International Journal of Engineering and Technology, Vol. 17, No. 2, 2025

Research on the Application of EDA Simulation Technology in Integrated Circuit Design

Yadi Lin

Electrical and Computer Engineering, Rutgers, The State University of New Jersey, New Brunswick, NJ 08901, USA.
Email: 1816883604@qq.com (Y.D.L.)
Manuscript received October 20, 2024; accepted December 7, 2024; published May 30, 2025.

***Abstract*—Network-oriented intelligent lighting system integrated circuit automation design, in accordance with the integrated circuit multi-layer grid, the shortest path wiring structure, the use of Cadence or PSPICE and other EDA (Electronic Design Automation) simulation software, to carry out the digital circuit, analog circuit partition component design, the reasonable setup of bipolar (triple-pole) tube, Resistor, capacitor, inductor location and system parameters, automated scanning and analysis of DC or AC circuit transient current, voltage, signal frequency, based on the bias circuit schematic diagram to make a polyharmony oscillator circuit logic design structure, analog operating temperature rules check and verification, in order to ensure that the integrated circuit emitter, base current or voltage and other parameters in line with the needs of semiconductor design, enhance the Internet of Things intelligent lighting system Overall performance and working safety of IoT intelligent lighting system.**

***Keywords*—Electronic Design Automation (EDA) simulation techniques, integrated circuits, design, applications**

I. Introduction

Today's Io intelligent lighting system involves a variety of environmental sensors, command controllers and other end devices within the system, the introduction of Computer Aided Design (CAD), PSPICE, Mentor PADS or Cadence and other electronic automation simulation and design software, to provide support for the design of the intelligent lighting system for the digital circuitry, analog circuitry component wiring design. It can realize the reasonable layout of IC components under different application scenarios. According to the conductivity of different components connected in the integrated circuit of the intelligent lighting system, as well as the noise of the data sampling of capacitive and inductive signals, the accuracy and completeness of the circuit signals are verified by using the HSPICE software, which can ensure that the command response of the system design and the PWM dimming curves can meet the service requirements of intelligent lighting [1].

II. Functional Control Circuits, Integrated Circuit Structure Analysis of the Internet of Things Intelligent Lighting System

A. Function Control Circuit Design in Intelligent Lighting System

The current indoor intelligent lighting system is mainly oriented to the control needs of light brightness and extinction, light intensity, color temperature and other adjustments, set up to cover the microprocessor unit, ambient light sensors, temperature sensors, LED drivers, photoresistors, power management units, storage units, the composition of the circuit structure [2]. Which selected ARM RTL8019AS 32-bit microcontroller, Vishay TEMT6000 photosensitive sensor, TI TMP117 temperature sensor, Infineon ILD6070 LED driver, as well as TCP/IP stack, CSMA/CD protocol, Zigbee/BLE protocol, ESP8266 interface and other communication Technology to construct the system functional circuit, forming the LED lamp remote PWM dimming, temperature control light regulation control.

ARM RTL8019AS main controller peripheral set AC-DC power supply interface, 3.3~5V output voltage, to provide DC / AC circuit power supply management; set ADC pin for the environment sensor collection of analog signals to provide support for the incoming, set GPIO pins, CSMA/CD protocol to meet the needs of the local area network within the control of the LED driver and connectivity requirements, set the I ² C / SPI Set up I²C/SPI bus, I/O pin interface and other connection ports to provide support for digital signal conversion, PWM dimming signal output control [3].

And then based on the UART interface, TXD/RXD pins to form a microcontroller and access to the serial communication of the lighting module, according to the system load, driver specifications to connect the LED string. Using the power manager +5V, rated current of 200μA half-duplex communication mode, the driver through the TXD/RXD terminal, digital to analog signal converter through the AD port and the microcontroller microprocessor connected to the analog-to-digital converted signals are output to the ARM RTL8019AS 32-bit microcontroller to make processing analysis [4]. LED driver is set by the current sampling resistor R6, inductor L1, current diode composition of the self-oscillating current mode, the LED sampling voltage and bias voltage comparison to generate PWM control signals, and then by the ARM RTL8019AS main controller of the power switch manipulation drive circuit conduction and shutdown, the same kind of signal components are centrally laid out, and through the resistor grounding [5].

B. Integrated Circuit Structure Analysis of Intelligent Lighting System

Intelligent lighting system integrated circuit design, the choice of digital circuits, analog circuits separate master-slave control mode, divided into light signal sampling circuit, A/D analog-to-digital conversion circuit and other components of the structure. The host is the ARM RTL8019AS master controller for high-frequency signal transceiver communication, and the slave is the MAX485 controller (Fig. 1) [6].

DOI: 10.7763/IJET.2025.V17.1313

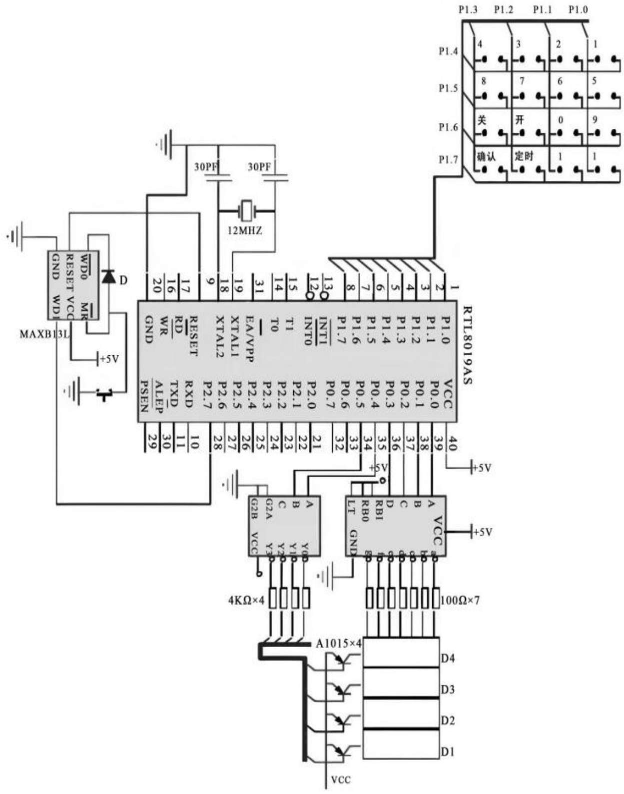

Fig. 1. Integrated circuit structure of ARM RTL8019AS host controller.

Master-slave controller is in half-duplex operating state, using P1.0~P2.7 pins on the analog-to-digital conversion after the transmission and reception of differential signal end of the control, and P pins connected to the A terminal and B terminal for sending and receiving differential signals, when the level of the A terminal is higher than the B terminal indicates that the signal is in the transmission state, the data is 1, otherwise indicates that the signal is in the reception state, the data is 0, the master controller ARM RTL8019AS control receiver to complete the digital signal transceiver communication. Based on the TLP521 photocoupler anti-jamming support, the host receives through the P1.0~P2.7 pins to control the transceiver's operating state, usually set P1.0 and other pins interface to a low level, so that the host's serial port is in the signal receiving state, when the serial communication interruptions to determine the address of the signal, and will be set to a high level on the P1.0 pin to the host's serial port is in the signal to send the answer state [7].

Fig. 1 in the A/D analog-to-digital converter analog-to-digital conversion is another core of the control circuit, the use of Texas Instruments TLC1549 analog-to-digital converter access to the microcontroller system, for the high impedance input electromagnetic induction, electrostatic coupling noise problems, the use of differential reference voltage high impedance input, in the reference voltage source power supply side to add LC-type filtering circuits, will be two 100pF~1nF capacitors respectively Connect the differential line and ground, constitute a low-pass filter circuit, used to inhibit and filter out high-frequency signal input ripple noise interference, the input denoising high-frequency signals are transmitted to the master-slave controller, the application of the microcontroller circuit to control the light of the intelligent switching, brightness and color temperature adjustment [8].

III. IC Design Execution Flow based on EDA Simulation Technology

PSPICE, Cadence or Mentor PADS and other EDA software electronic automation simulation design, mainly upholding the concept of systematic chip design, large-scale digital circuits, analog circuits, digital-analog hybrid simulation experimental design, according to the complexity of different integrated circuits of the functional requirements of the set schematic input, PR/EDIF netlist, component wiring, rule checking, layout extraction, parasitic parameter extraction, EDA simulation, etc., so that the layout of components in the circuit, circuit performance indicators to meet the actual scenarios and application requirements.

A. IC Design Flow Based on PSPICE Simulation Software

After the start of the PSPICE software, click the menu bar "File->New->Project" to create a new project, select "New Schematic" to open the schematic editing window, in the project manager click the menu bar in the project manager, click on the menu bar "Place->Part" to create the corresponding components, use the "Place -> Wire" command to draw the wires between the component pins. And then according to the drawn schematic for PR/EDIF netlist generation, select the menu bar of the "PSpice -> Create Netlist" command to generate the netlist file, the netlist file records the parameters of the circuit components, connectivity and other information to complete the schematic drawing centered on the needs of the intelligent lighting system [9].

After that, the netlist file will be imported into the PCB design module to make component wiring, in the PSPICE software menu bar click on "File -> "Export -> Netlist" command, to generate EDIF format netlist file, and then in the PCB design software click on "File-> Import" -> Netlist "import netlist, click" Place-> Component "command will be placed in the layout of components to the netlist area, click" Route-> Auto Route "to select the layout of the netlist. "Route->Auto Route" command for automatic routing operations.

After completing the netlist wiring of different components, click "Tools -> ERC" command in the schematic editing window, the simulation software to check the circuit whether there is too wide a line width, unreasonable spacing, open/short, pin overhang, etc. If there is no error problem, then use Layout Extraction tool to extract the layout and click "Analysis -> Parasitic Extraction" command to extract parasitic parameter information, including voltage, current, resistance, capacitance, inductance and other parameter information. Extract the layout and click "Analysis -> Parasitic Extraction" command to extract parasitic parameter information, including voltage, current, resistance, capacitance, inductance and other parameter information. Finally, a new simulation profile is created in the schematic editing window, and the parameter ranges for AC/DC scanning, DC-AC analysis, and transient analysis are set to make a simulation run of the circuit to ensure the accuracy of the electronic automation design.

B. IC Design Flow Based on Cadence Simulation Software

Open the software: Start the Cadence software to enter the main interface, click the menu bar of the "File->New->Cell View" to create a new project, in the process library to select the CSMC × nm integrated circuit process. After that, click the "Place->Instance->Browser" command in the menu bar, select the component models of resistors, capacitors, MOS tubes, etc., and draw the wires between the component pins using the "W" shortcut key in the "New Schematic" window of the schematic editor. Place the components, use the "W" shortcut key to draw the connection between the component pins [10].

And then according to the circuit schematic connection line for PR/EDIF netlist generation, in the schematic interface click on the menu bar of the "Tools->Simulation->Netlist->Create Netlist" command, selected Spectre, SPICE, EDIF, and other netlist formats to generate netlist files, the SPICE format netlist files exported to . cir files will be imported into the PCB design tools to make component wiring. In the schematic interface, click "Tools->Simulation->Netlist->Create Netlist" command in the menu bar, select Spectre, SPICE, EDIF and other netlist formats to generate the netlist file, and then export the netlist file in SPICE format to a .cir file, and then import the netlist file into PCB design tools to make the wiring of components. In the PCB design software menu bar click "Place->Manually->Quickplace" command, the use of manual or automatic way to place a variety of components, the use of "Verify-> Constraints Use the "Verify-> Constraints" command to set a reasonable wiring width, spacing.

Finally, in the schematic editor page, click "Verify-> DRC" command, select the CSMC×.drc rule file with different integration accuracy, and check the wiring results of the electrical components in the schematic diagram, so that the wrong wiring and marking information can be found in time, and then make the layout extraction after all the wiring is accurate. After all the wiring is accurate, the layout is extracted. Set extraction parameters through "Tools->Layout Extraction" command in the menu bar, and set parasitic parameters through "Analysis->Parasitic Extraction" command in the menu bar. After setting the parasitic parameters through the command "Analysis->Parasitic Extraction" in the menu bar, the layout and parasitic parameters can be extracted within a specific precision and range. Also after the completion of the integrated circuit design, set the AC/DC conversion signals, circuit transients of the simulation type of simulation run to ensure that the simulation of the circuit diagram drawing, component wiring placement accuracy.

IV. PSPICE Simulation Technology in the Intelligent Lighting System Integrated Circuit Design Application Program

A. Application of PSPICE Simulation Technology in the Design of Intelligent Lighting System Integrated Circuits

Based on the menu bar "File-> New-> Project" of PSPICE software, create a new project of "Intelligent Lighting System", place resistors, capacitors, filter capacitors, diodes and other components in the schematic diagram of the project, connect the wiring components by clicking on the command "Place-> Part->Wire", set the system voltage threshold as ＋5V~+30V, current threshold as 200μA~1A for the LED driver. Place resistors, capacitors, filter capacitors, diodes and other components in the schematic diagram of the project, connect the line components by clicking on the command "Place-> Part->Wire", set the voltage threshold of the system as ＋5V~+30V, and the current threshold as 200μA~1A, to provide support for the hysteresis control of the current of the LED driver.

Through the "Place->Manually->VPULSE" command to simulate the driver output PWM signal, according to the human eye to watch the intelligent lighting system, "visual retention" principle, set the LED driver to enable the frequency of PWM signals According to the human eye to watch the intelligent lighting system, "visual retention" principle, set the frequency of the LED driver enable PWM signal to 100Hz~1kHz, duty cycle of 0 ~ 100%, connected to the resistor, capacitor, diode opto-coupler isolation circuit, specific as shown in Fig. 2. As the maximum input voltage of the reference circuit is 30V, the integrated circuit common current control MOS tube withstand voltage of+5V, and thus add a gate-drain short between the MOS tube drain source for circuit protection to prevent the integrated circuit end of the high-voltage breakdown [10].

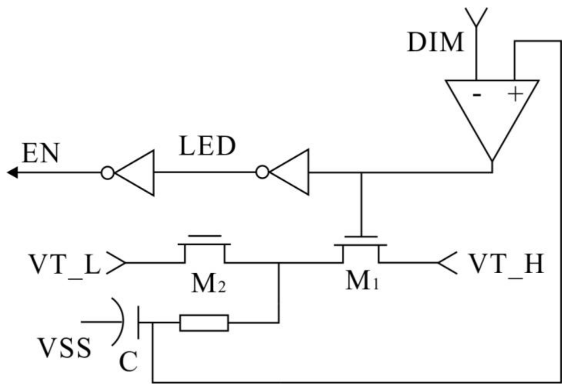

Fig. 2. Analog dimming PWM signal equivalent circuit diagram.

From the analog dimming PWM signal equivalent circuit diagram in Fig. 2 the output of the EN signal can be controlled by comparing the input signal voltage at the DIM pin with the size of the low threshold voltage VT_L and the high threshold voltage VT_H. In the DIM pin voltage from high to low, the current flowing through the LED tube I is also reduced proportionally, when the DIM pin voltage is lower than VT_L (0.5 V) when the power tube is cut off, when the DIM pin voltage is low to high, the LED tube to make the brightness and color temperature adjustment, higher than the high threshold voltage VT_H to complete the enable conversion.

B. Simulation Results Analysis

Based on the CSMC 1nm IC process model, the circuit design results of the "Intelligent Lighting System" are simulated and verified using HSPICE software. In 20ms simulation time, set the sampling resistor RS = 0.3 Ω, inductor L=100μH, and run "PSpice->Run" to check the output current accuracy [11].

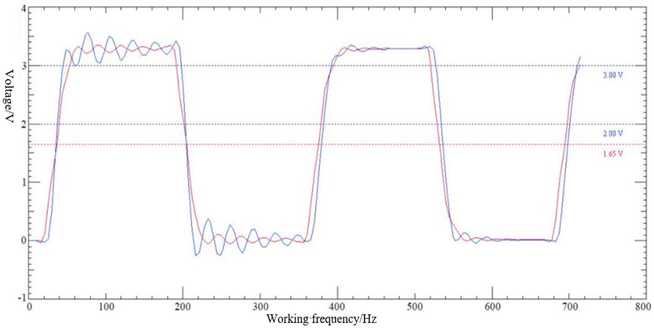

Fig. 3. Simulation experiment results.

The chip output current deviation is found to be 0.5%~3.5% when driving 5 LED light sources, and the output signal topology of the circuit after adding the matching resistor is shown in Fig. 3, which shows that the terminated signal waveform is gradually stabilized by the oscillation, and the simulation results are very satisfactory and can meet the practical requirements.

V. Conclusion

In summary, the intelligent lighting system IC design based on EDA simulation technology can be achieved by using Cadence, PSPICE simulation software technology to make a reasonable wiring of the rectangular grid channel, in the project engineering page of Cadence Allegro to make the analog power supply, micro-controller, photoresistor and other circuit components of the alignment layout design, by the micro-controller to PWM digital The microcontroller converts the PWM digital dimming signals into current control signals, performs transient simulation operation calculations of input/output voltages and currents, and realizes the optimization of the whole process from architectural design to simulation verification.

Conflict of Interest

The author declares no conflict of interest.

References

[1] F. F. Du, "Application of EDA simulation technology in integrated circuit design," *Integrated Circuit Applications*, vol. 03, pp. 10–11, 2022.

[2] T. Wu, "Application of simulation technology in integrated circuit industry," *Integrated Circuit Applications*, vol. 05, pp. 52–54, 2024.

[3] S. N. Liu, D. D. Kan, Y. L. Li, X. T. Wang, J. Nan, and Z. Y. Ma, "Current status of EDA development and standardization in integrated circuits," *China Standardization*, vol. 20, pp. 49–53, 2024.

[4] Y. G. Wu, Y. Zhang, and Y. N. Xu, "Application of EDA technology in the teaching of 'digital circuits' course," *China Electric Power Education*, vol. 08, pp. 72–73, 2021.

[5] L. X. Gao, Y. F. Chen, Z. J. Shi, J. F. Li, and X. H. Jia, "Discussion on technical issues of EDA software installation for integrated circuit layout," *Journal of Qingyuan Institute of Vocational Technology*, vol. 02, pp. 1–4, 2008.

[6] D. Liu, "Application of EDA simulation technology in integrated circuit design," *Information Technology and Informatization*, vol. 01, pp. 139–141+145, 2023.

[7] F. F. Qi, Y. H. Wang, and L. H. Wang, "Digital-analog mixed circuit simulation interactive system based on Ngspice," *Industrial Technology Innovation*, vol. 9, no. 4, pp. 22–29, 2022.

[8] X. M. Zhang, "Application of electronic circuit simulation technology in integrated circuit design," *Application of IC.* vol. 40, no. 3, pp. 20–21, 2023.

[9] J. F. Gai, W. L. Zhang, R. Q. Yu *et al.*, "Application of electronic circuit simulation technology in integrated circuit design," *Peak Data Science*, vol. 21, pp. 97–99, 2023.

[10] J. Zhang, "Application of electronic circuit simulation technology in integrated circuit design," *Electronic Components and Information Technology*, vol. 10, pp. 94–96, 2020.

[11] Y. Zhang, "Research on the difficulties and solutions of electronic circuit simulation technology in the development of electronic applications," *Electronic Components and Information Technology*. vol. 5, pp. 122–123, 2020.