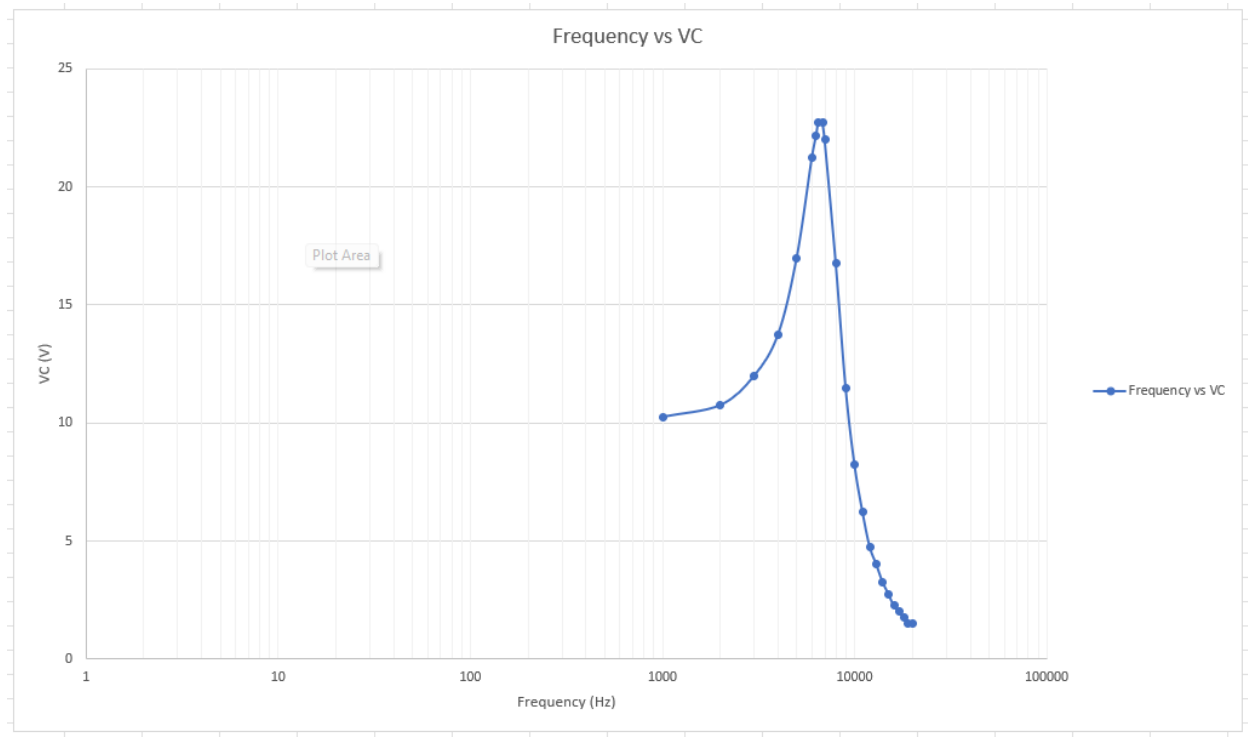


Figure 15 – frequency vs VC semi-log graph

The graph of the data reveals that the resonant frequency of the circuit is approximately 6650 Hz, where the peak response is observed. The voltage across the capacitor (VC) reaches a value of approximately 22.76 V.

5.2. Theoretical calculations:

Resonant Frequency Calculation:

To determine the resonant frequency of the series RLC circuit under investigation, the following parameters were considered:

Resistance (R): 100 ohms

Inductance (L): 5 mH (5×10^{-3} H)

Capacitance (C): 0.1 uF (0.1×10^{-6} F)

Using the formula for resonant frequency in a series RLC circuit:

$$f = 1 / (2\pi\sqrt{LC})$$

Substituting the values, we obtain:

$$f = 1 / (2\pi\sqrt{(5 \times 10^{-3}) \times (0.1 \times 10^{-6})})$$

After simplification, we have:

$$f \approx 1 / (2\pi\sqrt{5 \times 10^{-9}})$$

Calculating the square root and evaluating the reciprocal, we find:

$$f \approx 1 / (2\pi \times 2.236 \times 10^{-4})$$

$$f \approx 1 / (1.408 \times 10^{-3})$$

Hence, the resonant frequency of the series RLC circuit is approximately 7099 Hz.

5.3. Comparison between simulation and theoretical results

Theoretical Resonant Frequency: 7099 Hz

Simulated Resonant Frequency (Peak): 6650 Hz

Based on the comparison, it can be observed that the theoretical value of 7099 Hz and the simulated peak value of 6650 Hz demonstrate a reasonably close agreement. The slight discrepancy between the theoretical and simulated values may be attributed to factors shown in **6**.

It is important to consider these discrepancies to refine the theoretical model and improve the accuracy of future predictions in similar series RLC circuits.

6. Analytical improvements and further study areas

6.1. When comparing the simulation results to the theoretical calculations for both circuits, it is essential to consider several factors that could contribute to the differences observed:

- 6.1.1. Communication Error and Interpretation: Misinterpretation of the initial questions or any communication errors during the data collection process could introduce discrepancies between the intended values and the values used in the simulation or calculations. It is important to ensure clear communication and understanding of the parameters and assumptions made.
- 6.1.2. Assumptions and Rounding: Theoretical calculations often involve assumptions and idealized models. Any assumptions made during the calculations, such as neglecting certain factors or approximating component behaviour, can introduce variations from the simulation results. Additionally, rounding errors during calculations can accumulate and contribute to differences in the final values.
- 6.1.3. Calculation Errors (Human Error): Human error is always a possibility during the calculation process. Mistakes in inputting values, performing calculations, or transferring

data can lead to discrepancies between the theoretical calculations and the simulation results. Double-checking calculations and data entry can help minimize these errors [10]

- 6.1.4. Simulation Error: Simulations are based on mathematical models that approximate real-world behaviour. The accuracy of the simulation results depends on the accuracy of the models used and the assumptions made within the simulation software. In some cases, certain real-world effects or component behaviours may not be accurately captured in the simulation, leading to deviations from the theoretical calculations.

The issues stated above can be expressly seen through circuit one where the two sections have very large discrepancies from a combination of these.

6.2. Future study areas:

- 6.2.1. Refining Communication and Data Collection: To mitigate communication errors, it is crucial to establish clear and precise communication channels between researchers and simulation software users. Clearly defining the parameters, assumptions, and values used in simulations can help minimize misinterpretation. Additionally, implementing robust data collection processes, double-checking inputs, and ensuring accurate data transfer can enhance the reliability of simulation results [11].
- 6.2.2. Detailed Component Models: Enhancing the accuracy of simulation models by using detailed and accurate component models can reduce discrepancies between simulations and theoretical calculations. Incorporating component models that consider real-world variations, tolerances, and non-ideal characteristics can provide more realistic results. This includes considering variations in component values, temperature effects, and frequency-dependent characteristics [12]

7. Conclusion

In conclusion, this report highlighted the use of Proteus simulation software for performing electrical and electronic measurements. Two simulations were conducted to explore maximum power transfer and resonant frequency determination in circuits. The simulations were compared with theoretical calculations to validate their accuracy. Integrating theoretical calculations with simulations proved crucial in ensuring reliable results. The simulations provided practical insights into circuit behaviour, while theoretical calculations offered analytical predictions. Combining both approaches enhanced our understanding of circuit characteristics and performance.

Throughout the report, standard electrical laws and formulae were employed to analyse circuit parameters. Theoretical calculations and simulations facilitated the examination of voltage, current, resistance, and power relationships, enabling the evaluation of power transfer efficiency and resonant frequency.

Proteus simulation software offered several advantages, such as its user-friendly interface, extensive component library, and ability to visualize circuit responses. It allowed us to simulate complex circuit configurations, observe voltage waveforms, and gather valuable data.

However, discrepancies between simulation results and theoretical calculations should be considered. Possible factors contributing to these differences include communication errors, assumptions, rounding, calculation errors, and simulation limitations. Future studies should focus on minimizing these errors through improved communication, accurate data collection, detailed component models, advanced simulation methodologies, and thorough verification and error analysis.

In summary, combining theoretical calculations and Proteus simulations proved effective in analysing electrical and electronic circuits. The integration of these methods provided valuable insights into circuit behaviour, power transfer efficiency, and resonant frequency determination. Understanding the strengths and limitations of simulations and theoretical calculations enables informed decision-making in circuit design and analysis, driving progress in the field of electrical and electronic measurements.

8. Bibliography

- [1] R. E. Thomas and A. J. Rosa, *The Analysis and Design of Linear Circuits*, Toussaint: John Wiley & Sons, 2016.
- [2] K. M. Tenny KM, *Ohms Law.*, Treasure Island (FL): StatPearls Publishing, 2022.
- [3] L. Trombetta, *Experimental Verification of Kirchhoff's Voltage Law and Kirchhoff's Current Law*, 2013.
- [4] BYJU'S, "Ohm's Law," BYJU'S, [Online]. Available: <https://byjus.com/physics/ohms-law/>. [Accessed 29 06 2023].
- [5] K. K. J. & J. J. Lee, "Application of Proteus Simulation Software in Electric Circuit Education. *Journal of Electrical Engineering and Automation*," vol. 2, no. 2, pp. 23-32, 2021.
- [6] M. N. O. Sadiku, "Fundamentals of Electric Circuits," CRC Press, 2019.
- [7] M. & H. M. Z. Rahman, "Simulation and Implementation of Electrical Circuits Using Proteus," *International Journal of Scientific Research in Computer Science, Engineering and Information Technology*, vol. 4, no. 1, pp. 169-173, 2018.
- [8] K. M. M. L. & M. P. Dixit, "Analysis of Proteus for Simulation of Electrical Circuits.," *2020 International Conference on Computational Intelligence and Computing Research*, pp. 1-4, 2020.
- [9] Tektronix, "What is an oscilloscope?," Tektronix, [Online]. Available: <https://www.tek.com/en/blog/what-is-an-oscilloscope>. [Accessed 08 07 2023].

[10] Behavioral Research and Teaching team, “Mathematics, Error Analysis In,” University of Oregon, Oregon.

[11] “The importance of good communication in the workplace,” MichaelPage, [Online].

Available: <https://www.michaelpage.co.uk/advice/management-advice/development-and-retention/importance-good-communication-workplace>.

[Accessed 09 07 2023].

[12] “Component-Based Modeling,” mathworks, [Online]. Available:

<https://www.mathworks.com/help/simulink/model-components.html>. [Accessed 09 07 2023].

9. Appendix

WORD COUNT: $\approx$ [2440] not including, abstract, mathematical calculations, titles, bibliograph