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 to provide 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 eSim's major capabilities, its workflow for analog and digital design [2], and presents case studies demonstrating its practical applications. Additionally, free audio-video tutorials, called Spoken Tutorials [6], are available to help users learn the software 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 trend highlights the growing need for advanced CAD tools to manage and design these complex 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, especially 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 applications.

In this paper, we present eSim, an innovative open-source EDA software that integrates circuit design, simulation, and PCB layout functionalities into a single, unified platform. Designed for ease of use and affordability, eSim meets the demand for a comprehensive and user-friendly solution. It supports various operating systems, including Ubuntu Linux and Microsoft Windows, and leverages technologies such as Python, KiCad, Ngspice, and Scilab [23] [24] (version 5.4 or later). A notable feature of eSim is its Mini Circuit Simulator, based on Scilab, which provides detailed circuit equations at each simulation stage, 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 fosters innovation while minimizing costs. This paper discusses the structure and capabilities of eSim, how it supports the complete EDA workflow, and its potential applications in both academic and professional settings [5].

II. Major Capabilities

A. Schematic Capture

eSim provides an intuitive interface for drawing and editing circuit schematics [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 modeling software called OpenModelica. This interface converts SPICE netlists into Modelica format.

H. Schematic Converters

eSim also integrates with schematic converters, allowing users to convert schematics made in other proprietary software into eSim and then run simulations within eSim.

I. Integration with SKY130 and Other PDKs

eSim is integrated with the SKY130 primitive libraries and, with Ngspice in the backend, can easily integrate with other open-source PDKs, enabling process-level design and simulation.

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 Workflow Overview

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 unavailable.

B. Simulation Setup

Upon finalizing the schematic, users configure the simulation parameters, 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, allowing 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 (see Fig. 1).

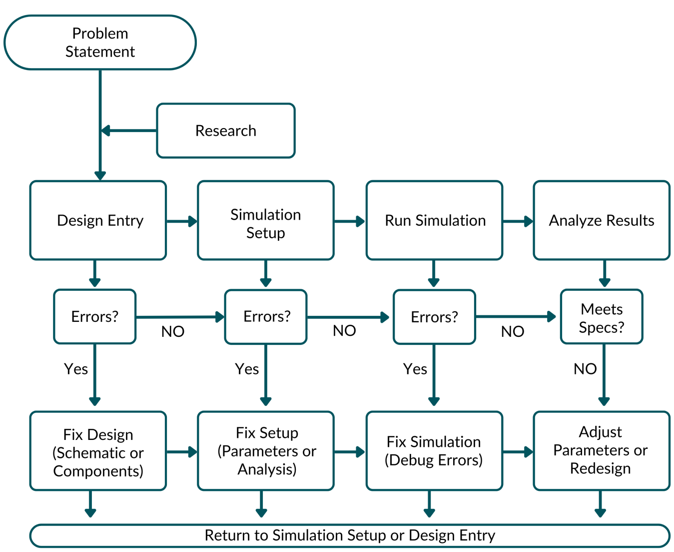


Fig. 1: Stages in Simulation Process in eSim

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 based on the datasheet specifications and is intended for inclusion in the eSim library [11]. The subcircuit feature enhances modular design and reuse, allowing seamless nesting of subcircuits.

Fig. 2 shows the LM342 test circuit, Fig. 3 shows the internal subcircuit, and Fig. 4 shows the Python-generated simulation plots of the LM342.

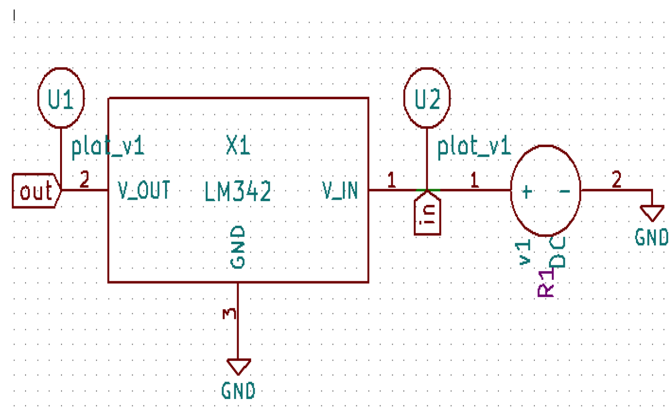


Fig. 2: Test Circuit for LM342 in eSim

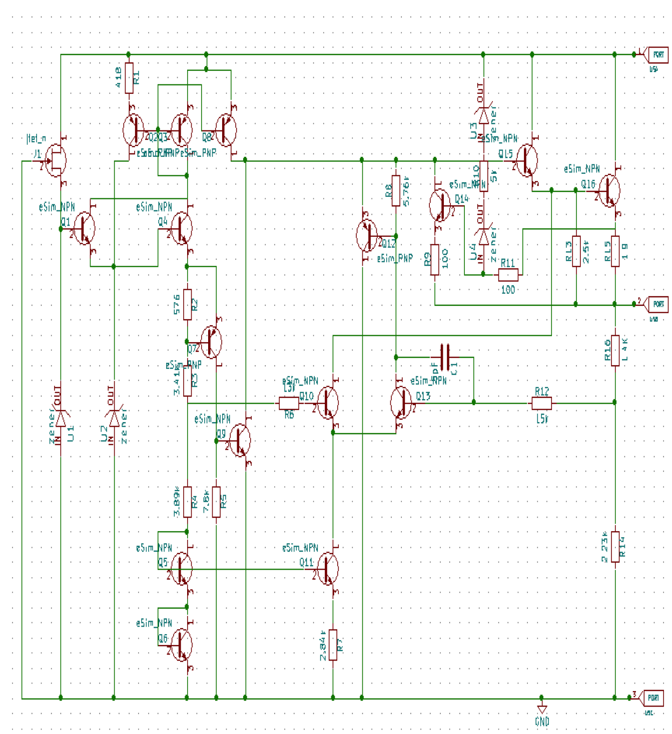


Fig. 3: Subcircuit for LM342 in eSim

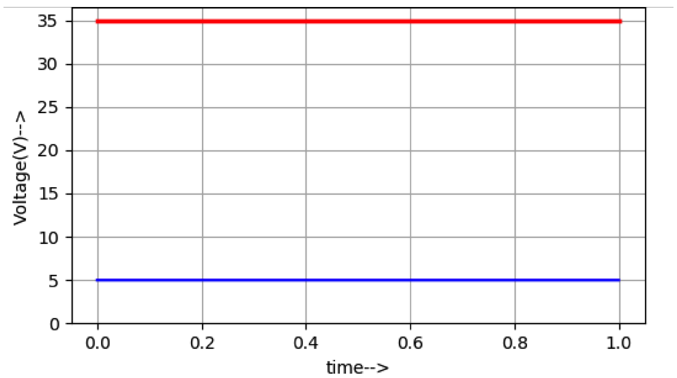


Fig. 4: Python Plot of Input & Output Waveform for LM342

V. Mixed-Signal Simulation Workflow and Overview

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

A. Module Conversion

Makerchip is a browser-based IDE used for simulating Verilog, SystemVerilog, 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 essential for mixed-signal simulation, enabling the combination of digital Verilog modules with analog components in eSim.

B. Design Entry

Mixed-signal circuit design involves both analog and digital components. Users begin by developing a schematic that incorporates both types of components as needed. This process involves placing analog and digital elements, which are either imported directly or created through conversion, to build a complete circuit design.

C. Simulation Setup

The simulation setup for mixed-signal [4] designs involves defining the interaction between analog and digital components. This typically includes setting 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, processing the netlist to enable co-simulation of both analog and digital signals. This integration ensures seamless handling of mixed-signal interactions. The simulation results provide a 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 verify proper interactions between analog and digital components. This step includes assessing signal integrity, timing, and the overall functionality of the mixed-signal design.

VI. Case Study

Academic institutions: Numerous students from various academic institutions [8] have integrated eSim into their curriculum with notable success. By incorporating eSim, these institutions have enhanced their educational offerings in electronics and electrical engineering.

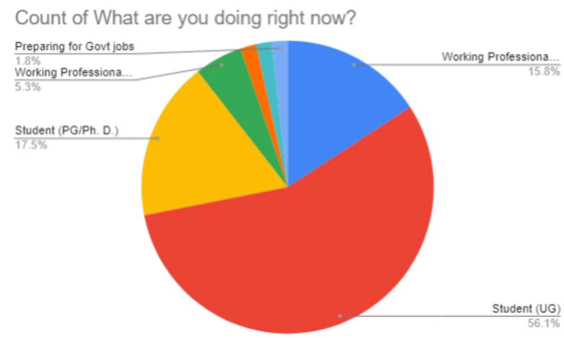
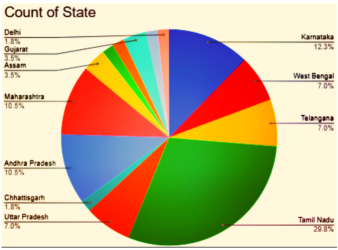


Fig. 5: Statistics showing the benefits of using eSim and participating in Marathons [4] [9] [10].

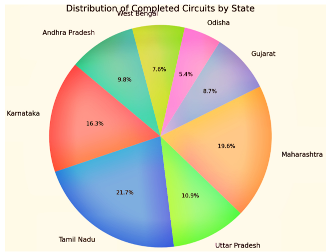
1) **Enhanced Learning Experience:** eSim provides students with hands-on experience in circuit design and simulation, bridging the gap between theoretical knowledge and practical application. Its user-friendly interface and robust simulation capabilities help students understand complex concepts more 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:** The 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 for various courses and projects.
5) **Educational Impact:** eSim delivers a hands-on learning experience for students, promoting a deeper understanding of circuit design and simulation.

A. Digital Circuit: RISC-V Processor

This work presents 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 includes 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 for arithmetic and logic operations. In contrast, the I-type, S-type, and B-type instructions handle two registers and incorporate a 12-bit immediate value for load,



(a) Global eSim Marathons [4] [9] [10].



(b) eSim Circuit Simulation Project [8].

Fig. 6: Categorization of participants from different states across India.

store, and branching operations, respectively. Additionally, the U-type and J-type instructions utilize a 20-bit immediate, enabling efficient address calculation and jump functionalities.

1) *RISC-V ALU Design*

The Arithmetic Logic Unit (ALU) is designed to perform a subset of operations available in a full processor ALU. The ALU generates a 32-bit output called 'Result' and a 1-bit flag 'Zero,' set to 'logic-1' if all bits in 'Result' are 0. Different operations are selected by a 3-bit control signal called 'ALUControl.'

2) *Control Unit*

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

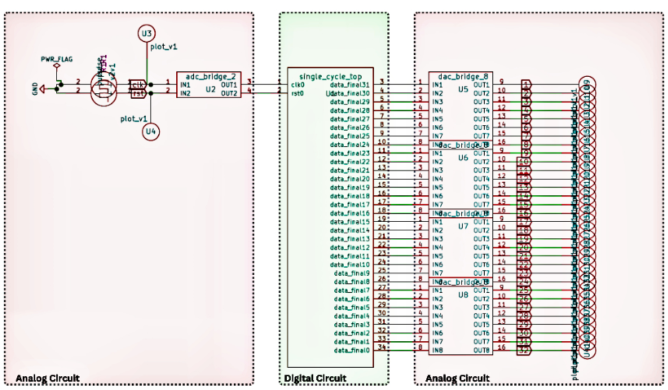


Fig. 7: Mixed Signal Schematic RISC-V.

3) *RISC-V Simulation*

The RISC-V Single Cycle Top module is initially simulated in Makerchip, followed by integration with 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 interfacing signals 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) *Instruction Simulation*

To begin the simulation of the model, an instruction file is included. For example, the instruction 0062E3B3 specifies a logical OR operation between the values 5 and 4, yielding the expected result of 5 [13].

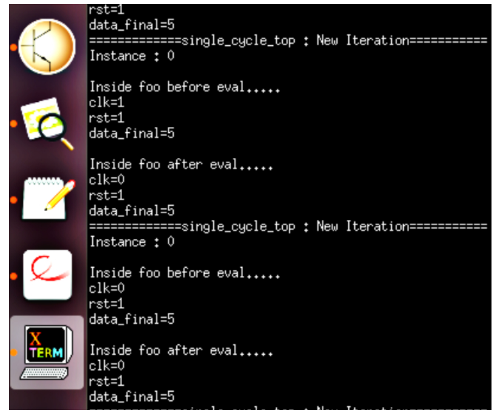


Fig. 8: Simulation of RISC-V Operation.

5) *Analysis and Verification*

Fig. 8 demonstrates the correct execution of the instruction. The seamless integration of eSim with Ngspice [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 confirm that the implemented designs meet specifications and operate as intended, ensuring reliability and correctness in the final output [13].

VII. Conclusion

eSim is 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] Merldsu, ”GitHub - merldsu/RISCV_Single_Cycle_Core: This repository contains the design files of RISC-V Single Cycle Core,” GitHub, 2024, [Online]. Available: https://github.com/merldsu/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. [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 Microelectronics,” 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].

[24] Y. Dilip Save et al., ”Oscad: An open source EDA tool for circuit design, simulation, analysis and PCB design,” 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.