BULLETIN of Cherkasy State Technological University

Journal homepage: https://bulletin-chstu.com.ua/en

Vol. 29 No. 2. 2024

UDC 004.94: 621.3.049.7:004.4
DOI: 10.62660/bcstu/2.2024.32

A study of the electric circuit modelling and simulation software efficiency and their accuracy, speed and ease of use comparison

Larysa Vasetska*

PhD in Technical Sciences, Associate Professor
Luhansk Taras Shevchenko National University
36000, 3 Ivan Bank Str., Poltava, Ukraine
https://orcid.org/0000-0002-4873-8835

Abstract. The increasing complexity of microchips and the limitations of miniaturisation are making the electronics design process more complex and time-consuming. The problem of Moore's Law losing its force is causing rapid development in the design and manufacture of electronic components. The study aimed to provide structured information on electrical circuit design tools by functionality and availability to help optimise operations, increase productivity and save money. The study provided a theoretical description of the principles of operation of software for modelling and simulation of electrical circuits, an overview of algorithms and methods used in automatic design programs, a classification of programs by key characteristics and a comparative analysis of popular software packages: LTspice, EAGLE, MATLAB/Simulink, Multisim, Proteus, KiCad. The analysis addressed the speed of modelling, accuracy of results, ease of use, functionality and accessibility, as well as user experience. The study recommended selecting software depending on the user's needs, project complexity and budget, accounting for the experience of other users. LTspice, Multisim, and KiCad were recommended for beginners and students: LTspice is powerful, free and easy to use, KiCad offers open source and an active community, and Multisim is suitable for learning. MATLAB/Simulink, Proteus and EAGLE are recommended for professionals: Simulink provides powerful modelling and integration with other MathWorks products; Proteus is optimal for microcontroller system developers; EAGLE is suitable for integration with CAD and automation systems. The practical value of the research results lies in the creation of a rating of EDA software by the criteria of functionality and performance, designed to help users make a choice based on their needs

Keywords: electronics design tools; circuit analysis; component library; supported standards; interface

INTRODUCTION

Electrical circuit simulation (EDA) software is a versatile tool for design, training, analysis and research in electronics that can be used to test models quickly and flexibly without the risk of damaging components, saving time and money and reducing the risk of errors. The graphical and textual interface software visualises electrical quantities that can be used to compare designs and investigate the effects of components, making learning dynamic and interactive. Although EDA software has undeniable advantages, it has limitations due to its cost and the possibility of inaccurate modelling. The development of electrical circuit simulation software began with early computers and is constantly evolving. The first programs for numerical solutions of differential equations appeared in the 1950s. The Simulation Program with Integrated Circuit Emphasis (SPICE) software, developed in 1973, was the first publicly available tool for analysing electrical circuits and quickly gained popularity due to its power and accuracy.

Article's History: Received: 09.01.2024; Revised: 11.04.2024; Accepted: 27.05.2024

Suggested Citation:
Vasetska, L. (2024). 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*, 29(2), 32-44. doi: 10.62660/bcstu/2.2024.32.

*Corresponding author

As noted in the review of the evolution, current state and trends of EDA software by N. Charig (2021), in the 1980s and 1990s, new versions of SPICE appeared with improved component models and analysis methods. Commercial products based on SPICE offered additional features and technical support. Graphical interfaces have simplified the process of creating and modelling schemes. In the 2000s and 2010s, modelling tools were integrated with Computer-Aided Design (CAD) systems. New features such as high-frequency (HF) circuit analysis, thermal analysis, and electromagnetic compatibility (EMC) analysis have been added. This development allowed engineers to perform complex analysis and design at one workstation.

According to K.I. Gubbi *et al.* (2022), modern modelling tools often employ cloud computing to increase capacity and availability, as well as artificial intelligence to automate and optimise design processes. Software that allows simultaneous modelling of various physical phenomena, such as electrical, thermal, mechanical and electromagnetic processes, is actively emerging. Tools such as Qucs and NGSPICE continue to evolve as open platforms that provide accessibility and flexibility for users. Modern research on EDA tools, by G. Huang *et al.* (2021), covers a wide range of methods and algorithms for modelling, analysis, specification, computer-aided design, testing, and verification of complex electronic systems, making them an indispensable tool for modern electronic design.

A report by G. Marinova & A. Bitri (2021), which assesses the dynamics of EDA software development, indicates that electronic systems design automation is one of the most complex and dynamic industries, requiring companies to invest heavily in research and development, as well as continuous product improvement and adaptation to changing market conditions. As noted by R.I. Bahar *et al.* (2020), changes in electronic design are driving new research: future advances in electronics rely on extreme-scale design automation to achieve breakthroughs in performance, power, and integration for next-generation systems. At the same time, C.-D. Curiac & A. Doboli (2022) argue that their analysis of the EDA industry has revealed a significant gap between basic research and applied development, which leads to a decrease in the industrial influence on the development of electronics. To address this problem, the researchers suggested stimulating cooperation between academic institutions and industry, focusing on current industry issues, and simplifying scientific formulations to make research results more accessible to a wide range of professionals.

In Ukraine, LTSpice, Proteus, Arduino, MATLAB/Simulink, and Multisim tools are used in engineering education and research. D. Arseniuk & Yu. Zinkovsky (2023) used LTSpice to model a bridgeless power factor corrector. V. Shamonia *et al.* (2019) employed the Proteus virtual environment to teach future IT professionals how to design printed circuit boards and develop microcontrollers. V. Andriienko & M. Bondarenko (2018) analysed robotic platforms for prototyping electronic devices and systems. O. Semenikhina *et al.* (2020) compared the most popular Proteus and Multisim programmes among university teachers. A pedagogical experiment has shown that using Multisim by IT students allows them to perform laboratory work more efficiently due to shorter task completion times. At the same time, both programmes promote the development of skills in modelling and analysing electronic circuits, which is confirmed by the reduction in student learning time.

The problem of choosing EDA software is relevant and complex, as it requires a thorough analysis of user needs, consideration of software characteristics, and constant updating of information about available solutions. Thus, the study aimed to compare the efficiency of software for modelling and simulating electrical circuits. The study is designed to help users make a correct choice, helping to optimise operations, increase productivity and save time and money.

MATERIALS AND METHODS

An overview of the mathematical foundations of EDA software was provided to enable a comparative analysis of EDA programs in terms of accuracy, speed, and usability. The flowchart of the electronic circuit modelling process explains the general algorithm for applying data processing methods in EDA applications. To systematise knowledge about design tools and form a holistic view of the process of developing electronic systems, a classification of software by the characteristics under study was created: by the area and industry of application, commercial availability, operating system, functionality, as well as by the method, type and complexity of modelling, which is the basis for a comparative analysis of different tools.

A comparative review of popular software for designing electrical circuits by key characteristics and functionalities was conducted. A detailed analysis of popular software packages was conducted with a focus on mathematical models and modelling capabilities. This provided a list and description of common data processing methods used in popular tools for circuit modelling. Methods for optimising the operation of EDA programs to increase computing speed and reduce memory consumption were analysed.

The quantitative analysis compared the characteristics of the programmes: modelling speed, accuracy of results, ease of use, functionality, and cost. EDA software developer websites were used for the quantitative analysis: LTspice, Simulink, Multisim Live, Proteus, KiCad, Easily Applicable Graphical Layout Editor (EAGLE) and literature review. Information from the manual on designing and analysing circuits using PSpice and MATLAB tools by W.Y. Yang *et al.* (2020), which focuses on how these tools can model electrical circuits, even though their functionality is significantly different was used: MATLAB and PSpice complement each other, allowing an engineer to effectively solve a wide range of tasks, from concept development to debugging a finished device. A book edited by S.L. Tripathi *et al.* (2022), which includes a comprehensive and up-to-date introduction to the basic concepts of digital design and digital design verification using Verilog/Hardware Description Language (HDL), was also used. Mathematical models, data processing methods and possibilities for optimising the operation of programs when modelling digital circuits in LTspice, Simulink, and Multisim tools were analysed separately. Simplified modelling capabilities of KiCad, EAGLE and Proteus tools with a focus on quick verification of circuits and evaluation of their performance, as well as on one of the significant advantages of these programs – openness for development, were highlighted. With support for plug-ins and scripts, these tools are more adaptable and allow users to create personal workflows and integrate applications with other tools.

The information obtained was systematised and presented in the form of summary tables, covering the components of the software functionality: set of models, calculation accuracy, simulation speed, commercial availability, complexity of use, and availability of additional features. The results are intended to provide users with structured information about the capabilities and limitations of EDA software, help optimise their work and increase productivity.

RESULTS

EDA software systems are based on mathematical models that describe the behaviour of various electronic components and systems: linear differential equations, nonlinear differential equations, algebraic equations, and partial differential equations. The use of systems of equations to describe circuits in EDA is a fundamental approach. The choice between linear and nonlinear systems of equations in EDA depends on the complexity of the circuit model, the specifics of the task analysis and the required accuracy of the results.

Linear differential equations are widely used to describe the behaviour of passive elements (resistors, capacitors, inductors), as well as active elements in linear operation (transistors, operational amplifiers). Nonlinear differential equations are used to model nonlinear elements (diodes, transistors in the nonlinear mode), as well as to describe complex physical phenomena such as saturation of a magnetic circuit, and electrical breakdown.

Algebraic equations are used to describe the static characteristics of elements (e.g., the volt-ampere characteristics of diodes). Partial differential equations are used to model distributed parameters of systems, such as transmission lines and chip structures. The modelling process in EDA software is shown in Figure 1.

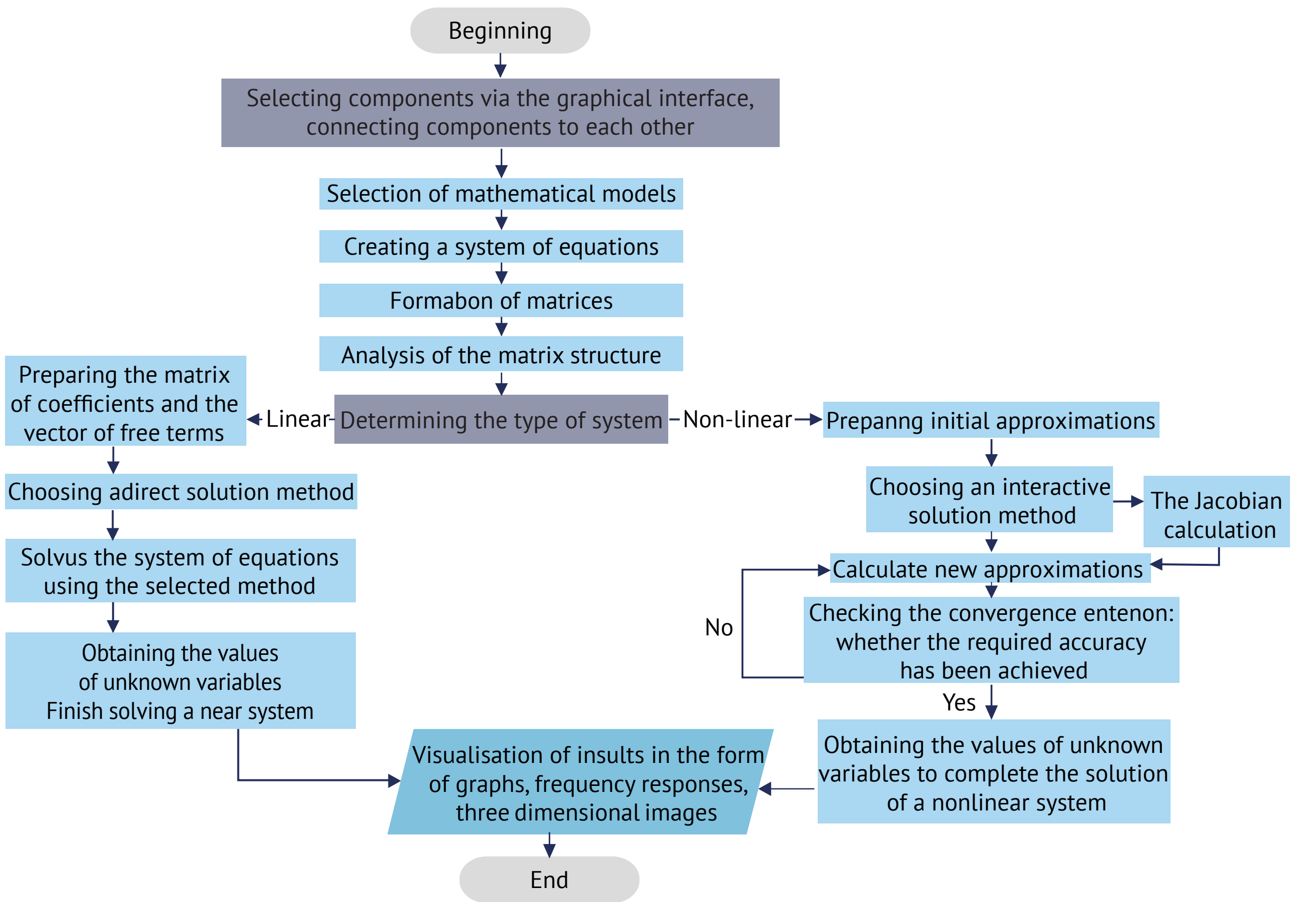

Figure 1. Flowchart of the electronic circuit modelling process

Source: compiled by the author

A distinctive feature of nonlinear differential equations is the absence of an analytical solution in the general case. Therefore, numerical methods are used to solve them. Electronic circuit modelling software uses numerical methods adapted to the specifics of electrical engineering tasks: numerical integration – for solving differential equations (the choice of method depends on the required accuracy and speed of calculations); matrix methods – for solving systems of linear algebraic equations; optimisation methods – for finding optimal model parameters. The methods applied are shown in Table 1.

Table 1. Data processing methods in EDA applications

Method category	Method	Description
Numerical integration	Euler's method	A simpler numerical method for solving ordinary differential equations that uses first derivatives.
	Runge-Kutta method	A more accurate numerical method that uses intermediate estimates of derivatives.
	The trapezoid method	A numerical method that uses the average of the derivatives at each step to integrate.
Matrix methods	Gaussian method	A method for solving systems of linear algebraic equations by successive elimination of variables.
	LU decomposition method	Splitting a matrix into a product of lower and upper triangular matrices to simplify solving systems of equations.
Optimisation methods	Least squares method	A method for minimising the difference between observed and theoretical values.
	Gradient descent method	An iterative method for finding local minima of functions by moving in the direction of the anti-gradient.
Discretisation methods	Finite difference method (FDM)	Converts differential equations into difference equations for numerical solution.
	Finite element method (FEM)	Break down complex geometries into small elements for numerical analysis.
Monte Carlo methods	Monte Carlo method	A statistical method for modelling random processes such as noise or temperature fluctuations.
Event-based modelling	Event-based modelling	Dynamic analysis of digital circuits, where the model is updated in response to events (changes in the state of signals).
Methods for analysing frequency response	Fourier method	Converting signals to the frequency domain for spectral component analysis.
	Laplace transform	Analysis of the behaviour of systems in the frequency domain and finding transfer functions.
Symbolic methods	Symbolic computing	Analytical solution of equations or simplification of mathematical expressions describing electronic circuits.
Methods of stability and sensitivity analysis	Resilience and sensitivity analysis	Evaluation of system stability and its sensitivity to changes in parameters to ensure the reliability and stability of the schemes.
Parallel computing	Parallel computing	Using multiprocessor systems or clusters to speed up computing processes.
Data-driven modelling	Data-driven modelling	Use of machine learning techniques to create models based on empirical data, such as neural networks and regression methods.

Source: compiled by the author

Optimising the performance of EDA is a key factor for effective electronic design. Modern tools use a variety of methods to speed up calculations and reduce memory consumption:

1. Automatic method selection. Most modern EDA tools automatically select the optimal solution method depending on the characteristics of the problem.

2. Hybrid methods combine direct and iterative methods to achieve high efficiency.

3. Simplification of models. To speed up calculations, simplified component models are often used, such as using ideal power supplies instead of real ones or reducing complex circuits to equivalent ones. The choice of model depends on the required accuracy of the results and the range of operating frequencies.

4. Parallel computing for complex models on multi-core processors or GPUs. Typical parallel programming technologies include the Message Passing Interface and the OpenMP standard.

5. Adaptive integration methods. To reduce computational costs, adaptive integration methods, such as the Runge-Kutta method, are used, which automatically select the integration step depending on the rate of change of the variables.

6. Compilation of models. To increase the speed of execution, models can be compiled into machine code, which avoids interpretation overheads.

7. Algorithm optimisation increases the speed of computation through more efficient data structures, reduced operations, and parallelisation.

8. The results are verified by comparing them with other simulators, analytical solutions and experimental data. Formal verification confirms the correctness of the scheme mathematically.

To create accurate models, EDA programs use HDL. In particular, the analysed tools use SPICE, Analysis of Basic Circuits and Devices (AB-CD), Verilog and Very-High-Speed Hardware Description Language (VHDL), each of which has its characteristics and applications. SPICE and AB-CD have traditionally been used for modelling analogue circuits, while Verilog and VHDL have become the standard for describing digital circuits. EDA tools can be used to effectively combine these languages to model mixed analogue and digital systems (Table 2). Figure 2 provides a classification of EDA software based on their functionality and modelling methods. For further analysis, packages widely used in the industry from such companies as LTspice with the software of the same name, MathWorks (MATLAB/Simulink tool), National Instruments (Multisim tool), Labcenter Electronics (Proteus tool), KiCad Developers (KiCad tool), Autodesk/CadSoft Computer (EAGLE tool) were used.

Table 2. Comparing HDL for modelling analogue and digital circuits

<table>
<tr><th>Method</th><th>Description</th><th>Usage</th><th>Connection with other methods</th></tr>
<tr><td>SPICE</td><td>A powerful tool for simulating the behaviour of analogue circuits.</td><td>Analyse voltage, current, power consumption, frequency response and other aspects of analogue circuits.</td><td>It can be used to model analogue components used in digital circuits described by Verilog or VHDL.</td></tr>
<tr><td>AB-CD</td><td>A SPICE extension that focuses on modelling basic electronic components and circuits.</td><td>Modelling simple resistors, capacitors, transistors and other basic components.</td><td>It can be used as an introduction to SPICE or for quick simulation of simple analogue circuits.</td></tr>
<tr><td>Verilog (Hardware Description Language)</td><td rowspan="2">A textual language for describing the behaviour of digital circuits based on the behaviour of components.</td><td>Modelling of logic gates, triggers, memory and other digital components.</td><td>It is not intended for modelling analogue components but can be used in conjunction with SPICE to model mixed analogue and digital systems.</td></tr>
<tr><td>VHDL</td><td>Similar capabilities as Verilog, with some differences in syntax and approach.</td><td>It is not intended for analogue modelling but can be used in conjunction with SPICE to simulate mixed analogue and digital systems.</td></tr>
</table>

Source: compiled by the author based on W.Y. Yang *et al.* (2020) and S.L. Tripathi *et al.* (2022)

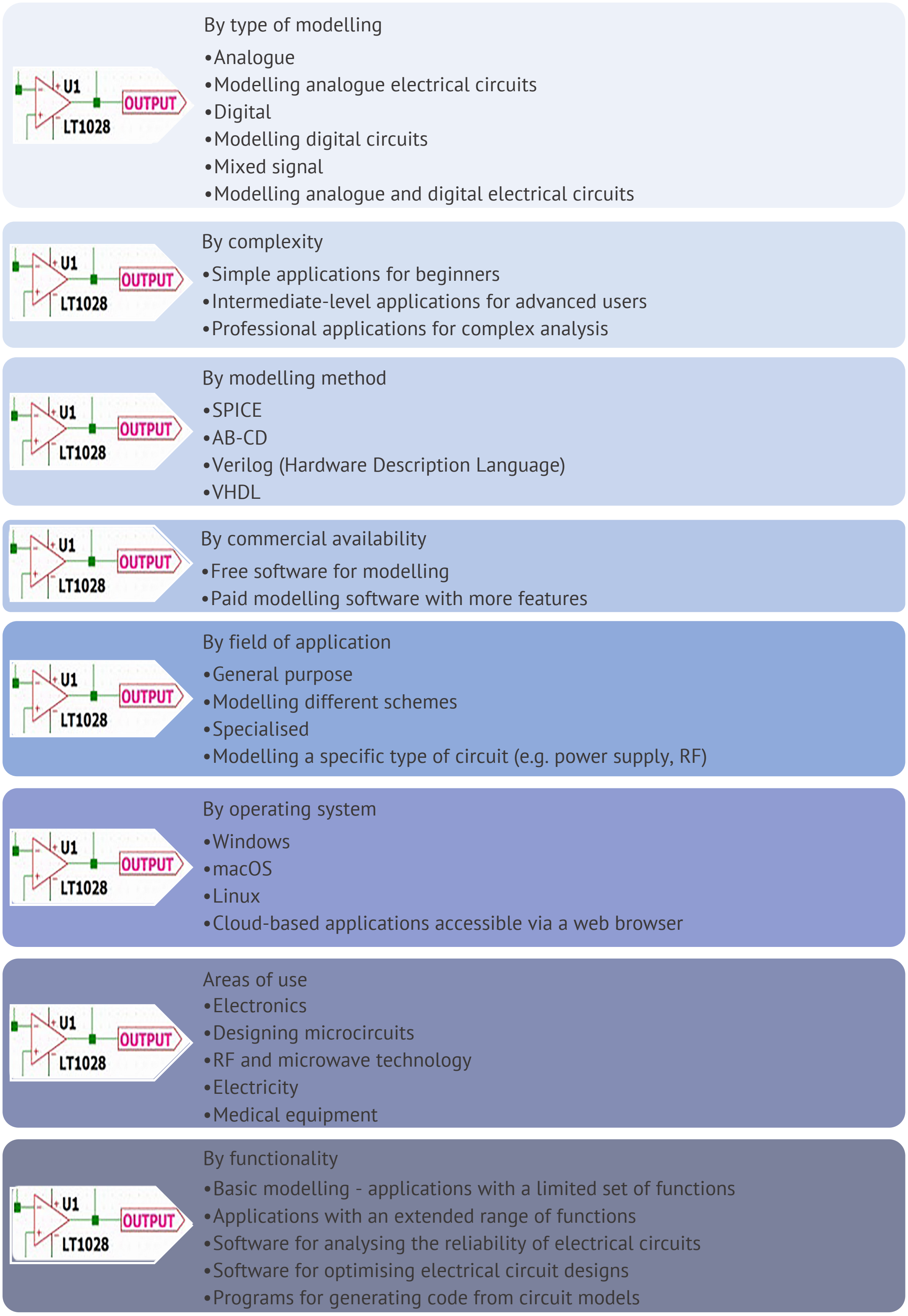

Figure 2. Classification of EDA software

Source: compiled by the author

LTspice, Simulink, and Multisim are powerful tools for modelling electronic circuits, including digital circuits. Each has its set of mathematical models, component libraries, and tools for analysis and optimisation. The differences in the use of methods between LTspice, Simulink and Multisim are due to their architecture and are the result of evolution: each programme has evolved to focus on certain types of tasks and adapt to the needs of different audiences. LTspice focuses on analysing analogue and mixed-signal circuits: and often uses numerical integration methods (Euler, Runge-Kutta, trapezoids), matrix methods (Gauss, LU decomposition), discretisation methods (FDM), and frequency response analysis methods (Fourier, Laplace). Simulink is a more versatile tool for modelling systems of various kinds. It supports a wide range of methods, including numerical integration methods, discretisation methods (FDM, FEM), optimisation methods, Monte Carlo methods, event-based modelling, and frequency response analysis methods, and offers advanced tools for optimising model parameters and performing Monte Carlo analysis due to its focus on system modelling. Depending on the configuration, the FEM discretisation method may be more common in Simulink due to its ability to model complex geometries. Multisim is similar in purpose to LTspice but may have some differences in the set of available methods and their implementation. Depending on the configuration, Simulink and Multisim can have more advanced event-based modelling capabilities for discrete and event-driven systems through the availability of special blocks and tools, as well as for data-driven modelling through integration with machine learning tools.

Although all three programs support some elements of symbolic computing, Simulink may have more advanced capabilities in this area due to integration with other MATLAB tools, depending on the configuration. Modern versions of all three programs support parallel computing to speed up simulations. In the following, the modelling of digital circuits in LTspice, Simulink, and Multisim will be discussed:

1. Mathematical models in these software packages for modelling digital circuits include boolean algebra to describe logical operations (AND, OR, NOT, etc.); switching functions that model the behaviour of digital elements (logic gates) depending on input signals; finite state machines to describe sequential logic circuits (registers, counters and controllers); delay models to account for signal propagation delays in real circuits; noise models to simulate the effect of noise on the operation of digital circuits.

2. Data processing during computer modelling of digital circuits in LTspice, Simulink and Multisim is carried out using the following methods: discrete simulation (e.g., Euler, Runge-Kutta), which allows to model the change of signals in time with a given step; Boolean algebraic logic equations that describe the functioning of digital circuits at the level of logical operations; truth tables used to determine the output values of logical elements depending on the input values; and methods of simplifying logical expressions to minimise the number of logical elements in a circuit.

3. The following methods are used to optimise digital circuits in these applications: automatic circuit synthesis, i.e. creating a circuit based on a given function or truth table, speed optimisation to minimise signal propagation delays, chip area optimisation to reduce the number of logic elements used, and power optimisation to reduce circuit power consumption.

Thus, LTspice and Multisim, which specialise in circuit analysis, use discrete simulation and logic equations to model digital components, but their optimisation capabilities are limited. Simulink, as a universal system modelling platform, offers a wider range of tools for working with digital circuits, including block diagrams and powerful optimisation tools.

The main function of KiCad, EAGLE and Proteus software systems is to perform circuit design and PCB development, and they can also include modelling elements. However, unlike specialised simulators (LTspice, Simulink), these programs usually provide more simplified modelling capabilities focused on quick circuit verification and evaluation of their performance. KiCad focuses on circuit design and PCB development. Simulations in KiCad are limited to simple DC and AC analyses. For more complex simulations, it is recommended to use external simulators such as Ngspice. EAGLE is also focused on circuit design. Simulations in EAGLE can be performed using the built-in simulator, but its capabilities are limited. Proteus provides more extensive simulation capabilities, including microcontrollers, analogue electronics, and digital circuits. This tool uses a proprietary simulator based on a SPICE-like kernel. All three programs use simplified models for basic electronic components (resistors, capacitors, transistors, etc.). These models are typically based on linear differential equations and may not account for complex nonlinear effects. Proteus provides detailed models of popular microcontrollers that can be used to simulate the operation of software. These models are based on CPU emulation and can be quite resource intensive.

Data processing in computer modelling of digital circuits in KiCad, EAGLE and Proteus software is carried out using the following methods: numerical integration for solving differential equations (Euler's method, Runge-Kutta method); matrix methods for solving systems of linear algebraic equations; Boolean algebraic logic equations to describe the functioning of digital circuits at the level of logical operations and truth tables to determine the output values of logical elements depending on the input values. The following methods are used to optimise digital circuits in these applications: simplifying models to speed up simulations; caching the results of previous simulations to speed up subsequent calculations; and using efficient algorithms for computing. One of the significant advantages of

KiCad, EAGLE, and Proteus is their openness to development: With support for plug-ins and scripts, these tools are more adaptable and allow users to create their workflows and integrate applications with other tools.

Available electronic design software such as KiCad, EAGLE and Proteus may have limited capabilities for modelling complex systems and lower accuracy compared to specialised simulators such as SPICE (e.g. LTspice, PSpice) or system modelling environments such as MATLAB/Simulink, with simulation speeds in all cases depending on circuit complexity and computer power. For the accurate modelling of complex electronic circuits, it is recommended to use more specialised software, such as LTspice, and Simulink. The summary results of the comparative analysis are presented in Table 3.

Table 3. Summary results of the comparative analysis of EDA software

Program	A set of models	Accuracy of calculations	Simulation speed	The cost of the licence	Difficulty of use	Additional features
LTspice	Extensive library of standard elements, support for non-linear elements, and the ability to create custom SPICE models.	High accuracy (depending on the component model and simulation settings), recognised in the industry for using the SPICE simulator to solve nonlinear differential equations.	High-speed thanks to an optimised simulator core. Depends on the complexity of the scheme and simulation settings.	Free	Intermediate, requires skills at SPICE basics but has a lot of documentation and examples.	Analysis of DC, AC, transients, noise, spectral analysis.
Simulink	A wide library of blocks for modelling physical and control systems. Supports co-simulation with other applications.	High accuracy due to the use of different numerical methods depends on the choice of integrator and integration step.	High, but may vary depending on the complexity of the model and available computing resources.	Paid (there are academic licences for students and teachers)	High, requires knowledge of mathematical modelling and the MATLAB environment. takes time to master, especially for complex models.	Control systems, digital signal processing, embedded systems development.
Multisim	A large library of components from different manufacturers, support for microcontrollers.	The high accuracy of the SPICE algorithm depends on the choice of component model and simulation settings.	High, optimised for fast electronic circuits.	Paid (trial version available)	Medium, intuitive interface, a large number of training materials.	DC, AC, transient and noise analysis, spectral analysis, and microcontroller simulation.
KiCad	Limited compared to other programs, but it is possible to import models. Library of standard elements, the ability to create custom models.	Satisfactory accuracy for most projects, but inferior to commercial applications. May be insufficient for RF or analogue circuits.	Average, depending on the complexity of the circuit and the simulator settings.	Free, open source	Medium requires knowledge of electronics and simulator setup, takes time to learn, but has a lot of documentation and a support community.	Design of printed circuit boards, and analysis of digital circuits.
EAGLE	Library of standard elements, support for external libraries.	High precision for electronic circuits, widely used in industry.	High, optimised for printed circuit boards.	Paid (there is a free version for small projects)	Medium, the interface may seem complicated for beginners.	Design of printed circuit boards, and analysis of digital circuits.
Proteus	A wide library of electronic components and analogue systems, integrated models for microcontrollers.	High accuracy, especially for microcontrollers in integrated systems.	High, optimised for real-time simulation.	Paid (academic licences are available)	Medium, intuitive interface with lots of examples and documentation.	Microcontroller simulation, embedded systems development, analogue modelling.

Source: compiled by the author

According to the review, LTspice is recognised as a popular choice among engineers and students for designing and modelling electronic circuits due to its affordability, ease of use fast, high-fidelity modelling, extensive component library, and extensive support with numerous online tutorials, forums, and video tutorials. LTspice's user interface is noted for its combination of intuitive text and graphical elements, making it convenient for users with different levels of experience. The software offers fast modelling, high accuracy, and the ability to automate Python tasks with external tools, making it useful for researchers and engineers who need fast and accurate results. At the same time, the software's graphics power is rated as average, which can be a limitation for users who need to visualise complex data or models.

EAGLE is another powerful EDA tool, particularly known for its ease of use and integration with Autodesk Fusion 360. EAGLE is recognised as a complete solution for professional electronics designers, offering detailed documentation and an active community to support users at all stages of their work through a wide range of features, regular updates and access to experts. It supports scripting and PCB design, making it a versatile tool for both hobbyists and professionals. The advantages are cross-platform, an extensive library, powerful graphics and a user-friendly graphical interface. MATLAB/Simulink is recognised by users as a powerful commercial product that provides a wide range of functions, including advanced mathematical modelling and system-level modelling, in addition to circuit simulation. It is preferred in professional and academic environments due to its full functionality and cross-platform compatibility. The Multisim software package has proved to be the most popular among users in the sense that engineers first acquire experience with it during their studies. The choice of educational institutions is based on the convenience of the graphical interface, a large library of components with the ability to add custom components, powerful graphics, and signal visualisation. Users praised the free version for its high functionality for educational needs, active forum, and the completeness and relevance of the user manual. Proteus is a commercial software recognised as a powerful tool for the full cycle of electronic device development. This software package is in demand among engineers and electronics developers since it combines tools for circuit design, PCB tracing and microcontroller simulation in a single environment. The advantages of the KiCad tool include open source, minimal system requirements and cross-platform compatibility. The new HyperText Transfer Protocol libraries feature of the latest KiCad version was noted, which creates opportunities for expanding the component library (the basic version has a limited library). The limitations of KiCad include the lack of built-in tools for signal integrity analysis, such as noise and EMC analysis, which may be important for some projects. The support, completeness and relevance of the documentation received no complaints and were considered sufficient.

DISCUSSION

According to the study, commercial tools MatLab/Simulink and Proteus offer a wide range of functionality, a user-friendly interface, and expert technical support from the developers. They can be beneficial for large companies and professional users who need sophisticated features and reliability. On the other hand, open-source software such as LTspice, KiCad, and Eagle are flexible and transparent, allowing them to be modified by users and adapted to specific needs, making them attractive to enthusiasts, researchers, and small teams. This idea is confirmed by the work of S. Grau *et al.* (2024), who used the open-source code of the KiCad tool to create the KiTorquer plug-in to extend the software's capabilities for designing and optimising magnetic activators on printed circuit boards.

Open-source software typically relies on a community of users for support, which can be less reliable. The communities of commercial and open-source software differ: the former tends to have a smaller community, while the latter is characterised by an active community where users share knowledge, collaborate on projects and help solve problems. The difference in the capabilities of proprietary and open EDA solutions was investigated by A.B. Kahng *et al.* (2022). The author emphasised their complementarity, stressing the importance of combining the strengths of both types of programmes to achieve a synergistic effect and solve complex problems in the field of chip design. Thus, this area of EDA development is recognised as promising and requires further in-depth research. In particular, the development of integrated environments that allow for the effective combination of the capabilities of different tools. An example of such innovations is the OpenROAD platform for automated synthesis of digital integrated circuits, described in J. Chen *et al.* (2020). This tool implements the full flow from the description at the register level to the generation of geometric data for production.

Based on the results obtained, LTspice, Multisim and KiCad can be recommended for beginners, students and enthusiasts, not least because of their affordability. The use of powerful commercial software provides a wider range of functions, including analysis, simulation and optimisation, while free software is usually less functional, but with the possibility of extensions through plug-ins and add-ons. P. Dang & H. Arolkar (2019) analysed software packages, including LTspice, noting its use in areas such as electronics, telecommunications, and medical equipment; he also explored its limitations. The author noted that LTspice does not directly support Verilog and VHDL, as it uses its format for describing components and connections and is not designed to simulate complex digital systems.

However, it is possible to create component models in Verilog-A (a subset of Verilog that is compatible with SPICE) and import them into LTspice. It is also possible to use external Verilog/VHDL simulators to simulate the digital part of the design and then import the results into LTspice to simulate the analogue part.

The results obtained in the comparative analysis showed the high accuracy of the LTspice modelling tool. These data are confirmed by O. Osadchuk *et al*. (2021). Using this tool, the research team developed a mathematical model of a microelectronic frequency humidity converter, which demonstrated high accuracy (error does not exceed 1.8%) when compared with the results of computer modelling and experimental studies. MatLab/Simulink provides a wide range of functions beyond circuit simulation, including advanced mathematical modelling and system-level modelling. It is preferred in professional and academic environments due to its full functionality. O. Tolochko (2020) considered the modelling of electrical and electromagnetic circuits, and electronic devices using Simulink by using virtual blocks of SimPowerSystems libraries. It is noted that users do not need to know the mathematical details of the elements, and the capabilities of the program provide analysis of models in the time, frequency and Laplace space for a complete understanding of the system.

Multisim offers advanced modelling capabilities and a user-friendly graphical interface. It is widely used in education and industry due to its detailed analysis tools, ease of use, and the availability of a free version. Ye. Dosymov *et al*. (2023) noted that the virtual circuit model verification function deepens students' physical perception of electromagnetic processes and provides a better understanding of the relationship between the mathematical model and its real-world analogue. A similar conclusion was reached by L. Li *et al*. (2021) and V. Nerubatskyi *et al*. (2023), who noted the speed, efficiency, intuitive presentation of the characteristics of circuit elements and signal simulation in Multisim when developing even, odd, and fractional frequency dividers for different modes of division.

Proteus has been praised for its ability to simulate microcontroller operation and integrate embedded software, making it useful for embedded system developers. The results highlighted its user-friendly interface and widespread use in both educational and professional contexts. M.A.A. Mareai (2024) focused on its application to electronic circuit modelling, noting how Proteus Simulation uses real-time technology to render the simulation. This allows users to see how the circuit performs in dynamics, which can be useful for debugging and optimising the design. The capabilities of the Proteus tool in teaching and developing skills in designing and modelling electronic circuits are demonstrated in the work of Z. Hashaam (2024), in which a RAM interface was modelled and tested using the Proteus Design Suite to understand the process of data transfer in embedded systems.

A valuable feature of the EAGLE tool in microcontroller PCB manufacturing is the synchronisation of changes in the circuit and on the board, as well as the ability to roll back changes. This makes it easier to work and saves time, provides flexibility and ensures project reliability. These features make EAGLE a versatile tool for both hobbyists and professionals. Similarly to other commercial tools, it offers a complete set of features tailored to the needs of electronics designers, as evidenced by the result of the development of a multilayer PCB by M.N. Islam *et al*. (2022). To design the board using the EAGLE layout editor module, he used an auto-router to generate a trace on the board, and the CAM Job tool to generate production data. After manufacturing, the board was successfully tested for functionality.

The introduction of artificial intelligence and big data analytics into EDA systems is leading to significant progress in the development of electronic devices. Automatic routing and other intelligent tools simplify and speed up the design process, making it more efficient. This is evidenced by the results of the work of M. Venkateswara *et al*. (2020). The paper describes the design and testing of an algorithm for automatically recognising hand-drawn electrical schematics and converting them into LTspice format using machine learning techniques. The implementation of this method is intended to automate the process of checking the circuit and save time and effort for engineers. As noted by M. Cirstea *et al*. (2024), modern EDA tools, by integrating different levels of abstraction, allow engineers to efficiently design complex electronic systems from the top down. By combining functional description and electronic design details, they provide a comprehensive optimisation approach, reducing development time and improving the quality of the final product. G. Cauwenberghs *et al*. (2023) pointed out that the automation of micro/nano circuits and system design is key to advances in information processing, but traditional methods face limitations in size, power consumption, and complexity. To overcome these challenges, new solutions such as machine learning, artificial intelligence, specific accelerators, and nanotechnology are needed, requiring interdisciplinary research and collaboration. Confirmation of expectations regarding AI and big data analytics was provided by the founder of the educational platform techexplorations.com (Generative AI..., 2024). The author noted that artificial intelligence has become a key force in enhancing the functionality and aesthetics of electronic products while minimising production costs and energy consumption. AI development applications and simulation technologies allow engineers to visualise and test products in virtual environments, significantly reducing material waste and supporting sustainability practices. They optimise the placement of components on printed circuit boards, making designs more compact and energy efficient. Sensor data analysis and fault prediction help identify potential problems before they occur, minimising rejects and repair costs.

In the electronics industry, artificial intelligence and EDA systems are also becoming key factors in development. Practical applications of generative AI in electronics include circuit design optimisation, power management, and fault detection. AI-driven CAD applications are leading to significant improvements in design development, verification, and implementation.

CONCLUSIONS

The study systematises and analyses information on available solutions for selecting software for modelling and simulating electrical circuits, addressing user experience. The study contains a general classification of EDA software, a theoretical overview of the physical and mathematical principles underlying the software, and a brief description of their algorithms, methods and key characteristics. The result obtained in the form of a comparative analysis of the key characteristics and functionalities of LTspice, MATLAB/Simulink, Multisim, Proteus, KiCad, and EAGLE software packages is intended to inform users in their choice, helping to optimise work, increase productivity, and save time and money.

The limitations of the study were: quantitative analysis data is limited to EDA developer websites and literature reviews, which may lead to insufficient completeness of data on modelling speed, accuracy of results, and ease of use; qualitative analysis is based on subjective aspects, which may affect the objectivity of the conclusions; qualitative analysis used a randomly available sample of respondents who agreed to participate, which may not represent the entire user community and affect the generalisability of the conclusions. Some programmes may have been excluded from the analysis due to their unavailability at the time of the study. The review of the commercial availability of software may not be complete due to constant changes in the software market and the emergence of new products. The use of EDA developers' websites to collect information may lead to bias, as these sources may not provide objective information about the shortcomings of their products.

Future research should focus on expanding and more detailed selection of programme evaluation criteria to cover all aspects of the use of EDA tools, monitoring their updates to take into account current technological capabilities. To confirm the results of the analysis of EDA programmes, it is useful to study the impact on engineering practice.

ACKNOWLEDGEMENTS

None.

CONFLICT OF INTEREST

None.

REFERENCES

[1] Andriienko V., & Bondarenko, M. (2018). Robotechnics systems in technical education. In *Abstracts of the 4th international scientific and practical conference "Information Technologies in Education, Science and Technology"* (pp. 44-45). Cherkasy: ChSTU.
[2] Arseniuk, D., & Zinkovsky, Yu. (2023). Model of bridgeless totem pole power factor corrector using wideband semiconductor devices. *Scientific Notes of Tavriya National University Vernadsky*, 34(6), 317-322. doi: 10.32782/2663-5941/2023.6/48.
[3] Bahar, R.I., Jones, A.K., Katkoori, S., Madden, P.H., Marculescu, D., & Markov, I.L. (2020). *Workshops on extreme scale design automation (ESDA) challenges and opportunities for 2025 and beyond*. Washington, DC: Computing Community Consortium.
[4] Cauwenberghs, G., Cong, J., Hu, X.Sh., Joshi, S., Mitra, S., Porod, W., & Wong, H.-S.P. (2023). Micro/nano circuits and systems design and design automation: Challenges and opportunities. *Proceedings of the IEEE*, 111(6), 561-574. doi: 10.1109/JPROC.2023.3276941.
[5] Charig, N. (2021). *The evolution of electronic design automation technology*. Retrieved from https://www.power-and-beyond.com/the-evolution-of-electronic-design-automation-technology-a-95ff88f226bb89c338b6d09047fc6d27/.
[6] Chen, J., Jiang, I.H.-R., Jung, J., Kahng, A.B., Kravets, V.N., Li, Y.-L., Lin, S.-T., & Woo, M. (2020). DATC RDF-2020: Strengthening the foundation for academic research in IC physical design. In *ICCAD '20: Proceedings of the 39th international conference on computer-aided design* (article number 71). New York: Association for Computing Machinery. doi: 10.1145/3400302.3415742.
[7] Cirstea, M., Benkrid, K., Dinu, A., Ghiriti, R., & Petreus, D. (2024). Digital electronic system-on-chip design: Methodologies, tools, evolution, and trends. *Micromachines*, 15(2), article number 247. doi: 10.3390/mi15020247.
[8] Curiac, C.-D., & Doboli, A. (2022). Combining informetrics and trend analysis to understand past and current directions in electronic design automation. *Scientometrics*, 127(10), 5661-5689. doi: 10.1007/s11192-022-04481-9.
[9] Dang, P., & Arolkar, H. (2019). Electronic design automation tool: A comparative study. *International Journal for Research in Applied Science & Engineering Technology*, 7(3), 2138-2144. doi: 10.22214/ijraset.2019.3395.

[10] Dosymov, Ye., Usembayeva, I., Polatuly, S., Ramankulov, S., Kurbanbekov, B., Mintassova, A., & Mussakhan, N. (2023). Effectiveness of computer modeling in the study of electrical circuits: Application and evaluation. *International Journal of Engineering Pedagogy*, 13(4), 93-112. doi: 10.3991/ijep.v13i4.34921.
[11] Generative AI for electronic circuit design – an exploration. (2024). Retrieved from https://techexplorations.com/blog/artificial-intelligence/ai-in-circuit-design/.
[12] Grau, S., Lopez, J.M.D., Roychowdhury, D., Chachowski, J., & Stoll, E. (2024). *Design automation of embedded air coils for CubeSat attitude control*. Retrieved from https://www.researchgate.net/publication/381708730_Design_Automation_of_Embedded_Air_Coils_for_CubeSat_Attitude_Control.
[13] Gubbi, K.I., Beheshti-Shirazi, S.A., Sheaves, T., Salehi, S., Manoj, S., Rafatirad, S., Sasan, A., & Homayoun, H. (2022). Survey of machine learning for electronic design automation. In *GLSVLSI '22: Proceedings of the Great Lakes symposium on VLSI 2022* (pp. 513-518). New York: Association for Computing Machinery. doi: 10.1145/3526241.3530834.
[14] Hashaam, Z. (2024). *Interface of RAM in proteus demonstration*. doi: 10.13140/RG.2.2.14168.06406.
[15] Huang, G., *et al*. (2021). *Machine learning for electronic design automation: A survey*. Retrieved from https://arxiv.org/pdf/2102.03357v2.
[16] Islam, M.N., Alam, M.S., & Haque, M.A.S. (2022). Development of eagle multi-layer printed circuit board with CAD and CAM. *International Journal of Systems Signal Control and Engineering Application*, 14(6), 77-80.
[17] Kahng, A.B. (2022). A mixed open-source and proprietary EDA commons for education and prototyping. In *ICCAD '22: Proceedings of the 41st IEEE/ACM international conference on computer-aided design* (article number 17). New York: Association for Computing Machinery. doi: 10.1145/3508352.3561378.
[18] Li, L., Meng, L., & Wang, F. (2021). Design and simulation of frequency divider circuit based on multisim. *E3S Web of Conferences*, 268, article number 01058. doi: 10.1051/e3sconf/202126801058.
[19] Mareai, M.A.A. (2024). *Basic electronic circuits by proteus simulating software*. Retrieved from https://www.researchgate.net/publication/381260714_Basic_Electronic_Circuits_by_Proteus_Simulating_Software.
[20] Marinova, G., & Bitri, A. (2021). Data analysis environment to study the dynamics in electronic design automation industry. *IFAC-PapersOnLine*, 54(13), 528-532. doi: 10.1016/j.ifacol.2021.10.503.
[21] Nerubatskyi, V., Plakhtii, O., Hordiienko, D., Philipjeva, M., & Bagach, R. (2023). Improving of simulation accuracy of transient processes and calculation of power losses of semiconductor converters in the NI Multisim program. *Information and Control Systems at Railway Transport*, 28(2), 22-35. doi: 10.18664/ikszt.v28i2.283312.
[22] Osadchuk, O., Krylyk, L., Zviahin, O., & Osadchuk, Ya. (2021). Mathematical model of a microelectronic humidity transducer with a humidity-sensitive resistive element. *Scientific Notes of Tavriya National University Vernadsky*, 32(1), 175-182. doi: 10.32838/2663-5941/2021.1-2/28.
[23] Semenikhina, O., Drushlyak, M., Lynnyk, S., Kharchenko, I., Kyryliuk, H., & Honcharenko, O. (2020). On computer support of the course "fundamentals of microelectronics" by specialized software: The results of the pedagogical experiment. *TEM Journal*, 9(1), 309-316. doi: 10.18421/TEM91-43.
[24] Shamonia, V., Semenikhina, O., Proshkin, V., Lebid, O., Kharchenko, S., & Lytvyn, O. (2019). Using the Proteus virtual environment to train future IT professionals. *Educational Dimension*, 1, 181-198. doi: 10.31812/educdim.v53i1.3842.
[25] Tolochko, O. (2020). *Application software packages for PC MATLAB, SIMULINK, SimPowerSystems*. Kyiv: National Technical University of Ukraine "Igor Sikorsky Kyiv Polytechnic Institute".
[26] Tripathi, S.L., Saxena, S., Sinha, S.K., & Patel, G.S. (2022). *Digital VLSI design and simulation with Verilog*. London: John Wiley & Sons. doi: 10.1002/9781119778097.
[27] Venkateswara, M., Rao, Hemanth, K., Razia, S., & Kumar, K.R. (2020). Synthesizing of hand-drawn electrical circuits using machine learning techniques. *International Journal of Emerging Trends in Engineering Research*, 8(9), 5523-5529. doi: 10.30534/ijeter/2020/100892020.
[28] Yang, W.Y., *et al*. (2020). *Electronic circuits with MATLAB, PSpice, and Smith chart*. London: John Wiley & Sons.

Дослідження ефективності програм для моделювання та симуляції електричних схем та їх порівняння за точністю, швидкістю та зручністю використання

Лариса Васецька

Кандидат технічних наук, доцент
Луганський національний університет імені Тараса Шевченка
36000, вул. Івана Банка, 3, м. Полтава, Україна
https://orcid.org/0000-0002-4873-8835

Анотація. Зростаюча складність мікросхем та обмеження мініатюризації роблять процес проектування електроніки складнішим та трудомісткішим. Проблема втрати сили законів Мура спричиняє стрімкий розвиток у сфері проектування та виробництва електронних компонентів. Метою дослідження було надати структуровану інформацію про інструменти проектування електричних схем за функціональними можливостями та доступністю, щоб сприяти оптимізації роботи, підвищенню продуктивності та економії. Виконано теоретичний опис принципів роботи програм для моделювання та симуляції електричних схем, огляд алгоритмів та методів, що використовуються в програмах автоматичного проектування, класифікацію програм за ключовими характеристиками та порівняльний аналіз популярних програмних пакетів: LTspice, EAGLE, MATLAB/Simulink, Multisim, Proteus, KiCad. Аналіз враховував швидкість моделювання, точність результатів, легкість використання, функціональні можливості та доступність, а також досвід користувачів. Рекомендовано вибирати програмне забезпечення залежно від потреб користувача, складності проекту та бюджету, з урахуванням досвіду інших користувачів. Для початківців та студентів рекомендуються LTspice, Multisim та KiCad: LTspice – потужний, безкоштовний і простий у використанні, KiCad пропонує відкритий код та активну спільноту, а Multisim підходить для навчання. MATLAB/Simulink, Proteus та EAGLE рекомендовані для професіоналів: Simulink забезпечує потужне моделювання та інтеграцію з іншими продуктами MathWorks; Proteus оптимальний для розробників мікроконтролерних систем; EAGLE підходить для інтеграції з системами автоматизованого проектування та автоматизації. Практична цінність результатів дослідження полягає у створенні рейтингу EDA-програм за критеріями функціональності та продуктивності, покликаного надати користувачам допомогу у виборі на основі їхніх потреб

Ключові слова: інструменти для проектування електроніки; аналіз схем; бібліотека компонентів; підтримувані стандарти; інтерфейс