

# The Exploration of the Application of Electronic Circuit Simulation Technology in Integrated Circuit Design

Shengjie He\*

Tongji University, Shanghai 200092, China

\*Corresponding author: Shengjie He, 2250323@tongji.edu.cn

**Copyright:** © 2025 Author(s). This is an open-access article distributed under the terms of the Creative Commons Attribution License (CC BY 4.0), permitting distribution and reproduction in any medium, provided the original work is cited.

**Abstract:** With the rapid development of Internet technology, the application of electronic circuit simulation technology is more and more extensive, and now it has been applied to integrated circuit design. Because the electronic circuit simulation technology has high efficiency, flexible and simple application, as well as stable performance, it has shown more and more good application prospects in integrated circuit design. Based on the strong development trend of electronic circuit simulation technology, it will be more and more widely used in daily life in the future, so the research on electronic circuit simulation technology is more and more in-depth. In this paper, the application of electronic circuit technology in integrated circuit design is studied, hoping that the technology can provide a more concise and efficient research and development way for electronic applications.

**Keywords:** Electronic circuit; Simulation technology; Integrated circuit

**Online publication:** March 28, 2025

## 1. Introduction

In the era of information technology, electronic products have become deeply integrated into all aspects of people's production and daily life. As a result, electronic circuit simulation technology has undergone continuous advancements, now providing strong support for the efficient design and development of integrated circuits. Integrated circuit is an important part that affects the function and operation of electronic products. To meet the needs of rapid upgrading of electronic products, integrated circuit design must keep up with the pace of The Times and ensure the development of electronic products. The classification of integrated circuits is mainly based on their density and size. The application of electronic circuit simulation technology in integrated circuit design offers new approaches for advancing electronic products. Therefore, electronic product designers should focus on integrated circuits during the design process, flexibly apply chip-level system concepts to product development, and continuously integrate and optimize circuit functions to achieve higher performance and greater miniaturization.

It is integrated within the chip, which shortens the development cycle of electronic products and provides strong support for their research and development. In this paper, the concept, function, and characteristics of electronic circuit simulation technology are analyzed, and the specific application strategy of this technology in integrated circuit development, project development, and circuit function is discussed. Finally, the future development trend of this technology is analyzed to provide some help for related research.

## **2. Overview of electronic circuit simulation technology**

Electronic circuit simulation technology, based on virtual reality, extends system simulation by relying on advanced computer system technology and requiring robust hardware and software platforms. It reflects the functional role of integrated circuit analysis, utilizing digital modeling and numerical simulation methods to replicate and analyze the operating states of electronic circuits. It can simulate the operation of the integrated circuit and then support the circuit to improve the accuracy. At present, the development of electronic equipment requires not only equipment manufacturing and commissioning but also the identification and timely resolution of problems during the development process. Due to the complexity of this manufacturing and commissioning workflow, the error rate is relatively high<sup>[1]</sup>. At present, the application of electronic circuit simulation technology in the development of electronic products reduces the workload of test work and improves the accuracy of test results.

At this stage, the integrated circuit performance test mainly relies on the tester to carry out. Compared with the manual test, the tester detection not only has high reliability, but also the test is more standardized and safer. The tester can simulate various software application scenarios, identify and correct problems in a timely manner, thereby improving the development efficiency of integrated circuits. This process not only promotes the continuous innovation of integrated circuit design concepts but also ensures the operational stability and functional diversity of integrated circuits, laying a solid foundation for the advancement of electronic components and the design of electronic products. In addition, electronic circuit simulation technology can accurately reflect the actual structure and operational characteristics of electronic circuits, enabling the integration, construction, and simulation of complete circuit systems. This provides strong support for R&D personnel in optimizing integrated circuit design, enhancing design efficiency and accuracy<sup>[2]</sup>.

## **3. Function and feature analysis of electronic circuit simulation software**

### **3.1. The function of electronic circuit simulation software**

There are two kinds of electronic circuit simulation software, one is OrCAD PSpice Designer software, the system is mainly composed of circuit simulation, component processing, and schematic processing of three parts which is based on Spice general language written component-level simulation software to carry out work that is widely used in electronic information retrieval. The software builds a circuit scheme and virtual circuit components, which can accurately describe the circuit components and present the circuit details. The software system can support circuit structure analysis, circuit operation simulation, and provide circuit debugging suggestions<sup>[3]</sup>. However, its analysis and calculation time is long, and the system simulation convergence is poor.

The second kind of electronic software is Saber software. It features advanced simulation technology and adaptable operating conditions, making it not only widely applied in the field of power electronics but also demonstrating excellent performance in areas such as machinery and optoelectronics. The software can be

compatible with hybrid simulation and analyze problems from multiple dimensions. The software can support integrated circuit operation simulation, data visualization analysis, etc., but its shortcomings lie in the low simulation success rate and complex operation<sup>[4]</sup>.

## **3.2. The characteristics of electronic circuit simulation technology**

### **3.2.1. High efficiency**

In the past, the testing of electronic products relied on manual operation, but the test method has more security risks and rough details, so the testing of electronic products under manual operation is more difficult. However, with the application of electronic circuit simulation technology, the testing of electronic products has become more comprehensive and precise<sup>[5]</sup>. Because electronic circuit simulation technology can simulate various scenarios in the operation and application of electronic products, it can also correct the shortcomings of electronic products in time, so it shortens the development cycle of electronic products and provides strong support for the research and development of electronic products.

### **3.2.2. Diversity**

The development of electronic products is more complex, debugging, production, and modification of the links of work may affect the quality of products. Electronic circuit simulation technology can be applied to all aspects of electronic product and integrated circuit development, supporting automatic debugging and providing more accurate modification schemes to promote the design and optimization of integrated circuits and electronic products. At the same time, due to the diverse application scenarios of integrated circuits and electronic products, electronic circuit simulation technology can simulate different application scenarios, provide more effective optimization suggestions for circuit simulation, ensure the quality of electronic products, and extend the life of electronic products<sup>[6]</sup>.

### **3.2.3. Stability**

With the development of information technology, the functions of electronic products are becoming more and more complex and the density of integrated circuits is increasing. Researchers apply chip-level system design technology to the development of electronic products, continuously enhancing the functional complexity of electronic products while significantly improving their operational stability. In the future, electronic circuit simulation technology will play an increasingly vital role in the development of electronic products.

## **4. Electronic circuit simulation technology for the embodiment of electronic application value**

### **4.1. Development of integrated circuits**

Electronic circuit simulation technology has brought revolutionary changes to integrated circuit design and promoted the rapid development of integrated circuit research. With the continuous development of information technology, electronic products have put forward higher requirements for integrated circuit design. As a key component of electronic products, integrated circuits should be able to support the operation of electronic products in complex scenarios. The electronic circuit simulation technology can ensure that the integrated circuit can be used in various working scenarios and meet the actual needs by simulating various working scenarios and debugging the integrated circuit repeatedly. Electronic circuit simulation technology updates the research and development of IC and ensures the function and efficiency of IC<sup>[7]</sup>.

## 4.2. Project development and circuit functions

After the development of an electronic product, it will not be directly mass-produced and put into use but must undergo feasibility testing to ensure that its safety and operational performance meet relevant standards. Only after passing these tests can the product enter the mass production stage and be officially applied. The electronic circuit simulation technology replaces the manual operation of the electronic product feasibility test so that the detection work efficiency is higher and the accuracy is higher. At the same time, the electronic circuit simulation technology can simulate the operation of the product through virtual simulation, so the detection is more comprehensive and efficient. The adjustment of parameters of high temperature, high voltage, and high current will have an important impact on the function of integrated circuits. By setting parameters in the electronic circuit simulation system, the system will simulate the operation of products in special environments and test the limit value of the circuit, which can improve the service ability of the circuit<sup>[8]</sup>. It is evident that applying circuit simulation technology to the testing of electronic products can significantly enhance the scientific rigor and accuracy of the testing process while also shortening the research and development cycle. It can be seen that electronic circuit simulation technology has many advantages in the development of integrated circuits.

## 4.3. Effective verification of circuit functions

Designing an integrated circuit scheme is the first and most crucial step in the entire research and development process of integrated circuits. The design must be scientific and functionally complete, and it must undergo various feasibility verifications to ensure it meets fundamental application requirements. Once the precision of components and circuits in electronic products changes, it will have a certain impact on the function and performance of the product. Therefore, it is important to carry out feasibility tests for integrated circuits. The application of electronic circuit simulation technology can effectively enhance the feasibility verification of electronic applications. By combining electronic circuit simulation technology with traditional circuit feasibility analysis, developers can conduct comprehensive testing of electronic products, promptly identify functional errors or issues that deviate from actual conditions, thereby reducing the likelihood of circuit design failures. This approach also enhances the functional stability of electronic products to a certain extent<sup>[9]</sup>.

## 4.4. Circuit design optimization

Temperature has a great impact on the stability of electronic equipment and the change of temperature can easily cause the change of the internal components of the chip, thus affecting its performance. The application of electronic circuit simulation technology can effectively avoid the above problems. By using simulation technology to replicate different temperature environments, developers can observe circuit changes, analyze circuit characteristics, and make targeted improvements. This process ultimately leads to the development of high-performance electronic devices. Electronic circuit simulation technology can simulate various environments or conditions that are difficult to observe in actual operation, helping designers analyze circuit behavior under extreme conditions and optimize the circuit design accordingly. The reliability of the circuit in practical application is ensured by conducting virtual tests on <sup>the circuit</sup><sup>[10]</sup>.

## 5. The specific application of electronic circuit simulation technology in integrated circuit design

Based on the electronic circuit simulation technology, users can set different parameters in the system according

to the work needs, simulate various operating scenarios, and provide ideas for integrated circuit debugging and design. The integrated circuit operation is simulated and its structure is described to support R&D personnel to optimize the technology. At the same time, through the simulation system, it can also design and process its frequency response, output signal, etc., to study and analyze in the future and optimize the research results. Compared with traditional integrated circuit design methods, the application of electronic circuit simulation technology significantly reduces the time and cost of circuit design and development, while greatly improving product quality and performance. The following is an analysis of the application of electronic circuit simulation technology in integrated circuit design.

## **5.1. PSPICE simulation software**

In practice, Cadence's PSPICE simulation software is widely used in large-scale integrated circuit design. The software can support repeated digital analog mixed simulation of the circuit and provide the corresponding correction suggestions so that the performance of the entire circuit is more perfect. The main function of PSPICE software is to simulate the circuit or text file so that researchers can understand various performance indicators of the circuit according to the results of simulation processing <sup>[11]</sup>.

## **5.2. The specific functions of PSPICE simulation software**

PSPICE simulation software has two basic functions: the basic analysis module (referred to as PSPICE AD) and the advanced analysis module (referred to as PSPICE AA). The basic analysis module primarily includes DC analysis, AC analysis, time domain analysis, and other functions. The advanced analysis module mainly includes several functions such as parameter scanning, temperature analysis, and worst-case analysis. Its simulation function is mainly the following modes:

- (1) Noise analysis simulation refers to the process of analyzing the equivalent output noise and equivalent input noise of a given input signal within a specified frequency range and along a defined calculation path.
- (2) DC bias simulation, in the presence of inductance and capacitor disconnect two cases in the line, is used to determine the steady-state operating point of the line. In the transient process of small interference and transient analysis, it will automatically find the DC balance point so as to provide the necessary initial conditions for the transient analysis, and the model parameters are nonlinearized.
- (3) DC scan analysis refers to analyzing and calculating the DC output characteristics of a circuit as a specific parameter varies within a defined range.
- (4) AC sweep analysis is used to calculate the amplitude-frequency and phase-frequency characteristics of small signals, as well as the input and output impedance of the circuit.
- (5) Parameter scanning analysis involves analyzing the circuit's performance as the parameters of specific components vary according to a defined pattern or rule.

The basic process of simulation using PSPICE software includes the following steps: drawing the circuit schematic, setting simulation parameters, running the simulation, observing and analyzing the simulation results, adjusting the circuit, and modifying the simulation parameters accordingly.

## **5.3. The application of electronic circuit simulation software in integrated circuit design**

### **5.3.1. Modeling classification and simulation process**

Pspice simulation software is commonly used in the design of large integrated circuits. During the circuit design

process, PSpice enables mixed-signal simulation testing, allowing designers to repeatedly modify the circuit based on test results. This iterative process helps to fully ensure that the performance indicators of the designed circuit meet the required design standards. The main function of PSpice simulation software is to simulate circuits and text files, etc., and allow relevant personnel to apply the final simulation results to various parameter data and performance of product circuit design<sup>[12]</sup>.

### 5.3.2. PSpice simulation technology function

#### (1) Electronic circuit simulation function design for electronic products

In the product-oriented LSI design, the PSpice simulation software of Cadence is used to simulate and analyze all kinds of complex circuit performance indicators, to obtain the corresponding circuit data and realize real-time visualization processing. In this way, it is convenient for the circuit designer to observe the simulation results during simulation and analysis. At the same time, the system can also evaluate the circuit design and various performance indicators of the whole system efficiently in a more intuitive way. In short, the simulation software mainly includes two parts: Basic Analysis Module (AD) and Advanced Analysis Module (AD). On the basis of this basic analysis module, the simulation and analysis of the performance of the DC circuit and AC circuit are completed, and the time domain of the entire circuit is comprehensively analyzed<sup>[13]</sup>.

In practical application, the method of noise simulation analysis is used to set the frequency, output equivalent, and input noise level of the test circuit. In addition, PSpice can analyze the bias point of DC circuits and evaluate the short-circuit factor, enabling the calculation of the circuit's operating state under short-circuit conditions or capacitor failure, thus accurately determining the overall condition and reliability of the circuit. On this basis, the transient characteristics of the system are further verified. From the perspective of high-level analysis module, this system mainly includes three modules: worst simulation analysis, temperature analysis, and parameter scanning. Finally, using the scanning function, the DC output characteristics of the product can be displayed. It can also be summed up and judged from the change of electrical parameters in the circuit. The simulation system has powerful functions such as numerical analysis, matrix calculation, scientific data visualization, modeling and simulation of nonlinear dynamic system, etc.

The R&D personnel can apply electronic simulation software to verify the calculation results and apply the system to automatically obtain the results of multivariate equations, differential equations of dynamic circuits, and equations of phases, etc. Electronic simulation experiments can be used to carry out online simulation experiments. Saving time in preparing experimental materials can significantly improve experimental efficiency<sup>[14]</sup>. For example, the application of Multisim to carry out the design of digital circuit board, directly input the graphics of circuit schematic diagram and circuit hardware description language into the system, and carry out online simulation, which greatly improves the work efficiency.

#### (2) Circuit design simulation process

The testing process of PSpice simulation software generally follows these steps: First, the simulation circuit of the electronic product is described, ensuring that all circuit nodes are clearly marked within the system, and simulation parameters are properly set. Second, the output waveform generated by the electronic simulation is recorded, and the simulation results are carefully observed and analyzed to determine whether further debugging of the product circuit is necessary. Finally, through iterative simulation debugging, a complete and optimized circuit design scheme is formed.

Taking a sinusoidal oscillation circuit as an example, the application of electronic circuit simulation

technology in its design can be described as follows: under specific excitation conditions, the transient response of the circuit is calculated, and the resulting transient data is saved in a file with a “.dat” extension. By using the analysis functions within the simulation window, the signal waveform obtained from the simulation can be visually analyzed. During the circuit design simulation experiment, the proposed integrated circuit design scheme is input into the electronic circuit simulation system. Through the circuit input module (Schematics) of PSpice, the operation of the electronic circuit is simulated, and a circuit diagram of the multi-harmonic oscillator is generated. This multi-harmonic oscillator, typically implemented using a 555 timer chip, provides output through its pin 3 (OUT), effectively demonstrating the circuit’s oscillation behavior.

### (3) DC operation mode

The PSpice software system will label and analyze the DC operating point of the integrated circuit design scheme. During this operation, the staff can select the menu bar of the software, click “Open”, and open the function of the DC operating point of the circuit in the pop-up dialog box. Then, the system can realize the automatic analysis of the parameters of the DC operating point of the circuit. The system generates the analysis results and saves them in the output file to support the researchers to query.

### (4) Fourier analysis

Based on the Fourier analysis module in the simulation system, researchers can analyze the DC waveform and harmonic components of the circuit. In the Fourier analysis module, researchers input the fundamental frequency of the integrated circuit, after which the system automatically performs harmonic calculations and saves the analysis results in a file. In this way, when the first spike is the output signal frequency, its digital parameter is 4 KHz, and the second spike’s second harmonic component is 8 KHz, so that the fourth spike is the fourth harmonic component, its value is 16 KHz. Finally, through comparative analysis, it is found that the frequency shown is consistent with the reality.

## **6. The future development trend of the application of electronic circuit simulation technology**

The application of electronic circuit simulation function design technology has opened a new road for electronic applications. It can significantly improve the development efficiency of electronic products, shorten the research and development cycle and reduce costs while ensuring the quality of electronic products through simulation technology. Although the electronic circuit simulation function design technology in the development of computers only accounts for a small part, it is also a part that can not be ignored. Its use on the development and change of electronic applications has played a role in promoting, and this technology is constantly improving and perfecting. The current electrical simulation technology can only be used for the simulation of circuits and computer hardware systems, but not for the digital system of computer CPU. However, in the near future, the simulation technology of electronic circuits will be further improved and perfected to make the algorithm more accurate and to simulate the program of the CPU<sup>[15]</sup>.

CPU program simulation is a form of electronic system simulation. With the continuous advancement and upgrading of circuit simulation technology, the application of simulation technology in CPU program development is expected to be realized in the future, enabling full simulation of program operation states. This will bring transformative changes to the research and development of electronic products. Based on the current development trend of electronic circuit simulation technology, it can be predicted that the future should be along the direction of

shortening the research and development time of electronic products and improving the research and development efficiency. It is believed that in the future, with the continuous improvement and refinement of electronic circuit simulation function design technology, the development methods of electronic products will become increasingly advanced, and development tools will become more efficient, greatly shortening the research and development cycle of electronic products.

The application of electronic circuit simulation function design technology ensures that electronic product research and development personnel can quickly discover the development direction and improvement methods, so the application of this technology has an important role in promoting the development of electronic applications. From the perspective of electronic circuit simulation function design technology, the technology still has a great space for development, after continuous improvement and improvement, the technology will play a more important role in the development of electronic applications, therefore, R&D personnel should continue to explore in the actual work, the technology to further improve, so that electronic circuit simulation technology better serve the People's Daily life.

## **7. Conclusion**

To sum up, as the structure and functions of electronic products become increasingly complex, the functional requirements for integrated circuits are also rising. At the same time, with the rapid replacement and upgrading of electronic products, the research and development of integrated circuits must continuously improve efficiency to keep pace with the demands of modern society. Based on electronic circuit simulation technology, integrated circuit (IC) research and development can simulate circuit operation under various application scenarios, providing strong support for optimizing circuit design concepts and improving R&D efficiency.

As an information tool, electronic circuit simulation technology overcomes the limitations of traditional manual testing, making system testing more standardized and safer. It greatly shortens the research and development cycle of integrated circuits, reduces development costs, and provides strong support for the advancement of electronic product development. It can be said that the application of electronic circuit simulation technology opens a new chapter in China's electronics industry, and promotes the research and development of electronic products to take a big step forward.

In the future, leveraging the advantages of efficient operation, flexible application, and stable performance of electronic circuit simulation technology, the development of electronic products is certain to move toward a faster and more efficient direction. However, electronic circuit simulation technology still has considerable room for development, both in terms of technical improvement and optimization. Its application scope also needs to be further expanded. Continuous support and innovation from relevant research and development personnel are essential to advance this technology, drive the sustained growth of China's electronics industry, and better serve the improvement of people's well-being and quality of life.

## **Disclosure statement**

The author declares no conflict of interest.



## References

- [1] Zhao W, Wei P, 2012, Application of PSpice Simulation Technology in Electronic Circuit Design. *Journal of Shaoxing University of Arts and Sciences (Natural Science)*, 32(09): 25–29.
- [2] Mao J, Shen H, Gu J, et al., 2016, Application of LabVIEW FPGA Real-Time Simulation in Modern Power Electronic Technology Experiment Teaching. *Laboratory Research and Exploration*, 35(09): 108–113.
- [3] Ma G, Wang J, Pan B, 2018, Application Research of Virtual Simulation Technology in Electronic Technology Course Teaching. *Compilation of Scientific Research Results of Research on Teacher Education Capacity Building*, College of Electronic Countermeasures, National University of Defense Technology.
- [4] Wang X, Zhang X, Yin J, et al., 2019, Instrument Availability Evaluation for Nuclear Power Plant Based on Circuit Simulation Technology. *Progress Report on Nuclear Science and Technology in China. Progress Report on Nuclear Science and Technology in China: Proceedings of the 2019 Annual Conference of the Chinese Nuclear Society*, School of Nuclear Science and Engineering, North China Electric Power University.
- [5] Sun D, Zhang X, Zhao T, et al., 2020, Exploration of Electronic Circuit Experiment Teaching Based on Virtual Simulation Technology. *Journal of Higher Education*, 2020(35): 108–111.
- [6] Zhang J, 2020, Exploration on Application of Electronic Circuit Simulation Technology in Integrated Circuit Design. *Electronic Components and Information Technology*, 4(10): 94–96.
- [7] Su J, Liu J, Jin Y, et al., 2020, Research on the Value of Electronic Circuit Simulation Technology in Electronic Application Development. *Paper Making Equipment & Materials*, 49(04): 118–119.
- [8] Liu M, 2024, Analysis on the Function of Electronic Circuit Simulation Technology in Electronic Application Development. *Electronic Components and Information Technology*, 8(01): 76–78 + 82.
- [9] Mo Z, Deng R, Yu H, et al., 2023, Application of Online Virtual Simulation Technology in Electronic Circuit Experiment. *Digital Technology and Application*, 41(6): 68–70.
- [10] Cao T, 2023, Analysis on the Role of Electronic Circuit Simulation Technology in Electronic Application Development. *Electronic Components and Information Technology*, 7(3): 29–31 + 40.
- [11] Ma C, Zhao M, Meng X, et al., 2019, Research on the Role of Electronic Circuit Simulation Technology in Electronic Application Development. *Science and Technology Communication*, 11(11): 120–121.
- [12] Fu N, 2019, Application of Circuit Simulation Software in Electronic Technology Teaching Practice. *Science and Technology Information*, 17(15): 194–195.
- [13] Hu G, 2019, The Role of Electronic Circuit Simulation Technology in the Development of Electronic Applications. *Electronic Testing*, 2019(8): 137–138.
- [14] Luo H, 2019, Application of Virtual Simulation Technology in Electronic Technology Teaching in Secondary Vocational Schools. *Wireless Internet Technology*, 16(4): 151–152+166.
- [15] Zhao Y, Li B, 2019, Application of EveryCircuit Circuit Simulation Software in Electronic Technology Teaching. *Computer Knowledge and Technology*, 15(6): 112–113.

### Publisher's note

Bio-Byword Scientific Publishing remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.