IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, January 2026
Impact Factor: 7.67

A Review on AI-Based Electronic Circuit Simulation Systems

Prof. Manish M. Patil[1] and Amol R. Ahire[2]
Professor & HOD, Department of Electronics and Tele-communication (E&TC) Engineering[1]
Department of Electronics and Tele-communication (E&TC) Engineering[2]
Gangamai College of Engineering, Dhule, MS, India

Abstract: *Artificial Intelligence (AI) has emerged as a transformative force in Electronic Design Automation (EDA), redefining how electronic circuits are designed, simulated, and analyzed. Conventional circuit simulators such as SPICE, Proteus, Multisim, and LTSpice, although accurate, demand extensive manual intervention and advanced technical expertise. This review paper presents a comprehensive analysis of AI-based electronic circuit simulation systems that integrate natural language processing, machine learning, and intelligent graphical interfaces to simplify circuit design. The paper reviews the evolution of electronic simulation tools, discusses the limitations of traditional approaches, and critically examines AI-driven circuit generation and simulation frameworks. The study further highlights educational and industrial applications, identifies existing research challenges, and outlines future directions toward autonomous and intelligent circuit design platforms.*

Keywords: AI-Based Circuit Simulator, Electronic Design Automation, SPICE, Natural Language Processing, Intelligent Simulation, Virtual Laboratories

I. INTRODUCTION

Electronic circuit simulation is a cornerstone of electrical and electronics engineering, supporting design validation, performance analysis, and educational experimentation. Traditional simulation tools rely on manual schematic creation and predefined component libraries, which can be time-consuming and difficult for beginners. With the rapid growth of Artificial Intelligence, simulation environments are evolving toward intelligent, user-centric platforms.

AI-based electronic circuit simulators aim to minimize manual complexity by enabling natural interaction through text prompts and drag-and-drop interfaces. These systems interpret user intent, automatically generate valid circuit topologies, and perform real-time simulations. This review examines such systems and their role in modern electronics education and research.

II. EVOLUTION OF ELECTRONIC CIRCUIT SIMULATION TOOLS

Early electronic simulation relied heavily on mathematical modelling and text-based net lists, with SPICE becoming the industry standard. Although powerful, SPICE-based tools require in-depth circuit knowledge. Graphical simulators such as Proteus, Multiuse, and Allspice improved usability but still demand manual wiring and component selection.

Web-based platforms like Falstad Circuit Simulator and Tinkercad Circuits increased accessibility, particularly in education. However, these tools lack intelligent automation and natural language interaction, limiting their ability to assist users in circuit reasoning and design.

III. LIMITATIONS OF TRADITIONAL SIMULATION SYSTEMS

Despite their accuracy, conventional simulators exhibit several drawbacks:

- Steep learning curve for beginners
- Manual wiring prone to design errors
- Absence of intelligent design assistance
- Limited natural language or intent-based interaction

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, January 2026
Impact Factor: 7.67

- High licensing and system requirements

These challenges restrict rapid prototyping and hinder conceptual learning.

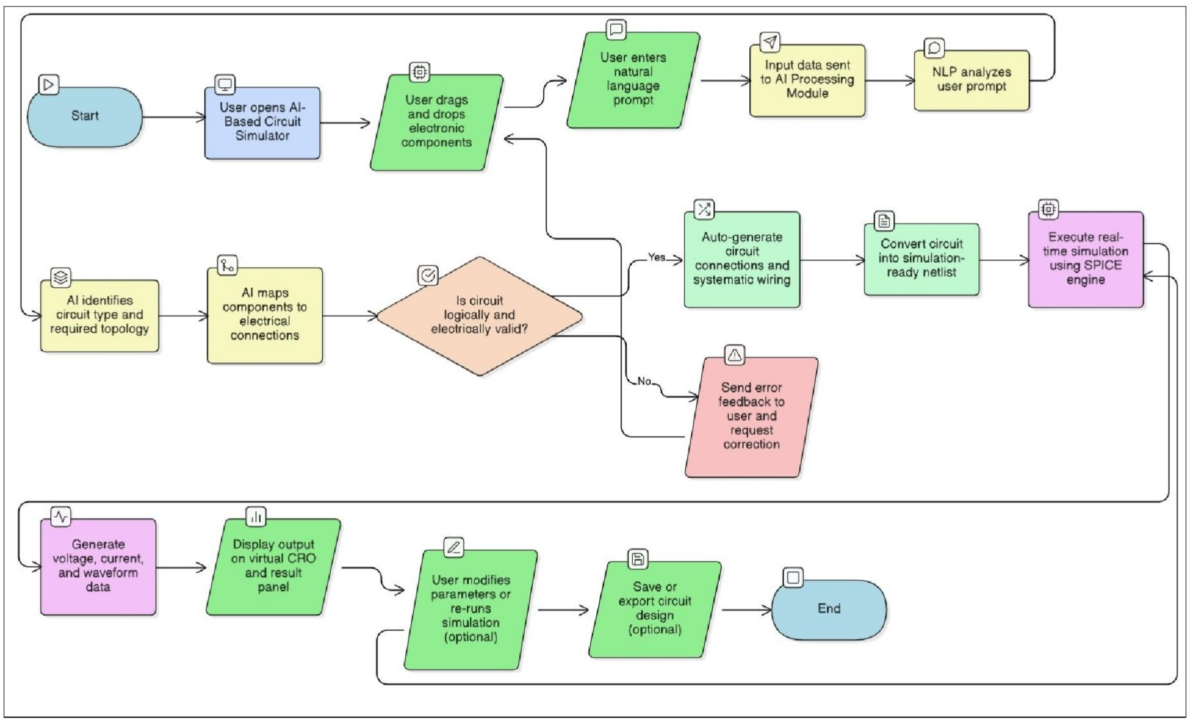
IV. AI-BASED ELECTRONIC CIRCUIT SIMULATION

AI-based circuit simulators integrate machine learning and NLP to automate circuit synthesis. Users can specify circuit functionality in plain language, and the system generates appropriate component configurations. Neural networks and graph-based models learn relationships among electronic components and predict feasible topologies.

The generated circuit is converted into a SPICE-compatible netlist and simulated using engines such as NgSpice or PySpice. Simulation results, including voltage and current waveforms, are visualized using virtual CRO interfaces.

V. SYSTEM ARCHITECTURE

A typical AI-based circuit simulator follows a layered architecture:



- **User Interface Layer** – Drag-and-drop canvas and text input
- **Input Parsing Layer** – NLP-based intent extraction
- **AI Inference Layer** – Circuit topology generation
- **Simulation Layer** – SPICE-based electrical analysis
- **Visualization Layer** – Real-time waveform display
- **Database and Cloud Layer** – Storage and model training support

This modular design ensures scalability and flexibility.

VI. ROLE OF NATURAL LANGUAGE PROCESSING

Natural Language Processing enables users to interact intuitively with the simulator. Transformer-based models identify circuit type, components, and relationships from textual prompts. This capability bridges the cognitive gap between human language and electronic design logic, making circuit simulation more accessible.

VII. EDUCATIONAL AND INDUSTRIAL APPLICATIONS

AI-based simulators enhance electronics education by encouraging experimentation and visual learning. Students can observe real-time circuit behaviour without physical components. In industry, these tools support rapid prototyping, early-stage validation, and design optimization, particularly in IoT and embedded systems.

VIII. CHALLENGES AND RESEARCH GAPS

Despite significant progress, challenges remain:

- Ensuring electrical correctness of AI-generated circuits
- Limited training datasets for analog and mixed-signal designs
- Lack of explain ability in AI decisions
- Integration with real hardware platforms
- Scalability for complex systems
- Addressing these challenges is essential for widespread adoption.

IX. FUTURE SCOPE

Future AI-based circuit simulators may incorporate reinforcement learning, automated fault detection, hardware-in-the-loop simulation, voice-based interaction, and PCB layout automation. Such advancements will move EDA toward fully autonomous design environments.

X. CONCLUSION

AI-based electronic circuit simulation systems represent a significant advancement in EDA by combining artificial intelligence, natural language interaction, and real-time simulation. These systems reduce design complexity, improve accessibility, and foster innovation in education and industry. Continued research will further refine these platforms, shaping the future of intelligent electronic design.

REFERENCES

[1] P. W. Tuinenga, *SPICE: A Guide to Circuit Simulation and Analysis*, Prentice Hall, 1995.
[2] J. Vlach and K. Singhal, *Computer Methods for Circuit Analysis and Design*, Springer, 2012.
[3] IEEE Xplore Digital Library, Research on AI-Assisted Circuit Design.
[4] ACM Digital Library, Intelligent Educational Simulation Systems.
[5] Recent studies on NLP and Machine Learning in EDA.
[6] PySpice Documentation — https://pyspice.fabrice-salvaire.fr
[7]TensorFlow Library — https://www.tensorflow.org
[8] NgSpice Simulation Engine — http://ngspice.sourceforge.net
[9] Falstad Circuit Simulator — https://falstad.com/circuit
[10]5. ReactJS Documentation — https://react.dev
[11]IEEE Papers on AI-Based Circuit Design — IEEE Xplore Digital Library
[12]BERT & GPT NLP Models — arXiv Research Archive

DOI: 10.48175/IJARSCT-30815