RSF Conference Proceeding Series: Engineering and Technology, Vol. 3 No. 1 (2023)
https://doi.org/10.31098/cset.v3i1.743

Check for updates

Research Paper

Analysis of I-V on Basic Electrical Circuit Using Ni Multisim App in Apply Physics

Indah Nurhidayati[1], Aryanti Fitrianingsih[1], Sinta Marito Siagian[2], Samaria Chrisna HS[2]
[1]Politeknik Ilmu Pelayaran Semarang, Indonesia
[2]Politeknik Negeri Medan, Indonesia

Received: Aug 31, 2023	Revised: Sept 7, 2023	Accepted: Sept 21, 2023	Online: Oct 5, 2023

Abstract

Utilization of the NI Multisim application is useful for designing a circuit, starting with designing schematic circuits and analyzing and simulating. Increasing the use of technology so that it is very much needed in physics learning at the Polytechnic of Maritime Sciences on electrical circuit material. This study aims to measure and characterize I-V currents in an electrical circuit by comparison with laboratories and multisim applications. Research activities were carried out using the I-V Current and Voltage comparison analysis method using theory in the manual formula, practice with the circuit kit, and NI Multisim simulation by building a circuit scheme on circuit composition namely resistance, voltage source, and using voltage measuring instruments and current measuring instruments which are all regulated in such a way concerning the theory of electric circuits. The results of this study are that there is a difference in the total resistance value between the values in the circuit kit module practice with the use of Multisim. But the difference is very slight with an error of about 1% for series circuits, and about 4.4% for parallel circuits. Voltage values in series circuits using NI Multisim circuit kits and applications have differences with errors ranging from 0 to 3.3 %. The comparison of the voltage values in direct measurements of the circuit kit and the NI Multisim simulation has the lowest error of 0% and the highest of 20% at the 6V source in R2.

Keywords *electrical circuit, Ni-Multisim, current, voltage*

INTRODUCTION

Information Technology and Communication have become primary needs for all people. The utilization of Information and Communication Technology has an impact on processes and activities that can be carried out more quickly, easily, and efficiently (Roza, Ananda, Siregar, Nasution, & Yanie, 2021). Therefore the use of software must also be further developed, several types of which are proteus applications, NI Multisim, Live Wire, Electronic Workbench, etc. The multi-sim application is an application that can run specific programs on electrical circuits, and this application is produced by National Instruments Inc. In this application, you can easily observe the symptoms that occur in an electric circuit through simulations to carry out accurate and specific measurement tests (Ridwan & Kembuan, 2021). Increasing the use of technology so that it is very much needed in learning physics at the Polytechnic of Shipping Sciences on electrical circuit material. This course requires current and voltage analysis in a circuit in general such as series-parallel and the use of Khirchoff's law. understanding of learning theory is still carried out manually

Corresponding author's email: indah.nurhidayati@pip-semarang.ac.id; aryanti@pip-semarang.ac.id; sintasiagian@polmed.ac.id; samariachrisna@polmed.ac.id

and has never been implemented by utilizing the NI Multisim application for calculation and analysis efficiency. So that this application still needs to be developed again for each type of electrical circuit. Taking this into account, the authors are interested in analyzing theory and practice regarding the use of NI Multisim Application technology so this research is expected to provide the latest information for electrical circuit courses, especially the application of technology in analyzing electrical circuits.

LITERATURE REVIEW

NI Multisim (formerly known as MultiSIM) is the development of electronic circuit simulation (Electronics Workbench) into an application that can implement and simulate analog and digital electronic circuits. This tool can be used for modeling the parameter properties of digital and analog circuits. Apart from that, it can also be used to analyze the characteristics of a circuit using transients or AC/DC (Eliza, Asnil, & Husnaini, 2019). With this Multisim software, we can model the properties of analog and digital circuit parameters. The capabilities provided by Multisim are being able to model various circuit designs, test a circuit with various possible components, and examine the properties of the entire circuit by performing AC / DC or transient analysis.

Based on research conducted by (Surahmat & Fu'ady, 2020) regarding the use of the National Instrument Multisim and Ultiboard 11.0 Software. the use of this application is useful for designing a circuit, starting with designing a schematic circuit, analyzing, simulating, checking errors, and making layouts on the PCB until finishing ready to be sent to the manufacturer giving a contribution to influence student learning outcomes in analog and digital electronics circuits subjects after utilizing learning media with NI Multisim 11.0 software. Based on Research Research (Muhammad & Herman, 2019) examining the characteristics of diodes using multisim, the results show that multisim can work well in simulating electronic components such as diodes.

METHODOLOGY

This research was carried out using the comparative method of current and voltage I-V analysis practice with circuit kits and NI Multisim simulations. I-V analysis research on electrical circuits and Khirchoff's law is simulated by the NI Multisim application using a research model in the form of a circuit schematic formed based on circuit theory. From this scheme, a circuit kit is built which becomes a model to obtain current and voltage measurement data, and the circuit scheme is applied to NI Multisim to obtain current and voltage data. The research design applied to the NI Multisim application is shown in Figures 1 and 2 below.

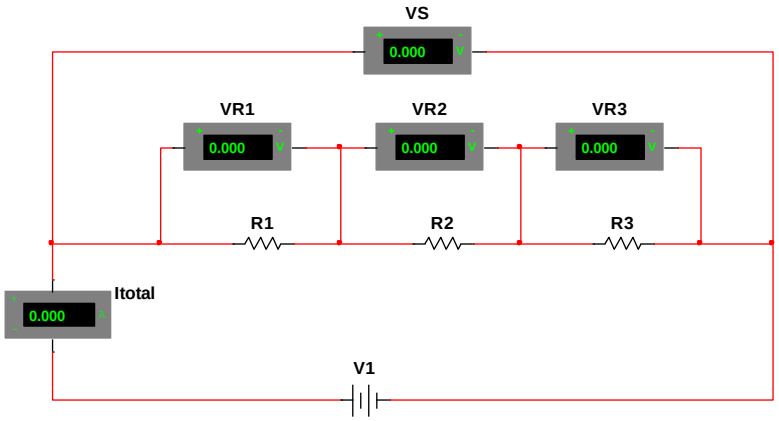


Figure 1. Series circuit scheme

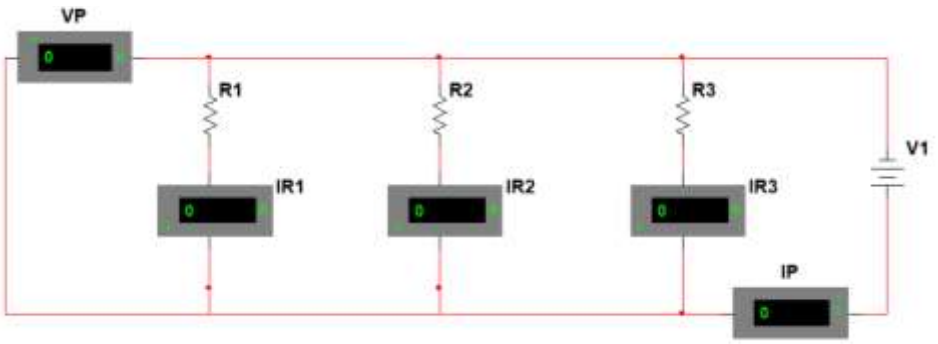
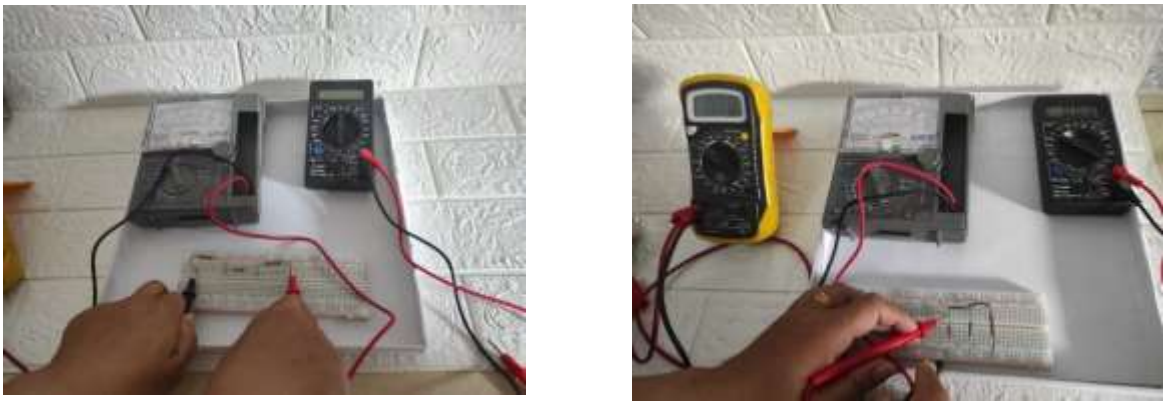


Figure 2. Parallel circuit schematic

FINDINGS AND DISCUSSION

Measurements and simulations of electrical circuits such as series circuits, and parallel circuits with a variety of voltage sources of 6 V, 9 V, and 12 V The measurements carried out on the circuit kit are current and voltage measurements as well as those carried out in the NI Multisim application simulation. The measurement and simulation process using the application can be seen in the following figure



(a) Series (b) Parallel

Figure 3. Resistance measurements in series and parallel

Measurement of the resistance value on the resistor value installed in the circuit kit shows different numbers on each measuring instrument. That is because the measurements are also carried out with different circuits, namely in series and parallel circuits. So that it can be calculated the value of resistance in each circuit in theory. Furthermore, the measurement of the resistance value was carried out using the NI Multisim application, which can be seen in the image below.

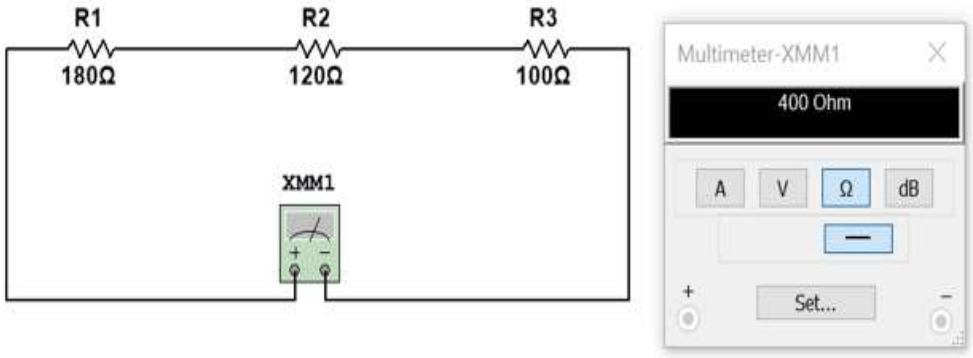


Figure 4. The value of the total resistance in series with Multisim

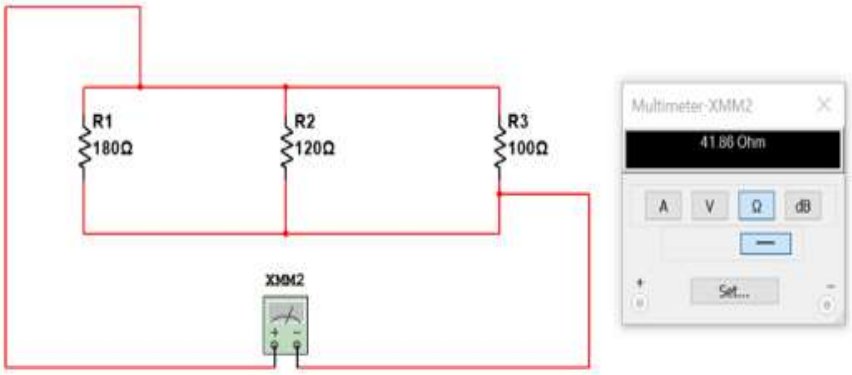


Figure 5. The value of the total resistance in a parallel circuit with Multisim

The results of the total resistance value in each electrical circuit also show values that vary based on the circuit. If we review the value of the total resistance in this application, it has a range of values that is quite close to the value in the circuit module practice. Resistance values for the circuit kit module and values for the NI Multisim application can be seen in detail in Table 1.

Table 1. Results of measurement of the value of total barriers

Type of Circuit	Total Resistance (Ω) Practice	Total Resistance (Ω) Theory (Multisim)	Error
Seri	395	400	1.25 %
Parallel	40	41.86	4.44 %

Based on the measurement results in Table 1, it can be seen there is a difference in the total resistance value between the values in the practice of the circuit kit module with the use of Multisim. But the difference is very slight with an error of about 1% for series circuits, and about 4.4% for parallel circuits. Penelitian ini juga telah dilaksanakan dengan melakukan pengukuran arus dan tegangan pada masing-masing rangkaian yang telah ditetapkan dengan memvariasikan nilai tegangan sumber yang diberikan yaitu pada 6V, 9V, 12V. The third variation was chosen by the researcher to observe the effect of the voltage on the current and the voltage on each resistance of the specified electrical circuit. The I-V analysis on the measurement of the NI Multisim circuit kit module and application can be seen in Figure 6 below.

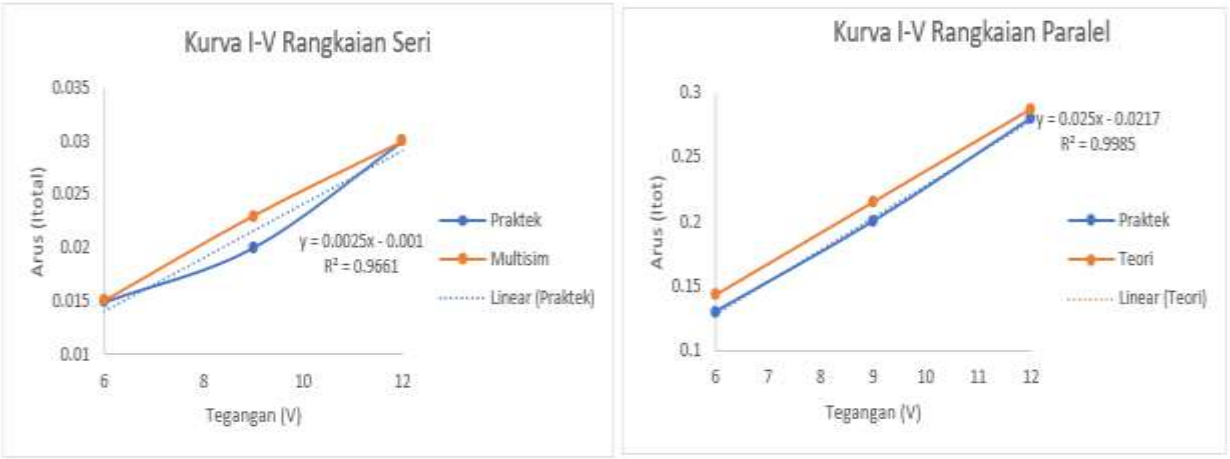


Figure 6. I-V curves in series and parallel circuits

In Figure 6 it can be seen that the current value in each circuit has linear characteristics both in

direct measurements with the circuit kit and by using the NI Multisim application. The higher the value of the source voltage given to each circuit, the resulting total current value also increases. (Saefullah et al., 2018). This happens by Ohm's law analysis, namely the value of electric current is proportional to the value of the applied voltage, where Ohm's law always discusses the relationship between voltage (V) and electric current (I).

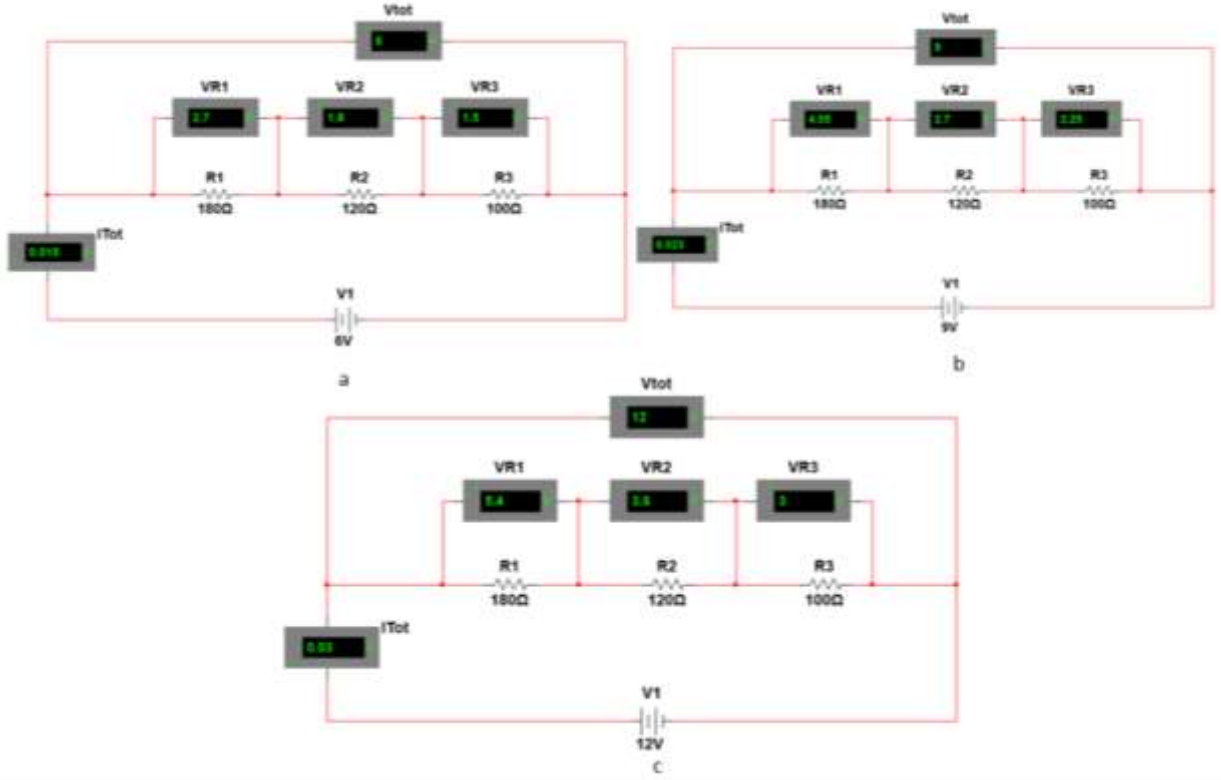


Figure 7. Series Circuit Voltage Value of Source Variation (a) 6V, (b) 9V, (c) 12V

Figure 7 is the result of measurements analyzed by simulation using the NI Multisim application by determining the voltage values that are connected in series to each resistance. Measurements of the voltage value with the NI Multisim circuit kit module and simulation have been carried out and the results are obtained in Table 2 below.

Table 2. The result of measuring the voltage value in a series circuit

V (in)	Resistance	Practice Voltage on R	Theoretical Voltage on R	Error
6V	R1	2.72	2.7	0.74 %
6V	R2	1.79	1.8	0.56 %
6V	R3	1.45	1.5	3.33 %
9V	R1	4.05	4.05	0.00 %
9V	R2	2.69	2.7	0.37 %
9V	R3	2.21	2.25	1.78 %
12V	R1	5.31	5.4	1.67 %
12V	R2	3.6	3.6	0.00 %
12V	R3	2.96	3	1.33 %

Based on table 2 it is known that the series circuit specified in the circuit kit has a resistance value of each, namely R1 = 180Ω, R2 = 120 Ω, R3 = 100 Ω. Based on the results of measuring the Voltage Value in a series circuit using the NI Multisim circuit kit and application, there is a difference with an error ranging from 0 to 3.3%. The different results between measurements and calculations are

caused by differences in viewing angles in reading the multimeter and the resistance values of different resistor materials or from rounding off numbers in calculations. Table 2 can also explain that with the decreasing value of resistance on resistors that are wired in series, the resulting voltage value both by measurement and NI Multisim simulation also decreases in value. This happens because the greater the resistivity of a conductor, the more difficult it is for electric current to pass through it.

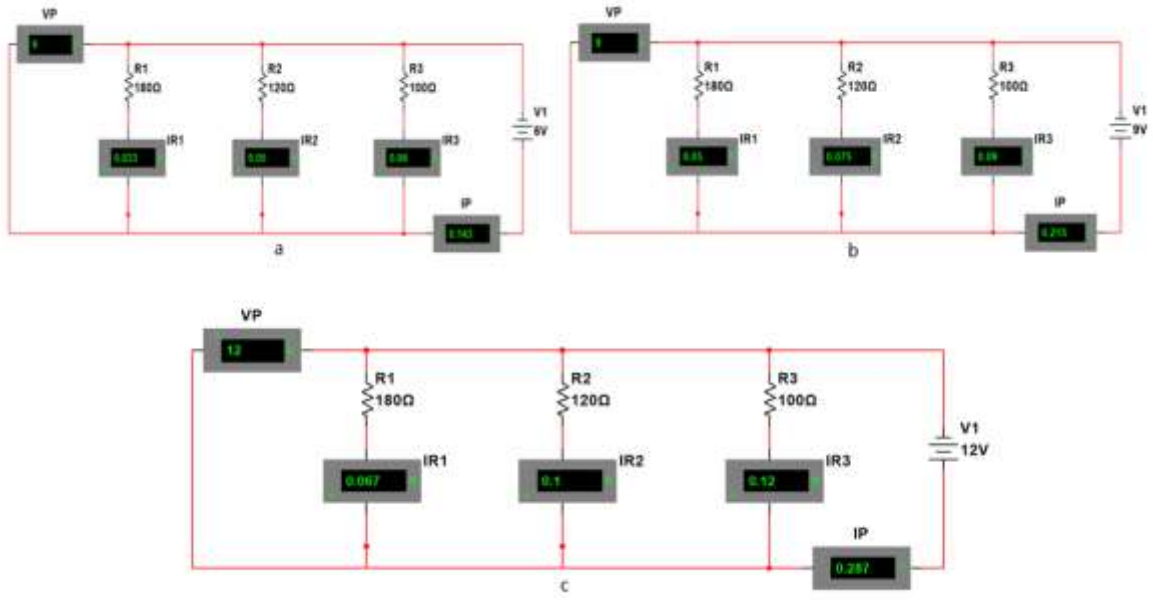


Figure 8. Current value of parallel circuit source variation (a) 6V, (b) 9V, (c) 12V

Data on measurement results for resistors connected in parallel and mixed can be seen in Table 3 below.

Table 3. Current Value Measurement Results in Parallel Circuits

V (in)	Resistance	Practice Current on R	Theoretical Current on R	Error
6V	R1	0.03	0.033	9.09 %
6V	R2	0.04	0.05	20.00 %
6V	R3	0.056	0.06	6.67 %
9V	R1	0.05	0.05	0.00 %
9V	R2	0.07	0.075	6.67 %
9V	R3	0.08	0.09	11.11 %
12V	R1	0.06	0.067	10.45 %
12V	R2	0.09	0.1	10.00 %
12V	R3	0.12	0.12	0.00 %

In Table 3 it is known that the electric current tends to increase with the addition of the value of the source voltage. In addition, the smaller the resistance value of a resistor that is arranged in parallel, the greater the resulting current value, both in terms of direct measurements and from the NI Multisim simulation. This happens because the voltage value is directly proportional to the electric current value in theory and practice. As for the voltage values in parallel circuits, in principle, it is the same for each resistor that is assembled, which is around 6V, 9V, and 12V. This voltage value will be the same on each resistor even though the value of each resistor has a different amount.

CONCLUSION AND FURTHER RESEARCH

From this study, it can be concluded that there is a difference in the total resistance value between the values in the circuit kit module practice with the use of Multisim. But the difference is very slight with an error of about 1% for series circuits, and about 4.4% for parallel circuits. Then, the current value in each circuit has linear characteristics both in direct measurements with the circuit kit and by using the NI Multisim application. The higher the value of the source voltage given to each circuit, the resulting total current value also increases with errors ranging from 0 to 3.3%. The comparison of the voltage values in direct measurements of the circuit kit and the NI Multisim simulation has the lowest error of 0% and the highest of 20% at the 6V source in R2. The current value for each resistance mixed can be seen that the practical and theoretical values have an error value of 0% to 24.24%.

REFERENCES

Eliza, F., Asnil, A., & Husnaini, I. (2019). Peningkatan Kompetensi Guru SMK Melalui Pelatihan Software engineering. *Jtev (jurnal teknik elektro dan vokasional), 5*(2), 67-74.

Muhammad, R. D., & Herman, H. (2019). Characteristic Test Of Transistor Based Multisim Software. *PROtek: Jurnal Ilmiah Teknik Elektro, 6*(2), 63-67.

Ridwan, R., & Kembuan, D. R. E. (2021). Efektivitas Penggunaan Simulasi dengan Multisim Berbantuan Virtual Laboratory untuk Meningkatkan Kemampuan Berpikir Kritis Mahasiswa Jurusan Pendidikan Teknik Elektro. *Jurnal Kiprah, 9*(1), 39-47.

Roza, I., Ananda, Y., Siregar, L. A., Nasution, A. A., & Yanie, A. (2021). Pelatihan Aplikasi Perangkat Lunak Multisim Pada Laboratorium Jurusan Kelistrikan SMK Swa Bina Karya. *Jurnal TUNAS, 2*(2), 26-30.

Saefullah, A., Fakhturrokhman, M., Oktarisa, Y., Arsy, R. D., Rosdiana, H., Gustiono, V., & Indriyanto, S. (2018). Rancang Bangun Alat Praktikum Hukum Ohm Untuk Memfasilitasi Kemampuan Berfikir Tingkat Tinggi (Higher Order Thinking Skills). *Gravity: Jurnal Ilmiah Penelitian dan Pembelajaran Fisika, 4*(2).

Surahmat, A., & Fu'ady, T. D. (2020). Simulasi Rangkaian Seven Segment Menggunakan Multisim pada Pembelajaran Rangkaian Elektronika Analog dan Digital di SMKS Informatika Sukma Mandiri. *Journal of Innovation And Future Technology (IFTECH), 2*(1), 15-28.