ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

# Technology and application of solar thermal power generation

**Yingqi Yu, Chunyan Wang**

*School of Information Technology, Jiangsu Maritime Institute, Nanjing, Jiangsu, 211170, China*

***Abstract:*** *Diode has become the most widely used device in the field of power electronics because of its unidirectional conductivity. It is very important to study the working principle and model of diode. This article first expounds the forming principle of P-N junction and PIN junction, then respectively the diode of the steady state and transient state model was established, and then separately from the Infineon website to download the IDW100E60 diode model and discrete device based on ideal diode model LTspice simulation, comparison and analysis the differences in the waveform, and the possible reasons for error is given. Finally, buck circuit was built, and the reverse recovery characteristics of the diode model were observed in the actual circuit.*

***Keywords:*** *Diode modeling, ideal device, reverse recovery characteristics*

## 1. Introduction

In the rapid development of power electronics technology, diodes, as semiconductor devices with significant unidirectional conductivity, are widely used in rectification, inversion, switching power supplies, and various protection circuits. Their performance directly affects the efficiency and stability of the circuit. In order to gain a deeper understanding and optimize the performance of diodes in various circuits, it is particularly important to study their working principles and precise modeling. The PN junction and PIN junction are the core structures of diodes, and their formation principles and characteristics directly determine the basic electrical behavior of diodes. [1]

This article aims to systematically study the modeling and simulation techniques of diodes through a combination of theoretical analysis and simulation experiments. Firstly, we will review the basic physical principles of PN and PIN junctions to provide theoretical basis for the subsequent model establishment. Subsequently, based on these principles, steady-state and transient models of the diode were constructed to comprehensively capture its electrical characteristics under different operating conditions. In order to verify the accuracy of the model, this article will adopt a comparative analysis method. On the one hand, LTspice simulation software will be used to simulate the official model of the commercial diode IDW100E60 obtained from the Infineon official website. On the other hand, a diode model will be constructed based on ideal discrete device parameters and simulated under the same conditions. By comparing and analyzing the waveform differences between the two in simulation, explore the possible causes of errors, such as model simplification, parameter approximation, and simulation environment settings.

In addition, in order to be closer to practical applications, this article also applies the established diode model to a typical Buck circuit, observes and analyzes the reverse recovery characteristics of the diode through actual circuit testing, which is crucial for evaluating the performance of the diode in high-frequency switching circuits. Through the above research, not only can we deepen our understanding of the working principle and model characteristics of diodes, but we can also provide strong theoretical support and practical guidance for the design and optimization of power electronic systems.

## 2. Diode Modeling Methods and Simulation Software

The primary methods for device modeling currently include physical-level modeling, ideal modeling, and numerical fitting modeling. Physical-level modeling involves creating a physical equation model based on the physical structure of the device (such as PN junctions, charge storage, MOS capacitance, etc.). This method directly describes the physical phenomena and closely reflects the actual characteristics of the device, though it is also more complex. Ideal device modeling, on the other hand,

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

uses ideal switch components and ideal passive components (such as resistors, capacitors, and inductors) to simulate the actual switching characteristics of the device. This approach is generally based on circuit models, serving as an idealized and simplified version of the physical model. It retains some circuit principles while making circuit simulation more convenient. Numerical fitting modeling involves using mathematical expressions to fit the switching characteristic curves based on the actual switching behavior of the device. This method rarely considers the physical background of the device and instead focuses on the device characteristics as they appear, with the advantages of speed and accurate curve fitting. This method is widely used in the device models provided by most device manufacturers. In this paper, ideal device modeling is selected to facilitate circuit simulation and analyze the diode’s switching process in circuits.

The mainstream circuit simulation software currently includes MATLAB/Simulink, PSIM, and LTspice. Simulink in MATLAB includes circuit simulation capabilities; however, its device models are relatively simple and unsuitable for simulations primarily focused on transient circuit changes. PSIM is a software specifically designed for power electronics and motor control design. Its component library mainly provides ideal components, such as ideal voltage sources and ideal switch models with dual-resistance. It also has a dynamic link function block (DLL module), allowing users to write code in C language, compile it into a dynamic link file, and have PSIM call this file at each simulation step, making it particularly suitable for piecewise numerical fitting models. LTspice is a high-performance SPICE simulation software known for being a standard SPICE simulator, with advantages like a small footprint and high computational speed. In addition to offering ideal devices and built-in system library models, LTspice can import various PSPICE library files, greatly enhancing its usability for simulation needs. In this paper, LTspice is used as the simulation software, employing ideal discrete components to build a diode model, which is then compared in simulations with a high-voltage diode model downloaded from Infineon’s official website to verify the correctness of the constructed diode model.

## 3. Diode Operating Principle and Modeling

### *3.1 Diode Transient Circuit Model*

When the PN junction is forward biased and in the conducting state, the junction is filled with moving charge carriers, and the current becomes stable. When the PN junction is reverse biased and in the blocking state, the space charge region of the junction acts as a barrier, leading to a small leakage current. These two states are referred to as steady states. In addition to the steady states, the transient behavior of the diode (turn-on and turn-off) has a significant impact on the diode's characteristics and cannot be ignored. During the diode's transient process, the most important characteristics to consider are its forward recovery characteristic and reverse recovery characteristic, as shown in Figure 1 and Figure 2, respectively.

In analyzing the turn-on transient, particular attention must be given to the influence of resistive and inductive mechanisms on the diode's forward voltage. [3]Therefore, for the turn-on transient, a small resistor and an equivalent parasitic inductance in series can be used to model the diode.

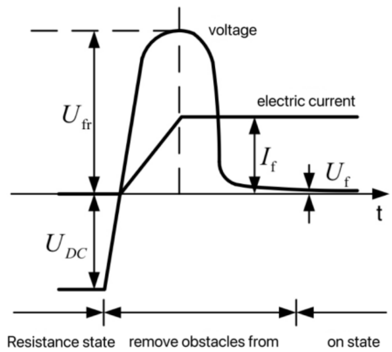


*Figure 1: Voltage and current variation during the forward recovery process*

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

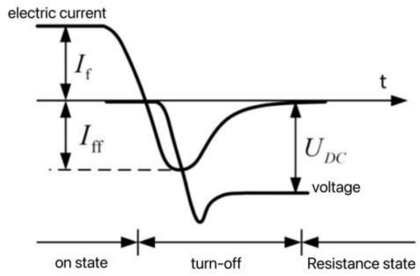


*Figure 2: Variation of voltage and current during the reverse recovery process*

From the above analysis, it can be concluded that the reverse recovery of a diode can be modeled by a capacitance, i.e., by connecting a capacitor in parallel across the diode. Furthermore, the reverse recovery characteristic is particularly prominent in Si-based diodes, whereas it is less noticeable in SiC diodes. Therefore, in the comparative simulation, the Si-based device from Infineon was selected for modeling in order to observe its reverse recovery characteristic.

### *3.2 Diode Steady-State Circuit Model*

The current-voltage (I-V) characteristic calculation for a PIN diode is highly complex, involving numerous factors. For instance, the applied bias voltage is not evenly distributed across the two junctions, and the carrier distribution in the intrinsic region is non-uniform. Generally, the forward I-V characteristic of a PIN diode can be expressed as:

$$U_F = k_0 + k_1 lnJ + k_2 J^m$$

The typical value of the coefficient m ranges between 0.6 and 0.8. The coefficients $k_0$, $k_1$, and $k_2$ are characteristic parameters that depend on temperature and the diode's structure. For the convenience of circuit analysis, the diode's current-voltage characteristics are typically represented by a threshold voltage $U_{F0}$ and a differential resistance $R_{on}$.

## 4. Transient Simulation Analysis of the Diode Model

To verify the accuracy of the constructed model, the SPICE simulation model of the IDW100E60 diode was downloaded from Infineon's official website. A comparative simulation was then conducted between the diode model built in this paper using ideal components and the official simulation model. This comparison of simulation waveforms confirms the accuracy of the model developed here.[2]

### *4.1 Simulation Based on Infineon's Official SPICE Model for the IDW100E60 Diode*

The simulation circuit is shown in Figure 3, with settings of Udc=300V and a current-limiting resistor R1 = 5 ohms. Parasitic inductance of 200 nH in the main circuit is considered. Since the SPICE model provided on Infineon's website already includes the device's internal parasitic inductance, additional internal inductance is not considered here.

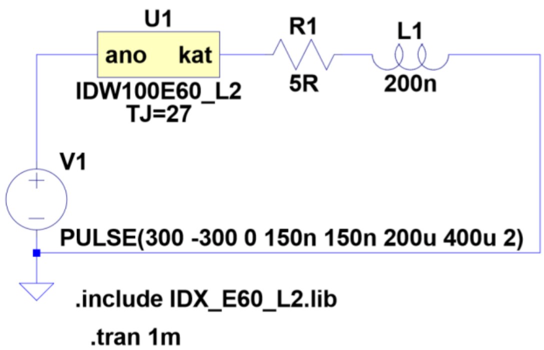


*Figure 3: Transient simulation circuit diagram based on Infineon IDW100E60*

The simulation data was exported in .txt format and then input into MATLAB for plotting, resulting

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

in voltage and current waveforms for the switching transients (see Figures 4 and 5).

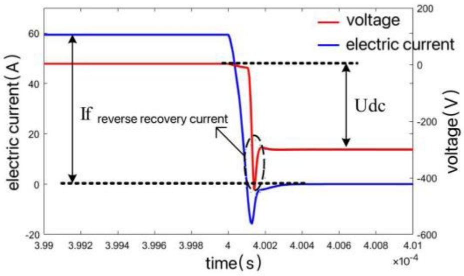


*Figure 4: Turn-off waveform based on Infineon IDW100E60*

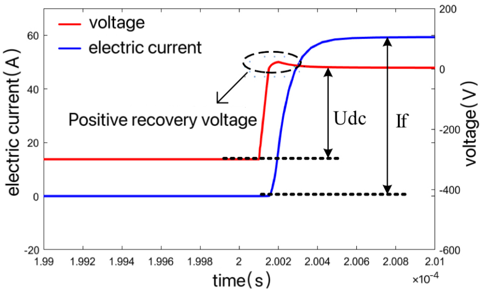


*Figure 5: Turn-on waveform based on Infineon IDW100E60*

From the simulation waveforms, it can be observed that the IDW100E60 diode model downloaded from Infineon's website exhibits a reverse recovery current during the turn-off transient and a forward recovery voltage during the turn-on transient. This behavior is consistent with the theoretical analysis presented earlier.

### *4.2 Based on Ideal Discrete Components*

From the analysis in Sections 3.1 and 3.2, it can be concluded that during the forward recovery process, the primary factors influencing the diode are the parasitic inductance and the conduction resistance within the diode. In the reverse recovery process, the dominant factor is the junction capacitance of the diode. Based on the above analysis, the equivalent model of the diode is established, as shown in Figure 6.

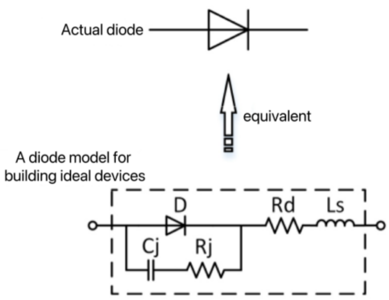


*Figure 6: Transient model of diode based on ideal discrete components*

The components in the diagram are all ideal components. D represents the ideal diode, Rd is the equivalent conduction resistance, Ls is the parasitic inductance within the diode, and Cj is the equivalent junction capacitance of the diode. The simulation circuit diagram of the diode model based on ideal components is shown in Figure 7.

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

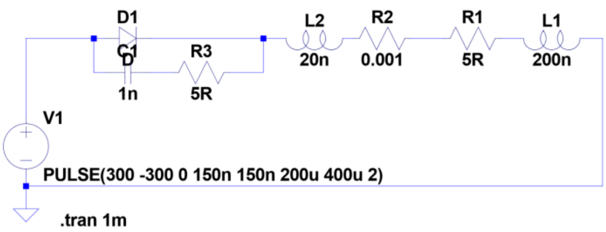


*Figure 7: Transient simulation circuit diagram of diode model based on discrete components*

The simulation data was exported in .txt format and then input into MATLAB for plotting, resulting in the voltage and current waveforms for the switching transients (see Figures 8 and 9).

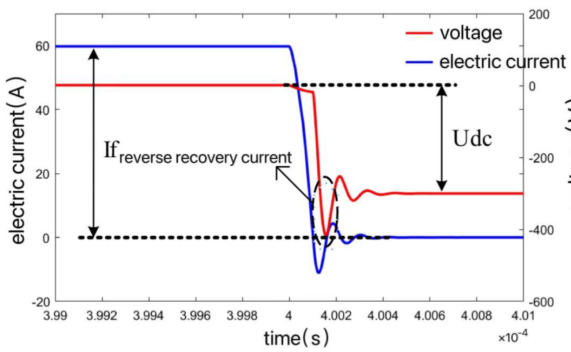


*Figure 8: Turn-off waveform based on discrete components*

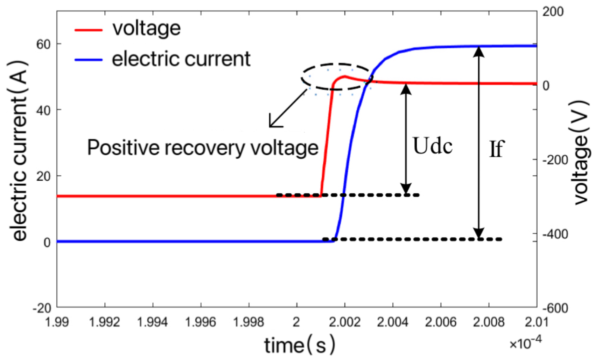


*Figure 9: Turn-on waveform based on discrete components*

From the simulation waveforms, it can be seen that the diode simulation model based on ideal components also closely matches the theoretical results: a reverse recovery current is observed during the turn-off transient, and a forward recovery voltage is observed during the turn-on transient.

To more intuitively demonstrate the accuracy of the constructed model, the simulation waveforms from the SPICE model downloaded from Infineon's official website were plotted alongside those from the model built using ideal components. The turn-off and turn-on waveforms are shown in Figures 9 and 10, respectively.

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

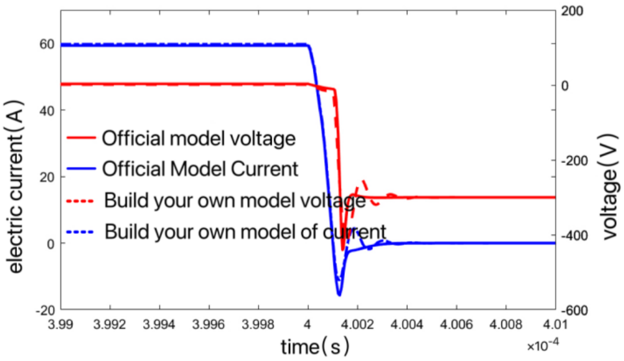


*Figure 10: Comparison of turn-off waveforms for the two models*

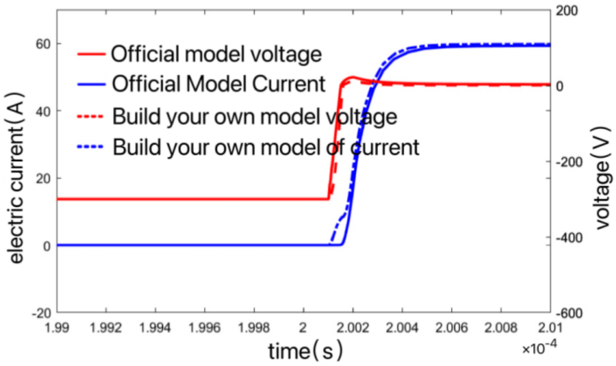


*Figure 11: Comparison of turn-on waveforms for the two models*

In Figure 11, the turn-on waveforms of the diode from the official SPICE model and the custom-built model using ideal components are compared. The red and blue solid lines represent the voltage and current waveforms of the official model, while the red and blue dashed lines correspond to the voltage and current waveforms of the custom-built model. Both models display a forward recovery voltage during the turn-on transient, consistent with theoretical expectations. However, slight discrepancies are observed between the two models. The official SPICE model, which includes a detailed numerical fitting approach, shows a more gradual voltage rise and a less abrupt current spike, likely due to its consideration of complex physical effects, such as the dynamic behavior of junction capacitance and conduction modulation. In contrast, the ideal component model simplifies these secondary factors, resulting in a sharper voltage rise and current spike. Despite these differences, the overall trends and transient characteristics align closely, demonstrating that the custom model provides a reasonable approximation of the diode's behavior during the turn-on process.

From the figures, it is evident that the simulation results of the diode model constructed in this paper are accurate and align with the results from the official diode model. The main reason for the simulation differences between the two models is that the official simulation model uses a numerical fitting approach to directly fit the diode's external characteristic curve and switching transient characteristics. In contrast, this paper uses ideal components to build the diode model, simplifying the diode's physical characteristics and neglecting many secondary factors, such as the variation of junction capacitance with external voltage and the conduction modulation effect.

## 5. Steady-State Simulation Analysis of the Diode Model

As described in Section 3.2, the steady-state conduction model of the diode can be approximated as a voltage source (threshold voltage) in series with a resistor. Since the voltage source in LTspice can be set as a linear source, the current-voltage (I-V) characteristic curve of the diode can be directly obtained. The simulation circuit diagram based on Infineon's official model and the equivalent simulation circuit diagram using the voltage source + resistor model are shown in Figures 12 and 13, respectively.

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

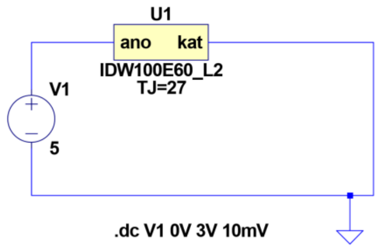


*Figure 12: Steady-state simulation circuit diagram based on Infineon IDW100E60*

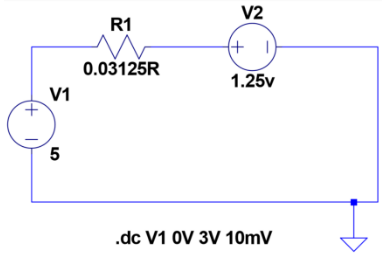


*Figure 13: Steady-state simulation circuit diagram based on Infineon IDW100E60*

The simulation data for both models was imported as .txt files into MATLAB and plotted on the same graph, resulting in Figure 14.

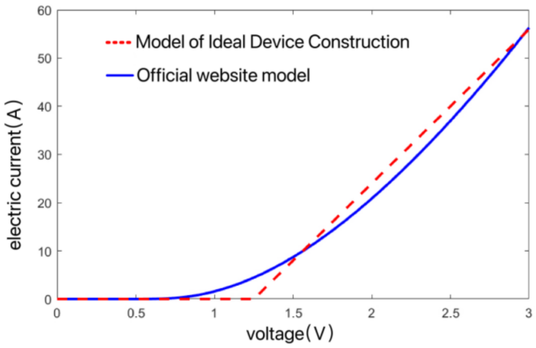


*Figure 14: Comparison of current-voltage characteristic waveforms for the two models*

From Figure 14, it can be observed that the current-voltage characteristic curve of the diode model using the threshold voltage + differential resistance exhibits a significant deviation only at the threshold voltage, while the rest of the characteristics match well with the model provided by the official source. Therefore, this method is very convenient for approximate calculation of device parameters (such as losses) and can also be used for simple comparisons between different types of diodes.

## 6. Simulation Analysis of the Diode Model Based on a Three-Level Clamping Circuit

After simulating the switching characteristics of the diode, a buck converter circuit was constructed to observe the forward and reverse recovery characteristics of the diode model based on ideal components in a practical circuit setting. The constructed buck circuit is shown in Figure 15.

The bus voltage was set to 300V with a duty cycle of 0.5, resulting in a buck-converted output voltage of 150V. The steady-state voltage and current data of the diode were exported and then plotted in MATLAB as shown. Figure 16 displays the steady-state continuous voltage and current waveforms of the diode. Expanding the turn-off and turn-on transients of the diode yields Figures 17 and 18, respectively.

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

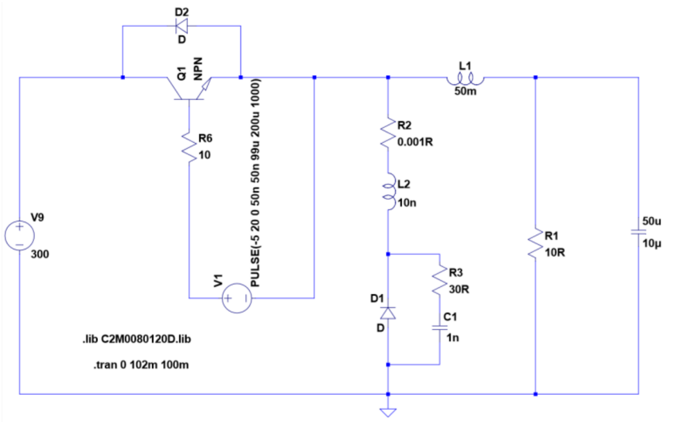


*Figure 15: Simulation circuit diagram of discrete component diode model based on buck converter*

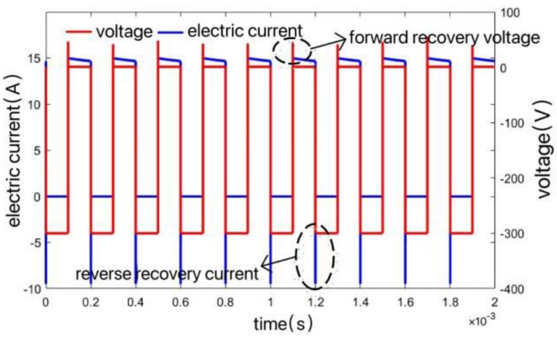


*Figure 16: Continuous operation waveform of diode model based on buck converter*

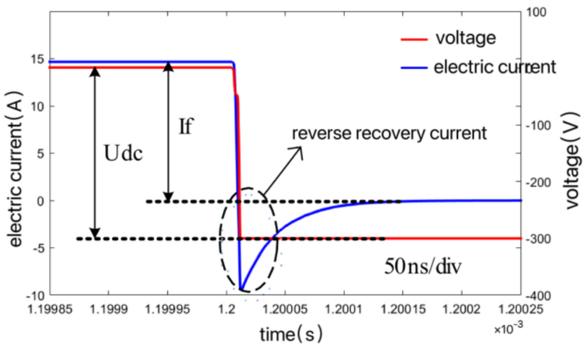


*Figure 17: Turn-off waveform of diode model based on buck converter*

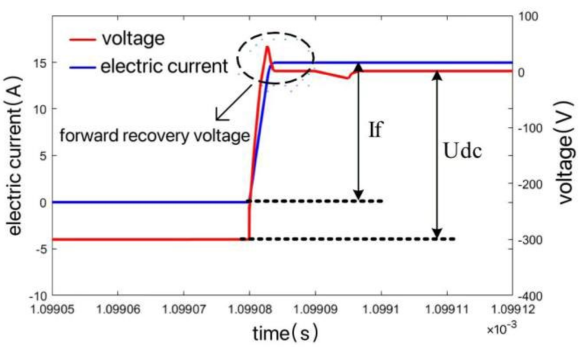


*Figure 18: Turn-on waveform of diode model based on buck converter*

From the simulation results, it can be observed that the diode's forward recovery voltage and reverse recovery current are still present in the actual circuit, verifying the accuracy of the constructed model.

ISSN 2706-655X Vol.6, Issue 6: 18-26, DOI: 10.25236/IJFET.2024.060603

## 7. Conclusion

This paper first describes the formation principles of PN and PIN junctions, then establishes both steady-state and transient models of the diode. LTspice simulations were performed for both the IDW100E60 diode model downloaded from Infineon's official website and the diode model based on ideal discrete components. A comparative analysis of the waveform differences between the two models was conducted, and possible sources of error were identified. Finally, a buck converter circuit was constructed, and the reverse recovery characteristics of the diode model were observed in the actual circuit.

## References

*[1] Orfao B, Vasallo B G, Moro-Melgar D, et al. Analysis of surface charge effects and edge fringing capacitance in planar GaAs and GaN Schottky barrier diodes[J]. IEEE Transactions on Electron Devices, 2020, 67(9): 3530-3535.*

*[2] Cui J, Zhang Y, Wei H, et al. Electro-Thermal Model for Schottky Barrier Diode Based on Self-Heating Effect[C]. 2021 IEEE 4th International Conference on Electronics and Communication Engineering (ICECE). IEEE, 2021: 330-334.*

*[3] Wu Q, Chen W, Yu C, et al. Multilayer machine learning-assisted optimization-based robust design and its applications to antennas and array[J]. IEEE Transactions on Antennas and Propagation, 2021, 69(9): 6052-6057.*