Автоматизація технологічних і бізнес-процесів Volume 17, Issue 3/2025
http://www.atbp.ontu.edu.ua/

УДК 621.316.925+621.3.049

МОДЕЛЮВАННЯ ПЕРЕХІДНИХ ПРОЦЕСІВ У ЕЛЕКТРИЧНИХ КОЛАХ ПОСТІЙНОГО СТРУМУ У ХМАРНОМУ СЕРЕДОВИЩІ MULTISIM LIVE

SIMULATION MODELLING OF TRANSIENT PROCESSES IN DC CIRCUITS USING THE MULTISIM LIVE CLOUD ENVIRONMENT

[1]Савьолова Е.В.
[1]Savolova E.V.
[1]Odesa Polytechnic National University, Odesa, Ukraine
ORCID: [1] https://orcid.org/0000-0001-9266-9323
E-mail: [1]savolova.ev@op.edu.ua

DOI: 10.15673/atbp.v17i3.3257

Abstract. *Under conditions of limited access to laboratory equipment, particularly during martial law, the implementation of simulation modelling to study the behaviour of electrical circuits becomes especially relevant. This article examines the potential of using the cloud-based simulator Multisim Live to model transient processes in first- and second-order linear DC circuits. Examples of typical circuits are presented, for which simulations were carried out in Multisim Live, analytical calculations were performed, and graphs of the target quantities were plotted using Microsoft Excel. The obtained dependencies were compared with simulation results, and a complete correspondence was demonstrated at characteristic time points.*

Using a third-order circuit as an example, the advantages of simulation modelling are demonstrated in situations where analytical calculations become significantly more complex - particularly when finding the roots of the characteristic equation and determining integration constants.

The article discusses the features of the software interface, probe types, available analysis modes, output formats, and limitations of the basic licence level, which were taken into account during circuit design. Practical ways of partially overcoming these limitations are proposed.

The results obtained confirm the feasibility and effectiveness of using the Multisim Live environment for conducting laboratory work on the study of transient processes in the context of distance learning for future technical specialists.

Анотація. *В умовах обмеженого доступу до лабораторного обладнання, зокрема під час воєнного стану, особливої актуальності набуває впровадження імітаційного моделювання для дослідження властивостей електричних кіл. У статті розглянуто можливість використання хмарного симулятора Multisim Live для моделювання перехідних процесів у лінійних електричних колах постійного струму першого та другого порядків. Наведено приклади типових схем, для яких здійснено моделювання у середовищі Multisim Live, виконано аналітичні розрахунки та побудовано графіки зміни шуканої величини у Microsoft Excel. Отримані залежності порівняно з результатами симуляції, продемонстровано повну відповідність результатів у характерних точках часу.*

На прикладі кола третього порядку показано переваги імітаційного моделювання у випадках, коли складність аналітичного розрахунку істотно зростає, зокрема на етапах визначення коренів характеристичного рівняння та знаходження сталих інтегрування.

Розглянуто особливості інтерфейсу програми, типи зондів, доступні режими аналізу, формат виводу результатів, а також обмеження базового рівня ліцензії, які враховано при побудові схем. Подано практичні шляхи часткового усунення цих обмежень.

Отримані результати підтверджують доцільність і ефективність використання середовища Multisim Live під час виконання лабораторних робіт із дослідження перехідних процесів в умовах дистанційного навчання майбутніх фахівців технічних спеціальностей.

Keywords: computer simulation modelling, cloud-based simulator, Multisim Live, transient processes, electric circuits, direct current circuits.

Ключові слова: комп'ютерне імітаційне моделювання, хмарний симулятор, Multisim Live, перехідні процеси, електричні кола, кола постійного струму.

Introduction

The analysis of transient processes occurring during the transition from one steady-state operating mode to another is an essential stage in the design aaand operation of any electrotechnical device, as it is precisely at these moments that the properties of the electrical circuit, its individual sections, and components are most clearly and fully revealed.

The study of transient processes makes it possible to determine how the shape and amplitude of signals change as they pass through amplifiers, filters, and other radio-frequency devices. It enables the identification of sections with extreme loading, where voltage or current may exceed permissible limits, and helps to prevent fault conditions by appropriately selecting circuit element parameters and protective devices [1, 2, 3].

On the one hand, this ensures the stable operation of devices; on the other hand, there is a wide range of electrotechnical and electronic systems for which transient modes are part of normal operation.

Therefore, a properly conducted analysis contributes to improving the efficiency, reliability, and safety of equipment performance under real operating conditions.

Analysis of literature data and the formulation of the problem

Mastering the analysis of transient processes requires a combination of theoretical calculations and experimental investigations, typically carried out during laboratory sessions using specialised equipment. However, there is currently a real threat to the lives of educators and students due to hostile shelling, which has resulted in damage to many educational institutions.

In accordance with the recommendations of the Ministry of Education and Science of Ukraine [4], the educational process in such institutions has been organised in a remote format. Nevertheless, without the acquisition of practical skills, it is impossible to ensure the high-quality training of technical specialists.

At present, a considerable body of positive international experience has been accumulated in addressing this issue [5, 6, 7]. However, unfortunately, it cannot be fully applied in Ukraine due to the conditions of martial law.

One of the ways to partially substitute work with real equipment is through the use of computer simulation software [8, 9]. In this context, the selection of simulation programmes that enable the modelling of electrical circuits - particularly for the analysis of first- and second-order transient processes - has become especially relevant.

Ideally, such software should offer a free version, feature an intuitive interface accessible regardless of the user's level of computer literacy, require minimal time to master, and be easily installable on digital devices with any technical specifications and operating systems.

According to recent studies [9, 10, 11, 12, 13], one of the most widely used circuit simulators in engineering education is NI Multisim [14], which offers a user-friendly graphical interface, built-in analysis tools, and virtual equivalents of real measuring instruments. However, in recent times, there has been a growing number of cases where students are unable to install the free version of this software on their personal devices. The situation becomes even more complicated when the device operates on an operating system other than Windows.

In such cases, it is advisable to use Multisim Live [15] - a cloud-based circuit simulator accessible via a web browser without the need to install any dedicated software. It features a built-in component library, real-time parameter adjustment for circuit elements, and enables the generation of transient response graphs in real time.

Purpose and objectives of the study

The purpose of this work is to demonstrate the feasibility and practical benefits of using the cloud-based environment Multisim Live for performing laboratory work in electrical engineering disciplines, particularly for studying transient processes in first- and second-order electrical circuits with a constant voltage source by distance learning students.

Methods and materials of research

In the following sections of this work, examples will be presented demonstrating the use of Multisim Live for the calculation and analysis of first- and second-order transient processes in electrical circuits with a constant power supply.

Before beginning the simulation, it is advisable to examine several important aspects in more detail, including: the choice of licence level, the selection of interactive components that enable switching, and the specification of the simulation type.

In the Multisim Live environment, transient processes can be analysed using two simulation modes: Interactive Simulation and Transient Simulation. The first mode emulates the real-time operation of an oscilloscope - the simulation runs continuously until manually stopped or paused. The second mode performs a time-domain analysis of the circuit's response over a specified simulation interval. In both modes, probes must be used to observe the results, displaying graphical representations of voltage or current over time.

Switching in a circuit can be modelled using elements such as switches, relays, or pulse power sources (in the case of DC circuits).

Within the basic Multisim Live licence, which allows the use of up to five components in a circuit (excluding ground), it is advisable to investigate transient processes that occur when a circuit with zero initial conditions is connected to a power source. This approach enables switching to be implemented solely via a pulse source (Pulse Voltage Source or Clock Voltage Source), allows for the modelling of a load circuit using four components, and thereby expands the range of possible circuit configurations for analysis.

Research results

The following examples are the result of practical confirmation of the effectiveness of using the Multisim Live environment for the study of transient processes.

Figure 1 shows an electric circuit with parameters: $R_1 = 10\ \Omega$, $R_2 = R_3 = 20\ \Omega$, $L_1 = 50$ mH, which is connected to a constant source $E = 90$ V, and a graph built in Microsoft Excel corresponding to the law of the voltage change on the resistor R$_3$ $u_{R_3}(t)$.

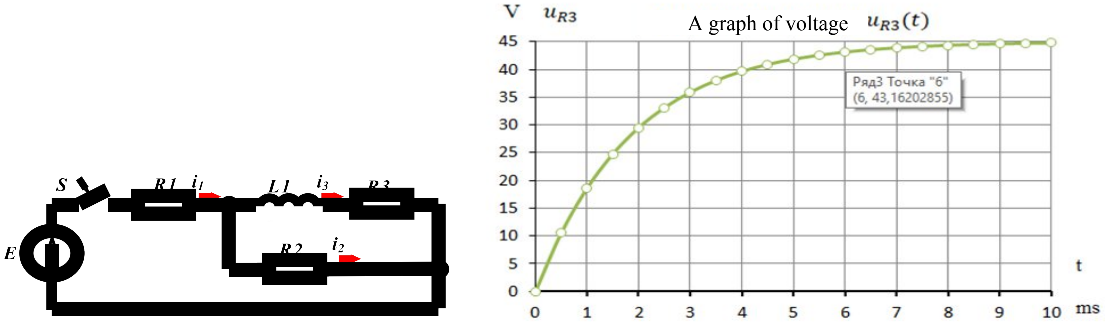


Fig. 1 - Schematic diagram of an electric circuit and a graph of voltage across resistor R$_3$ $u_{R_3}(t)$, built in *Microsoft Excel*

Taking into account the initial condition $i_{L_1}(0_+) = i_{L_1}(0\_) = 0\ A$ the voltage at the resistor terminals increases exponentially from zero to a steady value

$$u_{R_3}(t)_f = \frac{E}{R_1 + \dfrac{R_2 \cdot R_3}{R_2 + R_3}} \cdot \frac{R_2 \cdot R_3}{R_2 + R_3} = \frac{90}{10 + \dfrac{20 \cdot 20}{20 + 20}} \cdot \frac{20 \cdot 20}{20 + 20} = 45\ V$$

according to the law

$$u_{R_3}(t) = u_{R_3}(t)_f + u_{R_3}(t)_n = 45 - 45e^{-533t},\ V.$$

The value of the transient process root, $p \approx -533\ s^{-1}$, was determined using the input resistance method from the characteristic equation

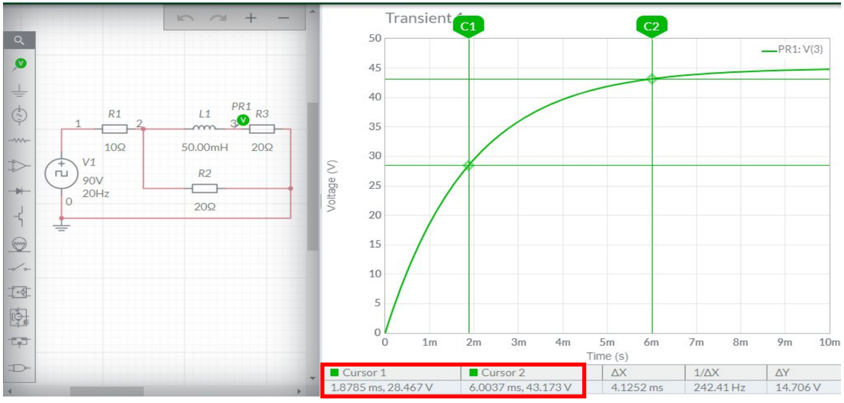
$$\frac{R_1 \cdot R_2}{R_1 + R_2} + R_3 + pL_1 = 0.$$

The time constant τ, the value of the voltage $u_{R_3}(\tau)$ and the time of the transient process are:

$$\tau = \frac{1}{|p|} = \frac{1}{|-533|} = 1{,}875\ ms;$$

$$u_{R_3}(\tau) = 45 - 45e^{-533\tau} = 28{,}435,\ V;$$

$$t_{tp} = 5 \cdot \tau = 5 \cdot 1{,}875 \cdot 10^{-3} = 9{,}375\ ms.$$



🟩 Cursor 1	🟩 Cursor 2	ΔX	1/ΔX	ΔY
1.8785 ms, 28.467 V	6.0037 ms, 43.173 V	4.1252 ms	242.41 Hz	14.706 V

Fig. 2 - Simulation result of the voltage across R$_3$ in the Multisim Live environment

The model of this electric circuit, in which the switching is carried out by a pulsed voltage source *(*Clock Voltage Source*) with the turn-on edge at time t = 0*, and the graph $u_{R_3}(t)$ in the cloud simulator Multisim Live are shown in Figure 2. A *Voltage Probe was used to observe the law of voltage change.*

For the electric circuit (Figure 3) with parameters: $R_1 = 10\ \Omega$; $R_2 = R_3 = 20\ \Omega$; $C_1 = 5\ \mu F$ the law of current change in resistor R_2 $i_2(t)$ is simulated (Figure 4).

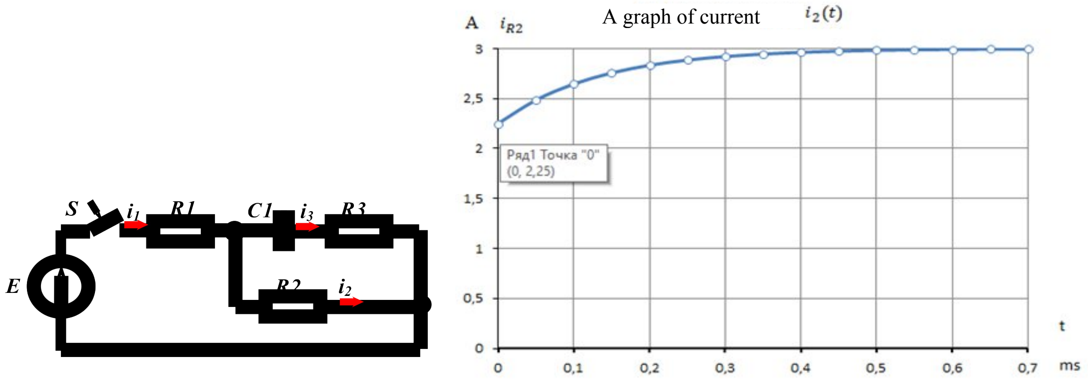


Fig. 3 - Schematic diagram of an electric circuit and a graph of current in resistor R_2 $i_2(t)$, built in *Microsoft Excel*

The law of change of current $i_2(t)$, the graph of which is shown in Figure 3, corresponds to the expression

$$i_2(t) = i_2(t)_f + i_2(t)_n = 3 - 0{,}75e^{-7500t}\ A;$$

where $i_2(t)_f$ – the value of the steady-state (forced) component

$$i_2(t)_f = \frac{E}{R_1 + R_2} = \frac{90}{10 + 20} = 3\ A;$$

$i_2(t)_n$ – is the natural (free) component, with $p \approx -7500\ s^{-1}$ - the characteristic root of the transient process, obtained using the input resistance method from the equation

$$\frac{R_1 \cdot R_2}{R_1 + R_2} + R_3 + \frac{1}{pC_1} = 0.$$

To determine the integration constant A, a system of equations was constructed for the post-switching circuit at the moment $t = 0_+$:

$$\begin{cases} i_1(0) = i_2(0) + i_3(0) \\ i_1(0) \cdot R_1 + i_2(0) \cdot R_2 = e(0) \\ i_2(0) \cdot R_2 - i_3(0) \cdot R_3 - u_{C_1}(0) = 0 \end{cases}.$$

Taking into account the independent initial condition $u_{C_1}(0_+) = u_{C_1}(0\_) = 0\ V$, the value

$$i_2(0) = \frac{e(0)}{R_1 + \frac{R_2}{R_3}R_1 + R_2} = \frac{90}{10 + \frac{20 \cdot 10}{20} + 20} = 2{,}25\ A \text{ was obtained,}$$

and the integration constant was found to be

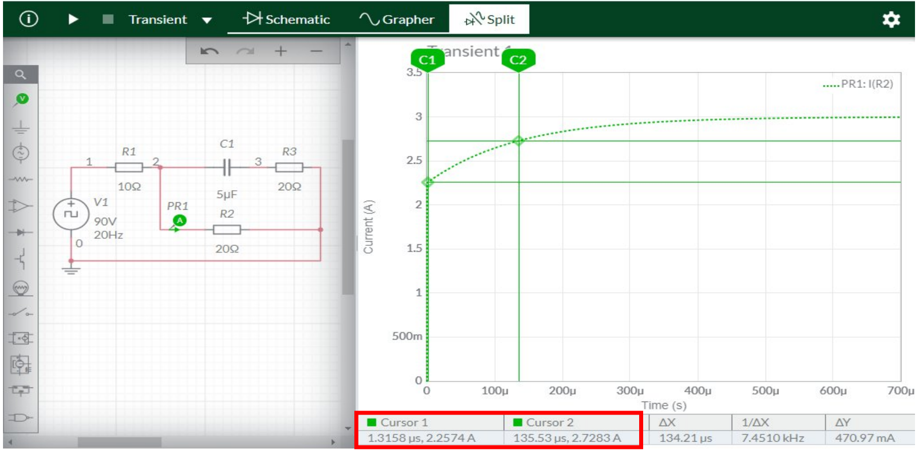
$$A = i_2(0) - i_2(0)_f = 2{,}25 - 3 = -0{,}75\ A.$$

The time constant τ, the duration of the transient process t_{tp}, and the current value $i_2(\tau)$ were calculated as follows:

$$\tau = \frac{1}{|p|} = \frac{1}{|-7500|} = 0{,}133\ ms;$$

$$t_{tp} = 5 \cdot \tau = 5 \cdot 0{,}133 \cdot 10^{-3} = 0{,}665\ ms;$$

$$i_2(\tau) = 3 - 0{,}75e^{-7500\tau} = 2{,}723\ A.$$



🟩 Cursor 1	🟩 Cursor 2	ΔX	1/ΔX	ΔY
1.3158 µs, 2.2574 A	135.53 µs, 2.7283 A	134.21 µs	7.4510 kHz	470.97 mA

Fig. 4 - Simulation result of the current $i_2(t)$ in the Multisim Live environment

To observe the law of current change $i_2(t)$ in the cloud simulator *Multisim Live*, a Current Probe was used.

The schematic diagram of the circuit with two reactive elements (with parameters: $R_1 = 32\ \Omega$, $R_2 = 50\ \Omega$, $L = 12$ mH, $C = 7\ \mu$F) and the graph of the voltage change across the resistor terminals R_1 $u_{R_1}(t)$, during the connection to a constant voltage source $E = 48$ V, are shown in Figure 5.

Since the roots of the characteristic equation

$$CL(R_1+R_2)p^2+\left(CR_1R_2+L\right)p+R_2=0$$

$$6{,}888\cdot 10^{-6}p^2+0{,}0232p+50=0$$

$$p_{1,2}=\frac{-0{,}0232\pm 0{,}029j}{13{,}776\cdot 10^{-6}}=-1683{,}5\pm j2103{,}6$$

- are complex conjugates with a negative real part, resulting in a transient response of oscillatory nature:

$$u_{R_1}(t)=u_{R_1f}+u_{R_1n}=u_{R_1f}+Ae^{\alpha t}\sin(\beta t+\psi)=u_{R_1f}+Ae^{-1683{,}5t}\sin(2103{,}6t+\psi)\ V,$$

where u_{R_1f} is the steady-state (forced) component $u_{R_1f}=0\ V$;

A, ψ – are integration constants determined from the system of equations:

$$\begin{cases} A\sin(\psi)=u_{R_1n}(0) \\ -1683{,}5A\sin(\psi)+2103{,}6A\cos(\psi)=\left.\dfrac{du_{R_1n}}{dt}\right|_{t=0} \end{cases}.$$

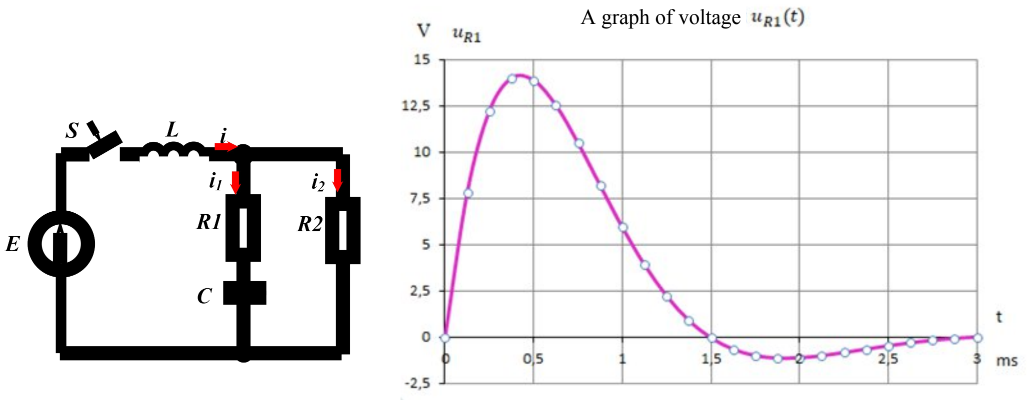


Fig. 5 - Schematic diagram of an electric circuit and a graph of voltage across resistor R_1 $u_{R_1}(t)$, built in *Microsoft Excel*

The value of the natural (free) component $u_{R_1n}(0)=u_{R_1}(0)=R_1i_1(0)=0\ V$ was obtained from the third Kirchhoff equation in the system composed for the time instant $t=0$:

$$\begin{cases} i(0) = i_1(0) + i_2(0) \\ L\left.\frac{di}{dt}\right|_{t=0} + R_1 i_1(0) + u_c(0) = e(0) \\ R_1 i_1(0) - R_2 i_2(0) - u_c(0) = 0. \end{cases} \tag{1}$$

To determine the derivative of the free component $\left.\frac{du_{R_1n}}{dt}\right|_{t=0}$ the first and second equations of system (1) were differentiated

$$\begin{cases} \left.\frac{di}{dt}\right|_{t=0} = \left.\frac{di_1}{dt}\right|_{t=0} + \left.\frac{di_2}{dt}\right|_{t=0} \\ R_1\left.\frac{di_1}{dt}\right|_{t=0} - R_2\left.\frac{di_2}{dt}\right|_{t=0} - \left.\frac{du_c}{dt}\right|_{t=0} = 0, \end{cases}$$

where $\left.\frac{di}{dt}\right|_{t=0}$ the derivative was calculated using equation two:

$$\left.\frac{di}{dt}\right|_{t=0} = \frac{E}{L} = \frac{48}{0{,}012} = 4000\ \frac{A}{s}.$$

The result of the solution is given by

$$\left.\frac{di_1}{dt}\right|_{t=0} = \frac{\left.\frac{di}{dt}\right|_{t=0}}{1+\frac{R_1}{R_2}} = \frac{4000}{1+\frac{32}{50}} = 2439\ \frac{A}{s};$$

$$\left.\frac{du_{R_1}}{dt}\right|_{t=0} = R_1\left.\frac{di_1}{dt}\right|_{t=0} = 32\cdot 2439 = 78048{,}8\ \frac{V}{s}.$$

Taking into account the derivative of the steady-state voltage component $\left.\frac{du_{R_1f}}{dt}\right|_{t=0} = 0\ \frac{V}{s}$, the derivative of the free component was found to be

$$\left.\frac{du_{R_1n}}{dt}\right|_{t=0} = \left.\frac{du_{R_1}}{dt}\right|_{t=0} - \left.\frac{du_{R_1f}}{dt}\right|_{t=0} = \left.\frac{du_{R_1}}{dt}\right|_{t=0} = 78048{,}8\ \frac{V}{s}.$$

Thus, the final form of the system for determining the integration constants is:

$$\begin{cases} A\sin(\psi) = 0 \\ -1683{,}5A\sin(\psi) + 2103{,}6A\cos(\psi) = 78048{,}8. \end{cases}$$

From the first equation, it follows that $\sin(\psi) = 0$, hence $\psi = 0°$; from second -

$$A = \frac{78048{,}8}{2103{,}6} = 37{,}1\ V.$$

The calculated expression for the voltage across resistor R_1 is:

$$u_{R_1}(t) = 37{,}1e^{-1684t}\sin(2104t)\ V.$$

The time constant τ, the voltage value $u_{R_1}(\tau)$ at t = τ, and the transient process duration t_{tp} are as follows:

$$\tau = \frac{1}{|-1683{,}5|} \approx 0{,}594\ ms;$$

$$u_{R_1}(\tau) = 37{,}1e^{-1684\tau}\sin(2104\tau) \approx 37{,}1e^{-1}\sin(71{,}6°) \approx 12{,}96\ V$$

$$t_{tp} = 5\tau = \frac{5}{|-1683{,}5|} \approx 3\ ms.$$

The result of the circuit simulation in the Multisim Live environment in two simulation modes (Interactive simulation and Transient simulation) is shown in Figure 6.

Cursor 1	Cursor 2	ΔX	1/ΔX	ΔY
592.91 µs, 12.963 V	1.5000 ms, -38.762 mV	907.09 µs	1.1024 kHz	-13.002 V

Cursor 1	Cursor 2	ΔX	1/ΔX	ΔY
590.73 µs, 12.989 V	1.5000 ms, -38.319 mV	909.27 µs	1.0998 kHz	-13.027 V

Fig. 6 – Simulation result of the voltage across R$_1$ in the Multisim Live environment

The discussion of the results

To demonstrate the correspondence between the results of analytical calculations and computer-based simulation, the time constant of the transient process and the equation describing the variation of the target quantity were determined for each of the presented examples. Based on this equation, the corresponding graph was plotted in *Microsoft Excel*. The following values were obtained:

- the duration of the transient process t_{tp};
- the value of the target quantity at the switching moment $t=0_+$ and at the moment corresponding to the time constant $t=\tau$;
- the steady-state (forced) value of the quantity.

For ease of comparison, the results of the analytical and simulation-based approaches are presented using the same scale. Cursors were placed on the oscillograms to mark the values of the target quantity at specific moments in time. The match between these values and the results of the analytical calculations confirms the adequacy of the developed models.

One of the key features of the Multisim Live simulation environment is the capability to measure physical quantities using probes, which, unlike real-world instruments, have only a single terminal. The *current probe* is placed directly on a conductor in such a way that the direction of its arrow corresponds to the conventionally positive direction of current flow within the branch. To measure the voltage between points in a circuit, the *voltage probe* is connected to the point with the higher potential, while a *Voltage Reference* is connected to the point with the lower potential. If the reference probe is not installed, the voltage is measured relative to the global ground of the circuit (see Fig. 6).

In addition, the environment provides a *combined probe* (*Voltage and Current Probe*), which allows for the simultaneous measurement of the current through a component and the voltage across its terminals. For correct voltage measurement, the global ground of the circuit should be connected to the terminal with the lower potential, as this type of probe does not support the use of a Voltage Reference.

The process of circuit construction is significantly accelerated by the use of keyboard shortcuts. Unlike simulators that require installation on a local computer, Multisim Live enables the rapid insertion of components using such shortcuts, which is particularly advantageous when working on mobile devices.

Activating the built-in browser translator allows both the interface and help materials to be translated into Ukrainian, thereby facilitating the learning process for users with only a basic command of English.

The simulator also serves as an effective tool for investigating transient processes in electrical circuits that contain a large number of reactive elements. The mathematical analysis of such circuits is often complex, particularly during the stages of determining the roots of the characteristic equation and calculating the integration constants, and frequently requires the use of supplementary online resources. The complexity of these calculations increases substantially with the order of the circuit.

In contrast, simulative modelling enables visual analysis of voltage and current waveforms without performing complex calculations, allowing for the identification of their peak values and evaluate the dynamics of change in time.

For the third-order electrical circuit (Figure 7) with the parameters: $R_1 = 32\ \Omega$, $R_2 = 50\ \Omega$, $L_1 = 12$ mH, $C = 70$ µF, $L_2 = 10$ mH, connected to a DC source with an EMF of E = 41 V, a voltage waveform across resistor R_1 $u_{R_1}(t)$, calculated using the classical method, is presented.

Since the roots of the characteristic equation

$$p^3 C L_1 L_2 + p^2 C(L_1 R_2 + L_2 R_1) + p(C R_1 R_2 + (L_1 + L_2)) + R_1 + R_2 = 0$$

$p_1 = -4730.84$ s^{-1}; $p_2 = -1165.74$ s^{-1}; $p_3 = -1770.08$ s^{-1} – are real and negative [16, 17], the voltage response is aperiodic.

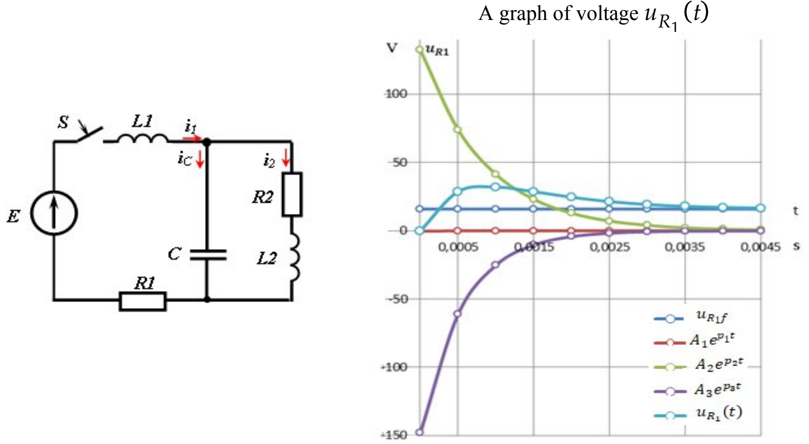

A graph of voltage $u_{R_1}(t)$

Fig. 7 – Schematic diagram of an electric circuit and a graph of voltage across resistor *R₁* $u_{R_1}(t)$, built in *Microsoft Excel*

The voltage across resistor R_1 is expressed as:

$$u_{R_1}(t) = u_{R_1f} + u_{R_1n} = u_{R_1f} + A_1e^{p_1t} + A_2e^{p_2t} + A_3e^{p_3t},$$

where the steady-state (forced) component is

$$u_{R_1f} = \frac{E}{R_1 + R_2}R_1 = \frac{41}{82}\cdot 32 = 16\ V;$$

A_1, A_2, A_3 – are integration constants determined from the system:

$$\begin{cases} u_{R_1n}(0) = A_1 + A_2 + A_3 \\ \left.\dfrac{du_{R_{1n}}}{dt}\right|_{t=0} = p_1A_1 + p_2A_2 + p_3A_3 \\ \left.\dfrac{d^2u_{R_{1n}}}{dt^2}\right|_{t=0} = p_1^2A_1 + p_2^2A_2 + p_3^2A_3. \end{cases}$$

Given zero initial conditions:

$$u_{R_1n}(0) = u_{R_1}(0) - u_{R_1f}(0) = i_1(0)R_1 - u_{R_1f}(0) = 0 - 16 = -16\ V.$$

From the second equation of the system:

$$\begin{cases} i_1(0) = i_C(0) + i_2(0) \\ L_1\left.\dfrac{di_1}{dt}\right|_{t=0} + R_1i_1(0) + u_c(0) = e(0) \\ L_2\left.\dfrac{di_2}{dt}\right|_{t=0} + R_2i_2(0) - u_c(0) = 0 \end{cases}$$

the derivative of the current $\left.\frac{di_1}{dt}\right|_{t=0}$ was determined to obtain the value $\left.\frac{du_{R_{1n}}}{dt}\right|_{t=0}$. The derivative of the steady-state voltage value is zero, therefore

$$\left.\frac{du_{R_{1n}}}{dt}\right|_{t=0} = R_1\left.\frac{di_1}{dt}\right|_{t=0} = R_1\frac{e(0)}{L_1} = 32\cdot\frac{41}{0.012} = 109333.3\ \frac{V}{s}.$$

The second derivative of the voltage $\left.\frac{d^2u_{R_{1n}}}{dt^2}\right|_{t=0}$ (the second derivative of the steady-state voltage value is zero)

$$\left.\frac{d^2u_{R_{1n}}}{dt^2}\right|_{t=0} = R_1\left.\frac{d^2i_1}{dt^2}\right|_{t=0} = -\frac{R_1}{L_1}\cdot\left.\frac{du_{R_{1n}}}{dt}\right|_{t=0} = -\frac{32}{0.012}\cdot 109333.3 = -\ 291\ 555\ 555{,}6\ \frac{V}{s}$$

calculated by differentiating the same equation with substitution of the value of the first derivative:

$$L_1 \left.\frac{d^2 i_1}{dt^2}\right|_{t=0} + R_1 \left.\frac{di_1}{dt}\right|_{t=0} + \left.\frac{du_C}{dt}\right|_{t=0} = \left.\frac{de}{dt}\right|_{t=0}$$

$$L_1 \left.\frac{d^2 i_1}{dt^2}\right|_{t=0} + R_1 \frac{e(0)}{L_1} + \frac{i_C(0)}{C} = 0.$$

According to the calculation results [16, 17] $A_1 = -0{,}34\ \frac{V}{s}$; $A_2 = 132{,}39\ \frac{V}{s}$; $A_3 = -148{,}05\ \frac{V}{s}$, and the complete expression is

$u_{R_1}(t) = u_{R_1 f} + u_{R_1 n} = 16 - 0{,}34e^{-4730.84t} + 132{,}39e^{-1165.74t} - 148{,}05e^{1770{,}08t}\ V.$

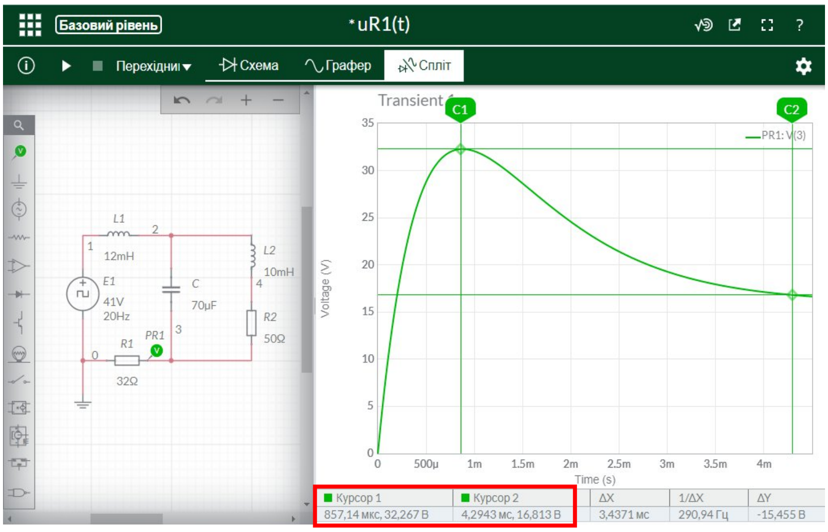
The time constant τ, the voltage value $u_{R_1}(\tau)$ at t = τ, and the transient process duration t_{tp} are as follows:

$$\tau = \frac{1}{|-1165{,}74|} \approx 0{,}858\ ms;$$

$$u_{R_1}(\tau) = 16 - 0{,}34e^{-4730.84\tau} + 132{,}39e^{-1165.74\tau} - 148{,}05e^{1770{,}08\tau} \approx 32{,}26\ V$$

$$t_{tp} = 5\tau = \frac{5}{|-1165{,}74|} \approx 4{,}29\ ms.$$

The simulation results (Figure 8) fully correspond to the analytical calculations, as confirmed by the cursor readings at points t = τ and t = 5τ.



🟩 Курсор 1	🟩 Курсор 2	ΔX	1/ΔX	ΔY
857,14 мкс, 32,267 В	4,2943 мс, 16,813 В	3,4371 мс	290,94 Гц	-15,455 В

Fig. 8 – Simulation result of the voltage across R₁ in the Multisim Live environment

When the capacitance of the capacitor is set to C = 7 μF the mathematical analysis becomes more complex. The characteristic equation yields one real and two complex-conjugate roots [16, 17]:

$$p_1 = -3930\ \text{s}^{-1};\ p_{2,3} = -1868{,}3 \pm j4620{,}4\ ,$$

as a result, the voltage response exhibits a periodic (oscillatory) behaviour:

$$u_{R_1}(t) = u_{R_1 f} + u_{R_1 n} == u_{R_1 f} + A_1 e^{p_1 t} + A_2 e^{\delta t}\sin\left(\omega_0 t + \psi\right) = u_{R_1 f} + A_1 e^{-3930t} + A_2 e^{-1868.3t}\sin\left(4620.4t + \psi\right)\ V,$$

where A_1, A_2, ψ – are integration constants determined from the system:

$$\begin{cases} u_{R_1 n}(0) = A_1 + A_2 \sin(\psi) \\ \left.\dfrac{du_{R_1 n}}{dt}\right|_{t=0} = p_1 A_1 + \omega_0 A_2 \cos(\psi) + \delta A_2 \sin(\psi) \\ \left.\dfrac{d^2 u_{R_1 n}}{dt^2}\right|_{t=0} = p_1^2 A_1 + 2\delta\omega_0 A_2 \cos(\psi) + (\delta^2 - \omega_0^2) A_2 \sin(\psi). \end{cases}$$

Taking into account the previously calculated values $u_{R_1 n}(0), \left.\frac{du_{R_{1n}}}{dt}\right|_{t=0}$ and $\left.\frac{d^2u_{R_{1n}}}{dt^2}\right|_{t=0}$, the integration constants were found to be: $A_1 = -10{,}96\ \frac{V}{s}$; $A_2 = 13{,}3\ \frac{V}{s}$; $\psi = -22{,}29°$ [16, 17]. The analytical expression describing the voltage across the resistor is given by:

$$u_{R_1}(t) = u_{R_1 f} + u_{R_1 n} = 16 - 10{,}96e^{-3930t} + 13{,}3e^{-1868.3t}\sin(4620{,}4t - 22{,}29°)\ V.$$

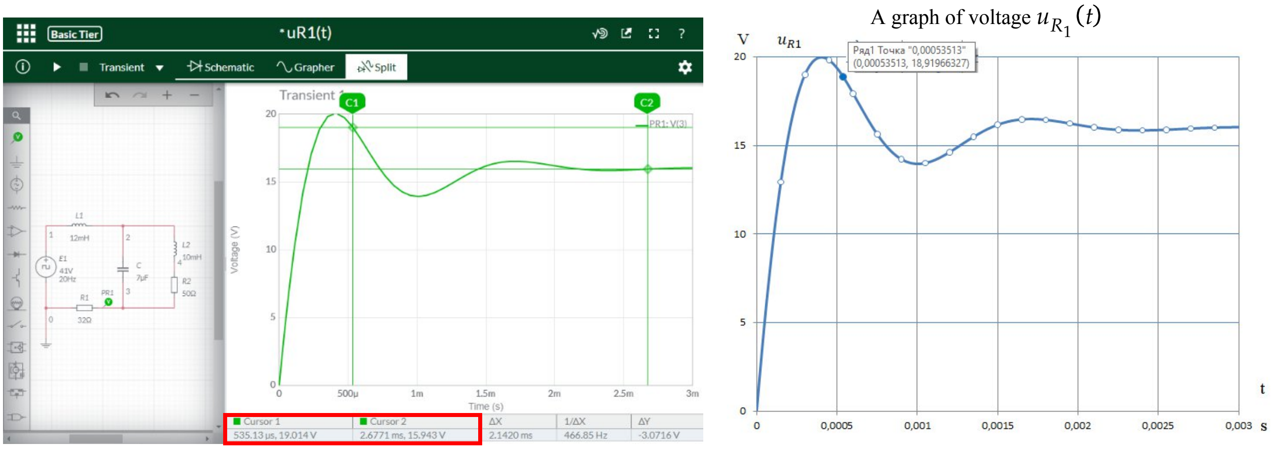
Time constant and response characteristics:

$$\tau = \frac{1}{|-1868{,}3|} \approx 0{,}535\ ms;$$

$$u_{R_1}(\tau) = 16 - 10.96e^{-3930\tau} + 13.3e^{-1868.3\tau}\sin(4620.4\tau - 22.29°) \approx 18{,}92\ \ V$$

$$t_{tp} = 5\tau = \frac{5}{|-1868{,}3|} \approx 2{,}675\ ms.$$

The corresponding results of the analytical calculation and simulation are presented in Figure 9. The relative error between the voltage values at the point t = τ is less than 0.5%.



Cursor 1	Cursor 2	ΔX	1/ΔX	ΔY
535.13 μs, 19.014 V	2.6771 ms, 15.943 V	2.1420 ms	466.85 Hz	-3.0716 V

Ряд1 Точка "0,00053513"
(0,00053513, 18,91966327)

Fig. 9 – Voltage variation law across R₁. Simulation result in Multisim Live and graph in Microsoft Excel

Conclusions

Multisim Live is an interactive online environment with several notable advantages, including:

- optimisation for touchscreen devices;
- availability of a free basic licence;
- compatibility with devices running any operating system (Windows, macOS, Linux, iOS, or Android);
- ease and speed of mastering the interface regardless of the user's proficiency in a foreign language;
- access to circuit designs from any device (mobile phone, tablet, or computer) and the ability to resume work at any time thanks to cloud-based project storage;
- availability of reference materials and an open library of circuits developed by other users.

However, it also has certain limitations:

- it cannot be used without a stable Internet connection, which poses challenges for learners in regions with poor connectivity;
- the free licence does not provide access to some components (e.g. Pulse or Arbitrary sources), limits the number of components in a circuit, and allows only four types of analysis;
- there is a potential risk of data loss in the event of a cloud service failure.

Some of these limitations can be partially mitigated by:

- opening and modifying circuits created by other users that include components unavailable in the basic licence version;
- upgrading to the standard licence, which allows the use of up to fifteen components per circuit and supports six types of analysis.

Thus, Multisim Live, when used with consideration of its licensing terms and functional constraints, can be effectively employed for organising laboratory work - particularly in the context of remote learning.

Список використаних джерел

1. Теоретичні основи електротехніки. Перехідні процеси в лінійних колах. Синтез лінійних кіл. Електричні та магнітні нелінійні кола : підручник / Ю. О. Карпов [та ін.] ; за ред. Ю. О. Карпова. Вінниця : ВНТУ, 2012. 530 с. URL: https://jetiq.vntu.edu.ua/fdb/671/Books/TOE_II_(1).pdf (дата звернення: 26.04.2025).

2. Розрахунок перехідних процесів у лінійних електричних колах : навч. посіб. / І. А. Курило [та ін.]. Київ : НТУУ «КПІ», 2013. 288 с. URL: https://ela.kpi.ua/server/api/core/bitstreams/b3cb3d9a-c73e-496f-976d-489f728898c5/content (дата звернення: 26.04.2025).

3. Рибалко М. П., Есауленко В. О., Костенко В. І. Теоретичні основи електротехніки. Лінійні електричні кола : підручник. Донецьк : Новий світ, 2003. 513 с.

4. Міністерство освіти і науки України. Про рекомендацію навчання в дистанційному форматі : лист МОН № 1/2484-23 від 21.02.2023 р. [Електронний ресурс]. Режим доступу: http://osvita.ua/legislation/Ser_osv/88527/ (дата звернення: 27.04.2025).

5. Trunova I., Arhun S., Hnatov A., Apse-Apsitis P., Kunicina N., Myhal V. Sustainable Approach Development for Education of Electrical Engineers in Long-Term Online Education Conditions // Sustainability. 2023. Vol. 15, No. 18. Article No. 13289.DOI: https://doi.org/10.3390/su151813289. Режим доступу: https://link.gale.com/apps/doc/A780021682/AONE?u=anon~8bf04038&sid=googleScholar&xid=4cd057d8 (дата звернення: 26.04.2025).

6. Stoyanova D., Stoyanova-Petrova S., Kafadarova N., Sotirov S., Mileva N. Our experience from the use of innovative technologies in teaching “electrical engineering” during the COVID-19 pandemic // *Proceedings of SOCIOINT 2022: 9th International Conference on Education & Education of Social Sciences*, 12–14 червня 2022 р., Istanbul, Turkey. Istanbul: OCERINT, 2022. С. 23–27. DOI: https://doi.org/10.46529/socioint.202204 (дата звернення: 26.04.2025).

7. Архун С., Трунова І., Хнатов А., Хнатова Х. Інноваційні підходи до навчання студентів технічних спеціальностей в умовах війни в Україні // Автомобіль і електроніка. Сучасні технології. 2023. № 24. С. 83–90. DOI: 10.30977/VEIT.2023.24.0.4.

8. Dosymov Y. Effectiveness of Computer Modeling in the Study of Electrical Circuits: Application and Evaluation // Int. J. Eng. Ped. 2023. Vol. 13, No. 4. С. 93–112. DOI: https://doi.org/10.3991/ijep.v13i4.34921 (дата звернення: 06.05.2025).

9. Todos P., Vîrlan P, Tertea Gh. Training of practical engineering skills in the context of the COVID - 19 pandemic // Journal of Social Sciences. 2021. Vol. 4, No. 3. С. 18–27. DOI: https://doi.org/10.52326/jss.utm.2021.4(3).02 (дата звернення: 06.05.2025).

10. Бурячок В. Л, Коршун Н. В., Шевченко С. М., Складанний П. М. Застосування середовища NI MULTISIM при формуванні практичних навичок студентів спеціальності 125 «Кібербезпека» // Кібербезпека:освіта, наука, техніка. 2020. Vol. 1, No. 9. С. 159–169. DOI: https://doi.org/10.28925/2663-4023.2020.9.159169 (дата звернення: 06.05.2025).

11. Семенюк Д. П., Якушенко Є. М. Проєктування схем та моделювання без програмного забезпечення при викладанні електротехнічних дисциплін // International scientific conference «Information technologies and management in higher education and sciences»: тези доповідей. Частина 3. Fergana, Republic of Uzbekistan, November 28, 2022. С. 99–102. DOI: https://doi.org/10.30525/978-9934-26-277-7-208 (дата звернення: 30.05.2025).

12. Vasetska, L. A study of the electric circuit modelling and simulation software efficiency and their accuracy, speed and ease of use comparison // Bulletin of Cherkasy State Technological University. 2024. Т. 29, № 2. С. 32–44. DOI: https://doi.org/10.62660/bcstu/2.2024.32 (дата звернення: 30.05.2025).

13. Павловська Н., Павловський Ю. Особливості вивчення основ електротехніки на заняттях з технологій в умовах дистанційного навчання // Український педагогічний журнал. 2023. Вип. 1. С. 133–140. DOI: https://doi.org/10.32405/2411-1317-2023-1-133-140 (дата звернення: 06.05.2025).

14. National Instruments. What is NI Multisim™? Circuit Simulator Software [Електронний ресурс]. – Режим доступу: https://www.ni.com/en/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html – Назва з екрана.

15. National Instruments. Multisim Live Online Circuit Simulator [Електронний ресурс]. – Режим доступу: https://www.multisim.com/ – Назва з екрана.

16. MathDF. Equation Solver - Calculate Equations, Inequalities & Systems [Електронний ресурс] / Онлайн калькулятор для розв’язування рівнянь та нерівностей. – Режим доступу: https://mathdf.com (дата звернення: 11.06.2025).

17. OpenAI. ChatGPT – solving equation systems and cubic equations [Електронний ресурс]. – Режим доступу: https://chatgpt.com (дата звернення: 12.06.2025).

References

[1] Yu. O. Karpov *et al.*, *Teoretychni osnovy elektrotekhniky. Perekhidni protsesy v liniinykh kolakh. Syntes liniinykh kil. Elektrychni ta mahnetni nelineini kola* [Textbook], Yu. O. Karpov (Ed.). Vinnytsia: VNTU, 2012, 530 p. [Online]. Available: https://jetiq.vntu.edu.ua/fdb/671/Books/TOE_II_(1).pdf [Accessed: Apr. 26, 2025].

[2] I. A. Kurylo *et al.*, *Rozrakhunok perekhidnykh protsesiv u liniinykh elektrychnykh kolakh* [Study guide]. Kyiv: NTUU "KPI", 2013, 288 p. [Online]. Available: https://ela.kpi.ua/server/api/core/bitstreams/b3cb3d9a-c73e-496f-976d-489f728898c5/content [Accessed: Apr. 26, 2025].

[3] M. P. Rybalko, V. O. Esaulenko, and V. I. Kostenko, *Teoretychni osnovy elektrotekhniky. Liniini elektrychni kola* [Textbook]. Donets'k: Novyi svit, 2003, 513 p.

[4] Ministerstvo osvity i nauky Ukrainy, “Pro rekomendatsiiu navchannia v dystantsiinomu formati. Lyst MON No. 1/2484-23 vid 21 liutoho 2023 roku,” Feb. 21, 2023. [Online]. Available: http://osvita.ua/legislation/Ser_osv/88527/ [Accessed: Apr. 27, 2025].

[5] I. Trunova, S. Arhun, A. Hnatov, P. Apse-Apsitis, N. Kunicina, and V. Myhal, “Sustainable Approach Development for Education of Electrical Engineers in Long-Term Online Education Conditions,” *Sustainability*, vol. 15, no. 18, Art. no. 13289, 2023. [Online]. Available: https://doi.org/10.3390/su151813289 [Accessed: Apr. 26, 2025].

[6] D. Stoyanova, S. Stoyanova-Petrova, N. Kafadarova, S. Sotirov, and N. Mileva, “Our experience from the use of innovative technologies in teaching ‘electrical engineering’ during the COVID-19 pandemic,” in *Proc. SOCIOINT 2022 - 9th Int. Conf. Educ. & Educ. of Social Sciences*, Istanbul, Turkey, Jun. 12–14, 2022, pp. 23–27. [Online]. Available: https://doi.org/10.46529/socioint.202204 [Accessed: Apr. 26, 2025].

[7] S. Arkhun, I. Trunova, A. Khnatov, and K. Khnatova, “Innovatsiini pidkhody do navchannia studentiv tekhnichnykh spetsialnostei v umovakh viiny v Ukraini,” *Avtomobil i elektronika. Suchasni tekhnolohii*, no. 24, pp. 83–90, 2023. doi: 10.30977/VEIT.2023.24.0.4.

[8] Y. Dosymov, “Effectiveness of Computer Modeling in the Study of Electrical Circuits: Application and Evaluation”, *Int. J. Eng. Ped.*, vol. 13, no. 4, pp. 93–112, Jun. 2023. Available: https://doi.org/10.3991/ijep.v13i4.34921 [Accessed: May 06, 2025].

[9] P. Todos, P. Vîrlan, Gh. Tertea, “Training of practical engineering skills in the context of the COVID - 19 pandemic”, *Journal of Social Sciences,* Vol. 4, No. 3, pp. 18–27, September 2021. Available: https://doi.org/10.52326/jss.utm.2021.4(3).02 [Accessed: May 06, 2025].

[10] V. Buriachok, N. Korshun, S. Shevchenko, i P. Skladannyi. ” Application of NI MULTISIM environment in the practical skills building for students of 125 "Cybersecurity" specialty”, *Cybersecurity: Education, Science, Technique*, Vol. 1, No. 9, pp. 159–169, September 2020. Available: https:// https://doi.org/10.28925/2663-4023.2020.9.159169 [Accessed: May 06, 2025].

[11] D. P. Semenyuk and Y. M. Yakushenko, "Proektuvannia skhem ta modeliuvannia bez prohramnoho zabezpechennia pry vykladanni elektrotekhnichnykh dystsyplin," in *Proceedings of the International Scientific Conference on Information Technologies and Management in Higher Education and Sciences,* Part 3, Fergana, Republic of Uzbekistan, Nov. 28, 2022, pp. 99–102. DOI: https://doi.org/10.30525/978-9934-26-277-7-208 [Accessed: May 30, 2025].

[12] L. Vasetska, "A study of the electric circuit modelling and simulation software efficiency and their accuracy, speed and ease of use comparison," *Bulletin of Cherkasy State Technological University,* vol. 29, no. 2, pp. 32–44, 2024. DOI: https://doi.org/10.62660/bcstu/2.2024.32 [Accessed: May 30, 2025].

[13] N. Pavlovska and Yu. Pavlovskyi, “Osoblyvosti vyvchennia osnov elektrotekhniky na zaniattiakh z tekhnolohii v umovakh dystantsiinoho navchannia,” *Ukrainskyi pedahohichnyi zhurnal*, no. 1, pp. 133–140, 2023. [Online]. Available: https://doi.org/10.32405/2411-1317-2023-1-133-140. [Accessed: May 6, 2025].

[14] National Instruments, “What is NI Multisim™? Circuit Simulator Software.” [Online]. Available: https://www.ni.com/en/shop/electronic-test-instrumentation/application-software-for-electronic-test-and-instrumentation-category/what-is-multisim.html. [Accessed: Jul. 4, 2025].

[15] National Instruments, “Multisim Live Online Circuit Simulator.” [Online]. Available: https://www.multisim.com/. [Accessed: Jul. 4, 2025].

[16] MathDF, "Equation Solver — Calculate Equations, Inequalities & Systems," [Online]. Available: https://mathdf.com. [Accessed: Jun. 11, 2025].

[17] OpenAI, “ChatGPT – solving equation systems and cubic equations,” [Online]. Available: https://chatgpt.com. [Accessed: Jun. 12, 2025].

Отримана в редакції 12.06.2025. Прийнята до друку 18.06.2025. Received 12 June 2025. Approved 18 June 2025. Available in Internet 30 June 2025