

# eSim: An Open-Source EDA Tool for Education

Varad Patil

Electrical Engineering  
SGGSIE&T

Nanded, India

varadpatilofficial@gmail.com

Roshan Binu Paul

Elec. & Comm. Engineering  
MITS

Kerala, India

roshanbinu2002@gmail.com

Sumanto Kar

Indian Institute of  
Technology Bombay

Mumbai, India

sumantokar@iitb.ac.in

Kannan M. Moudgalya

Indian Institute of Technology Bombay  
Mumbai, India

kannan@iitb.ac.in

**Abstract**—eSim is an open-source Electronic Design Automation (EDA) tool designed for circuit design, simulation, and analysis [1]. Developed with a focus on providing an accessible and cost-effective solution for engineers, students, and hobbyists, eSim integrates various open-source tools to offer a comprehensive platform for analog and digital circuit design. This paper explores the major capabilities of eSim, its workflow for analog and digital design, [2] and presents case studies demonstrating its practical applications. Along with its capabilities, there are free audio-video tutorials available called Spoken Tutorials [6] through which the users can learn the software very easily

## I. INTRODUCTION

Electronic systems are vital to our daily lives, and the intricacy of integrated circuits is rapidly increasing, with transistor counts growing significantly over time. This development highlights the growing need for advanced CAD tools to manage and design these elaborate electronic systems efficiently. While many commercial tools such as OrCAD [22] provide comprehensive functionalities for designing, laying out, and simulating circuits, their high costs can be a major barrier, particularly for educational and research purposes. Open-source solutions like Ngspice [16] and kTechLab [21] offer circuit simulation and design capabilities, while KiCad [17] and freePCB [19] handle PCB layout. Thus, there is a pressing need for affordable, integrated tools that can efficiently combine these functions for both educational and research purposes.

In this paper, we present eSim, an innovative open-source EDA software that merges circuit design, simulation, and combines PCB layout functionalities into one unified platform. Designed for ease of use and affordability, eSim meets the demand for a complete and user-friendly solution. It supports various operating systems, including Ubuntu Linux, Microsoft Windows and leverages technologies including Python, KiCad, Ngspice, and Scilab [23] (version 5.4 or later). A standout feature of eSim is its Mini Circuit Simulator, based on Scilab, which offers in-depth circuit equations at each simulation stage, thereby enhancing the tool's analytical capabilities.

“By allowing designers to utilize open-source tools such as KiCad [17] for PCB design and Ngspice [16] for simulation, eSim provides a robust platform that fosters innovation while minimizing costs. This paper will discuss the structure and capabilities of eSim, how it supports the complete EDA workflow, and its potential applications in both academic and professional environments [5].

## II. MAJOR CAPABILITIES

### A. Schematic Capture

eSim provides an intuitive interface for drawing and editing circuit schematic. [2] It includes a library of components and supports custom component creation.

### B. Simulation

eSim integrates with Ngspice for analog and digital simulations, including transient, AC, DC, and noise analysis [2], and with KiCad for PCB design [17]. It offers a comprehensive toolchain for circuit development with strong educational support and active community involvement.

### C. PCB Design

eSim integrates with KiCad for PCB design, enabling efficient schematic-to-PCB translation and supporting multilayer designs with advanced routing for complex circuits.

### D. Digital Simulation

eSim includes a Mini Circuit Simulator built on Scilab, offering step-by-step circuit equations. It also supports digital simulation for testing and validating digital designs, ensuring accurate performance analysis before hardware implementation.

### E. Import and Export

eSim supports importing and exporting various formats, including SPICE netlists and PCB layouts, ensuring compatibility with other EDA tools and seamless integration into diverse design workflows.

### F. KiCad to Ngspice

eSim facilitates the generation of netlists from KiCad schematics and includes an Ngspice console for parameter configuration. This feature provides detailed control over simulation parameters, allowing users to tailor the analysis to achieve specific outcomes

### G. OpenModelica Interface

eSim can be interfaced with a modelling software called OpenModelica. This interface is designed to convert SPICE netlists into Modelica format.

### H. Schematic Converters

eSim can also be integrated with Schematic Converters where the users can convert schematics made in other proprietary software to eSim and then run the simulation in eSim.

### I. Integration with SKY130 and other PDKs

eSim has been integrated with the SKY130 primitive libraries and with Ngspice in the backend, it can easily be in-

tegrated with other Open Source PDKs, thus ensuring process level design and simulation possible.

### III. LEARNING eSIM THROUGH SPOKEN TUTORIALS

eSim can be easily learned through free Spoken Tutorial videos [6], designed to teach Open Source Software. These 10-15 minute tutorials cover everything from installation to usage, enhancing job opportunities and providing a seamless learning experience.

### IV. ANALOG DESIGN WORKING AND FLOW BRIEF

#### A. Design Entry

Users initiate the design process by constructing the circuit schematic using eSim's schematic capture tool. Components are either selected from the existing library or custom-created if they are not available.

#### B. Simulation Setup

Upon finalizing the schematic, users proceed to configure the simulation parameters. This involves selecting the appropriate analysis type (e.g., transient, AC) and specifying sources and initial conditions[8].

#### C. Running Simulations

After converting the schematic into a SPICE netlist, Ngspice executes simulations, performing analyses like transient, AC, or DC. Results are presented visually, enabling users to assess circuit performance by visualizing voltages, currents, and other parameters under various conditions.

#### D. Analysis and Verification

Users evaluate simulation results to ensure the circuit meets design specifications by analyzing voltage, current, and other parameters. If discrepancies arise, they may adjust component values or modify the circuit topology to optimize performance.

#### E. Flowchart

The flowchart illustrates the system's workflow and component interactions, providing a clear overview of the methodology. It enhances understanding by visually representing complex processes and their integration. The system's workflow and component interactions are illustrated, providing a clear & concise overview of the methodology (see Fig. 1).

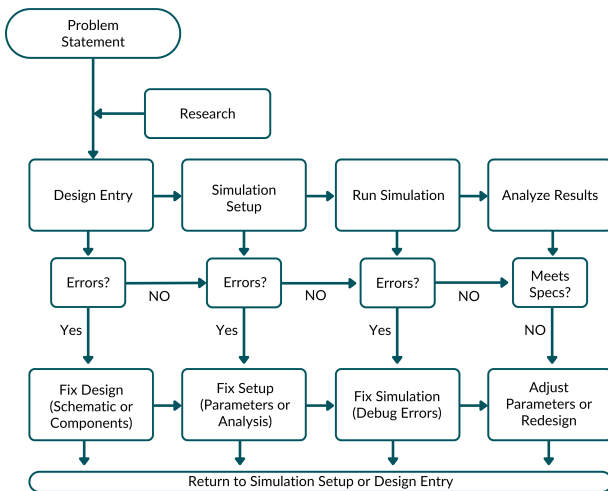


Fig. 1: Preliminary Stages in Simulation Process

#### F. Analog IC (Subcircuit) Example

For this study, the LM342 voltage regulator was used to design and simulate a 5V regulator circuit for comparing eSim with proprietary EDA tools. The LM342 was preferred due to its consistent reliability, providing a solid benchmark for simulation accuracy. A subcircuit was created and is intended to be added to the eSim library, based on the datasheet specifications. [11]. The subcircuit feature enhances modular designing and design reuse. Nesting of the subcircuits are also allowed thus featuring seamless design experience Fig. 2 shows the LM342 test circuit, Fig. 3 show the internal subcircuit, and Fig. 4 Python shows the simulation plots of the of LM342.

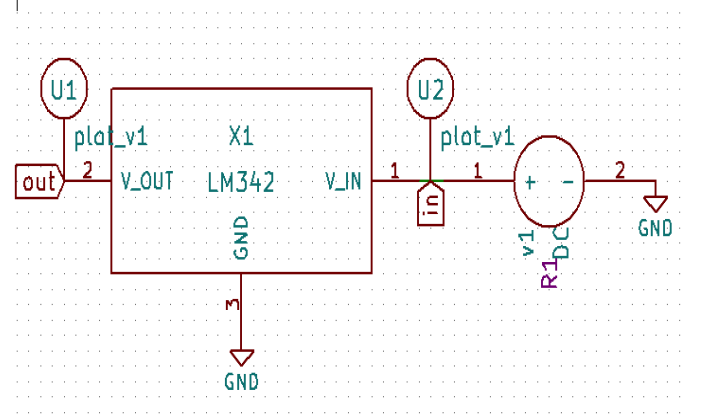


Fig. 2: Test Circuit LM342

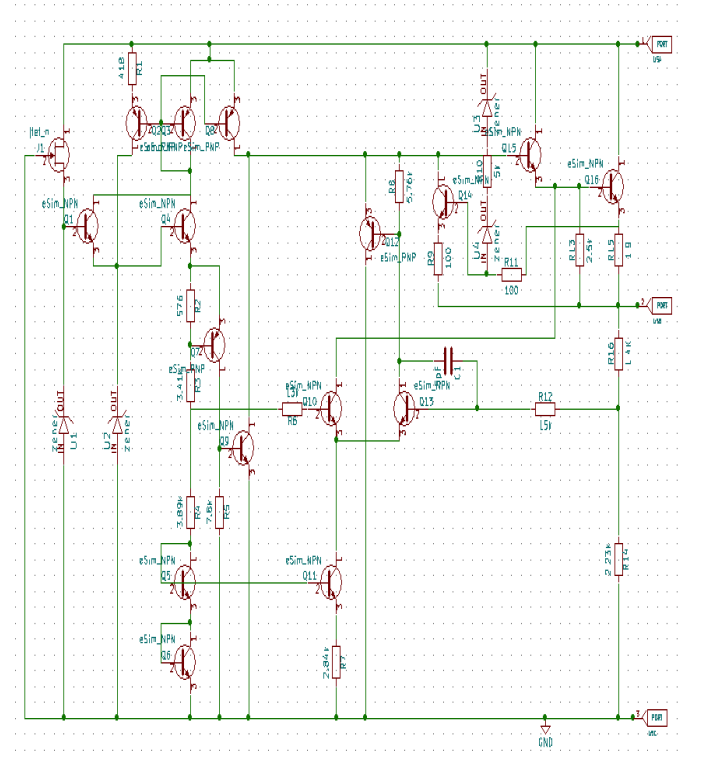


Fig. 3: Sub Circuit LM342

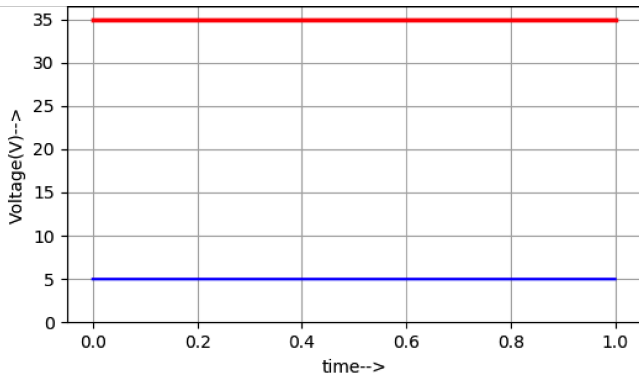


Fig. 4: Python Plotting of Input & Output Waveform for LM342

## V. MIXED-SIGNAL SIMULATION WORKING & FLOW BRIEF

Efficient mixed-signal [2] circuit design and simulation are critical for accelerating innovation in various domains. The framework supports a wide range of circuit topology and simulation parameters, enabling comprehensive analysis and optimization. Additionally, eSim provides an extensible platform for developing custom simulation algorithms and models, fostering innovation in circuit design methodologies.

### A. Conversion of modules

Makerchip is a browser-based IDE used for simulating Verilog, System Verilog, and TL-Verilog files. It provides a platform for initial verification of digital designs. Verilator, an open-source tool, converts Verilog [10] code into C++ objects, which can then be integrated into eSim. This conversion is crucial for mixed-signal simulation, enabling the combination of digital Verilog modules with analog components in eSim.

### B. Design Entry

Mixed-signal circuit design encompasses both analog and digital components. Users begin by developing the schematic, incorporating both types of components as required. This involves placing analog and digital elements, which are either obtained directly or created through conversion, to build the complete circuit design.

### C. Simulation Setup

The simulation setup for mixed-signal [4] designs includes defining the interaction between analog and digital components. This often involves setting up voltage levels for digital signals and specifying how digital control signals interact with analog circuitry.

### D. Running Simulations

eSim employs Ngspice for mixed-signal simulations, effectively processing the netlist to perform co-simulation of both analog and digital signals. This integration ensures seamless handling of mixed-signal interactions. The simulation results provide comprehensive time-domain waveform, capturing both analog voltages and digital logic levels with precision.

### E. Analysis and Verification

Users analyze the mixed-signal [9] simulation results to ensure proper interaction between analog and digital compo-

nents. This step may include verifying signal integrity, timing, and overall functionality of the mixed-signal design.

## VI. CASE STUDY

**Academic Institutions:** Several students from different academic institutions [8] have effectively integrated eSim into their curriculum, yielding notable benefits. By incorporating eSim, these institutions have enhanced their educational offerings in electronics and electrical engineering. feedback from users

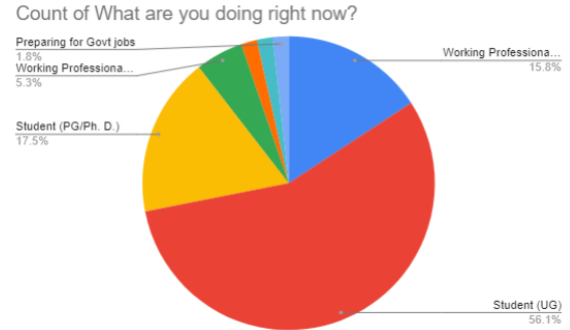


Fig. 5: A statistics showing how people got benefited by using eSim and participating in Marathons [4] [9] [10]

- 1) **Enhanced Learning Experience:** eSim provides students with hands-on experience in circuit design and simulation, bridging the gap between theoretical knowledge and practical application. The tool's user-friendly interface and robust simulation capabilities help students understand complex concepts more effectively.
- 2) **Cost-Effective Solution:** As an open-source EDA tool, eSim offers a cost-effective alternative to proprietary software, making advanced design and simulation tools accessible to a broader range of students and institutions.
- 3) **Improved Practical Skills:** Integration of eSim allows students to work with real-world design and simulation scenarios, acquiring hands-on skills that are directly relevant to industry needs.
- 4) **Flexible and Comprehensive Tool:** eSim supports a wide range of simulation types, including analog, digital, and mixed-signal simulations, providing students with a versatile tool that can be used across various courses and projects.
- 5) **Educational Impact:** eSim provides a hands-on learning experience for students, promoting a deeper understanding of circuit design and simulation.

### A. Digital Circuit: RISC-V Processor

In this work, we present the design and simulation of a 32-bit single-cycle processor based on the RISC-V instruction set architecture, implemented in Verilog. The RISC-V ISA, known for its simplicity and modularity, operates using 32-bit instructions and is structured around six key instruction formats: R-type, I-type, S-type, B-type, U-type, and J-type. The R-type instructions perform operations across three registers, making them fundamental to arithmetic and logic operations. In contrast, the I-type, S-type, and B-type instructions handle two

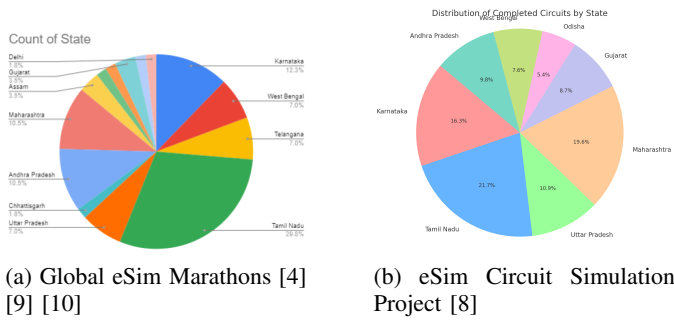


Fig. 6: A categorization of participants from different states across India

registers and incorporate a 12-bit immediate value for load, store, and branching operations, respectively. Additionally, the U-type and J-type instructions leverage a 20-bit immediate, enabling efficient address calculation and jump functionalities.

### 1) Design of RISC V

Arithmetic Logic Unit (ALU), Design an ALU that can perform a subset of the ALU operations of a full Processor ALU. The ALU will generate a 32-bit output that we will call 'Result' and an additional 1-bit flag 'Zero' that will be set to 'logic-1' if all the bits of 'Result' are 0. The different operations will be selected by a 3-bit control signal called 'ALUControl'.

### 2) Control Unit

The control unit plays a critical role, generating control [12] signals according to the opcode and function fields obtained from the instruction (Instr[31:25], Instr[14:12], and Instr[6:0]). While the opcode provides the majority of the control information, the function fields further specify the exact operation to be performed.

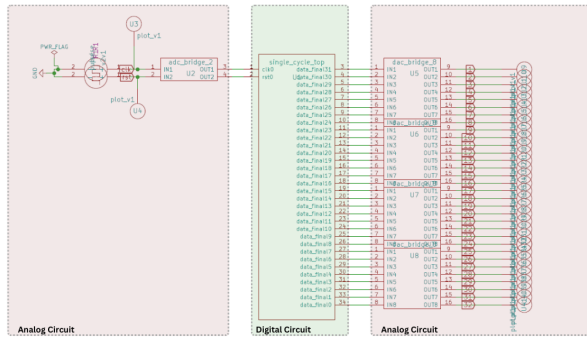


Fig. 7: Mixed Signal Schematic RISC V

### 3) Simulation of RISC V

The RISC-V Single Cycle Top module is simulated initially using Makerchip, followed by the incorporation of additional dependencies for model creation in NgVeri and implementation in eSim. The design inputs include clock (clk) and reset (rst) signals, with ADC and DAC converters employed for signal interfacing between analog and digital domains. [1]

The final output is observed on the 'Result' wire from the multiplexer, with signal plots used for validation.

### 4) Simulation

To start the simulation of the model, an instruction file is included. As an example, we use the instruction 0062E3B3, which specifies the logical OR operation between the values 5 and 4. This operation successfully outputs the expected result of 5 [13].

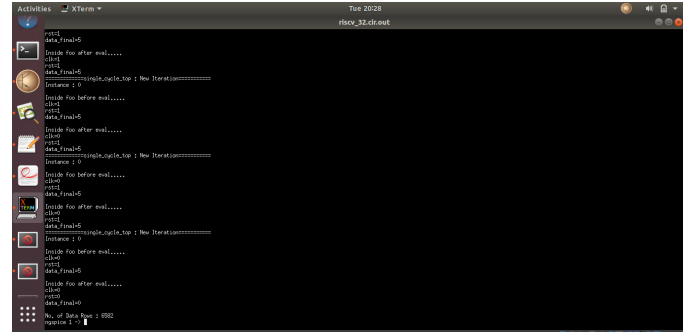


Fig. 8: Simulation RISC V Operation

### 5) Analysis and Verification

The figure 3.2.4 shows the correct execution of the instruction. The seamless integration of eSim with Ngsipice [16] provides designers with a powerful platform for in-depth waveform analysis, enabling precise verification of circuit functionality and performance metrics. Through this analysis, designers can confirm that the implemented designs adhere to the required specifications and operate as intended, ensuring reliability and correctness in the final output. [13]

## VII. CONCLUSION

In conclusion, eSim stands out as a robust and versatile open-source Electronic Design Automation (EDA) tool, offering comprehensive support for both analog and digital circuit design. Its strength lies in the seamless integration of various open-source tools, providing a holistic and cost-effective solution for circuit design and simulation needs. The user-friendly interface of eSim, coupled with its comprehensive range of features, makes it an excellent platform for a wide range of applications, from educational settings and academic research to professional design projects. The case studies presented in this paper highlight the practical applicability and reliability of eSim in real-world scenarios. These examples demonstrate how eSim can be effectively utilized to tackle complex design challenges across different domains of electronic engineering. By offering a powerful yet accessible toolset, eSim not only facilitates the design process but also promotes innovation and learning in the field of electronics. As an open-source solution, eSim contributes significantly to the democratization of electronic design tools, making advanced EDA capabilities available to a broader audience. This accessibility, combined with its robust performance, positions eSim as a valuable asset in the EDA landscape, capable of meeting the evolving needs of both novice learners and experienced professionals in the dynamic field of electronic circuit design.

## ACKNOWLEDGEMENT

The authors are deeply grateful to the FOSSEE (Free/Libre and Open Source Software for Education) project for their invaluable support and resources, which have significantly contributed to the success of this research. Additional gratitude is expressed to the National Mission on Education through ICT (NMEICT) for their continuous efforts in promoting technology-driven education and for their support of initiatives that have enabled this work

## REFERENCES

- [1] M. K. Morais, "Teaching Power Electronics with the Aid of Open Source Simulation Tool eSim," in *2020 IEEE Bombay Section Signature Conference (IBSSC)*, Mumbai, India, 2020, pp. 158-162.
- [2] R. Paknikar, S. Bansode, G. Nandihal, M. P. Desai, K. M. Moudgalya, and A. Jha, "eSim: An Open Source EDA Tool for Mixed-Signal and Microcontroller Simulations," in *2021 4th International Conference on Circuits, Systems and Simulation (ICCSS)*, Kuala Lumpur, Malaysia, 2021, pp. 212-217.
- [3] A. Kumar, K. Ghosh, S. Kar, and R. Paknikar, "Design of 4-bit servo tracking type ADC using Sky-Water SKY130 PDK and eSim," in *2023 International Conference on Artificial Intelligence and Applications (ICAIA) Alliance Technology Conference (ATCON-1)*, Bangalore, India, 2023, pp. 1-3.
- [4] S. Kar et al., "Mixed Signal Simulation Marathon for Education and Employment," in *2022 International Conference on Electrical, Computer, Communications and Mechatronics Engineering (ICECCME)*, Maldives, Maldives, 2022, pp. 1-6.
- [5] FOSSEE, IIT Bombay, "eSim EDA Tool Source Code," 2022, [Online]. Available: <https://github.com/FOSSEE/eSim>. [Accessed: Sep. 19, 2024].
- [6] Spoken Tutorial, IIT Bombay, "eSim Tutorials," 2022, [Online]. Available: <https://spoken-tutorial.org/>. [Accessed: Sep. 19, 2024].
- [7] K. M. Moudgalya, "Results of Scilab Toolbox Hackathon," 2022, [Online]. Available: <https://www.linkedin.com/pulse/results-scilab-toolbox-hackathon-kannan-moudgalya/>. [Accessed: Sep. 19, 2024].
- [8] eSim Team, "Completed eSim Circuit Simulations," 2024, [Online]. Available: <https://esim.fossee.in/circuit-simulation-project/completed-circuits>. [Accessed: Sep. 19, 2024].
- [9] FOSSEE, IIT Bombay, "Mixed Signal Circuit Design and Simulation Marathon - Completed Circuits," 2022, [Online]. Available: <https://esim.fossee.in/mixed-signal-design-marathon/download/completed-circuit>. [Accessed: Sep. 19, 2024].
- [10] K. M. Moudgalya, "Mixed Signal Simulation Marathon using eSim and Verilog," 2022, [Online]. Available: <https://www.linkedin.com/pulse/mixed-signal-simulation-marathan-using-esim-verilog-kannan-moudgalya/>. [Accessed: Sep. 19, 2024].
- [11] alldatasheet.com, "ALLDATASHEET.COM - Datasheet search site for electronic components, semiconductors, and other related devices," 2024, [Online]. Available: <https://pdf1.alldatasheet.com/datasheet-%20pdf/view/8857/NSC/LM342.html>. [Accessed: Sep. 19, 2024].
- [12] Merlidsu, "GitHub - merlidsu/RISCV\_Single\_Cycle\_Core: This repository contains the design files of RISC-V Single Cycle Core," GitHub, 2024, [Online]. Available: [https://github.com/merlidsu/RISCV\\_Single\\_Cycle\\_Core](https://github.com/merlidsu/RISCV_Single_Cycle_Core). [Accessed: Sep. 19, 2024].
- [13] FOSSEE, "FOSSEE Fellowship 2023," 2024, [Online]. Available: [https://static.fossee.in/fossee/reports-2023/fellowship2023/eSim/eSim\\_Digital\\_Abhinav\\_Roshan\\_Bhargav.pdf](https://static.fossee.in/fossee/reports-2023/fellowship2023/eSim/eSim_Digital_Abhinav_Roshan_Bhargav.pdf). [Accessed: Sep. 19, 2024].
- [14] X. Li et al., "iEDA: An Open-source Infrastructure of EDA," in *\*2024 29th Asia and South Pacific Design Automation Conference (ASP-DAC)\**, Incheon, Korea, Republic of, 2024, pp. 77-82, doi: 10.1109/ASP-DAC58780.2024.10473983.
- [15] I. Galán-Benítez, R. Carmona-Galán, and J. M. de la Rosa, "On the Use of Open-Source EDA Tools for Teaching and Learning Micro-electronics," in *\*2024 XVI Congreso de Tecnología, Aprendizaje y Enseñanza de la Electrónica (TAEE)\**, Malaga, Spain, 2024, pp. 1-6, doi: 10.1109/TAEE59541.2024.10605001.
- [16] Ngspice, "Ngspice SPICE Simulator," [Online]. Available: <http://ngspice.sourceforge.net/>. [Accessed: Sep. 19, 2024].
- [17] KiCad EDA, "KiCad EDA Software," 2024, [Online]. Available: <https://www.kicad.org>. [Accessed: Sep. 19, 2024].
- [18] Y. Dilip Save et al., "Oscad: An open source EDA tool for circuit design, simulation, analysis and PCB design," in *2013 IEEE 20th International Conference on Electronics, Circuits, and Systems (ICECS)*, Abu Dhabi, United Arab Emirates, 2013, pp. 851-854, doi: 10.1109/ICECS.2013.6815548.
- [19] FreePCB, "FreePCB Homepage," [Online]. Available: <http://www.freepcb.com>. [Accessed: Sep. 19, 2024].
- [20] Static Free Software, "Static Free Software Homepage," [Online]. Available: <http://www.staticfreesoft.com>. [Accessed: Sep. 19, 2024].
- [21] George John, "KTechLab User Documentation," [Online]. Available: <https://github.com/ktechlab/ktechlab/wiki>. [Accessed: June 11, 2007].
- [22] Cadence OrCAD Solutions, "Cadence OrCAD Products," [Online]. Available: <http://www.cadence.com/products/orcad/pages/default.aspx>. [Accessed: Sep. 19, 2024].
- [23] Scilab, "Scilab: Free and Open Source Software for Numerical Computation," [Online]. Available: <http://www.scilab.org>. [Accessed: Sep. 19, 2024].