Comparative Analysis of Software Environments for Computer Modeling In Electrical and Electronic Engineering

Authors:

Nikolay Hinov – Technical University of Sofia, Bulgaria

Stoyan Popov – Technical University of Sofia, Bulgaria

Content
INTRODUCTION
MAIN CHARACTERISTICS AND DESCRIPTION OF THE SOFTWARE ENVIRONMENTS
CONCLUSION
ACKNOWLEDGMENT

Introduction

Computer modeling of electrical and electronic circuits is the process of using specialized software tools to create, analyze, and optimize electrical and electronic devices. This approach allows engineers to develop, test, and optimize circuits prior to their physical prototype.

Main characteristics and description of the software environment

A. LTspice capabilities for computer modeling and simulations of electrical and electronic circuits

LTspice is a high-performance SPICE (Simulation Program with Integrated Circuit Emphasis) simulation software developed by analog chip manufacturer Linear Technology (now part of Analog Devices). It is available for free download and is designed for simulations of analog circuits and mixed analog-digital systems.

Main characteristics and description of the software environment

B. PySpice capabilities for computer modeling and simulations of electrical and electronic circuits

PySpice is a Python library that provides an interface to the Ngspice program, which is a free and open source electronic circuit simulation software. Ngspice is based on the old SPICE3, but has been updated and extended by the free software community.

Main characteristics and description of the software environment

C. Matlab/Simulink capabilities for computer modeling and simulations of electrical and electronic circuits

Simulink is software developed by MathWorks that allows users to model, simulate, and analyze dynamic systems. It is integrated with MATLAB, allowing easy access to MATLAB algorithms and functions.

TABLE 1. COMPARISON OF ANALYSIS TYPES, MODELS, LIBRARIES AND MODULES

Software	Types of analyses	Models	Libraries and modules
LTspice	Transient analysis DC analysis AC analysis Noise analysis Sensitivity analysis Parametric analysis Thermal analysis	Resistors Capacitors Inductors Semiconductors Operational amplifier Integrated circuit Transformers Thermistors Microcontrollers etc.	LTspice beads standard library LTspice bipolar standard library LTspice capacitors standard library LTspice diodes standard library LTspice inductors standard library LTspice jfet standard library LTspice mos standard library LTspice resistors standard library Schematic editor Waveform viewer Simulator
PySpice	Operating point analysis Transient analysis Frequency analysis Sensitivity analysis Noise analysis DC Sweep AC Sweep Parameter sweep	Resistors Capacitors Inductors Diodes BJTs MOSFETs JFETs Operational amplifier etc.	Spice Probe Netlist Library Plot Analysis NumPy Matplotlib Pandas SciPy CFFI
Simulink	Analysis of temporal characteristics Analysis of frequency characteristics Sustainability analysis Sensitivity analysis Variability and Uncertainty Analysis Optimization analysis Performance analysis Wear analysis and long-term performance System error and reliability analysis Power supply and energy efficiency analysis	Mathematical and algebraic models Models of control systems Signal processing models Models of communication systems Physical models Models of discrete and queuing systems	Simulink block libraries Simscape Stateflow Signal Processing Blockset DSP System Toolbox Communication System Toolbox Control System Toolbox Simulink Control Design Simulink Design Optimization Embedded Coder Simulink Real-Time SimEvents Stateflow Simulink PLC Coder Simscape SimRF Simulink Verification and Validation

TABLE 2. COMPARISON OF LAWS, MATHEMATICAL METHODS, ADVANTAGES AND DISADVANTAGES

Software	Laws	Mathematical methods	Advantages and disadvantages
LTspice	Ohm's Law Kirchhoff's Laws Coulomb's Law Faraday's Law of Electromagnetic Induction Planck's Law etc.	Solved ODEs (Gear method etc.) Newton-Raphson method Fourier analysis Matrix operations Monte Carlo method etc.	Advantages: Free, Wide range of features, Large number of built-in component models, Powerful and accurate Disadvantages: Non-intuitive interface, Limited support for third-party models, Limitation in visualization, Limited modeling capabilities for digital circuits
PySpice	Ohm's Law Kirchhoff's Laws Maxwell equations Shockley equation Faraday's Law of Electromagnetic Induction etc.	Numerical solution of differential equations (Euler's method, Runge-Kutta methods etc.) Solving linear systems of equations (Gaussian method, Newton-Raphson method, etc.) Fourier analysis Monte Carlo method etc.	Advantages: Python usability, Power of SPICE, Integration with other Python libraries, Open source Disadvantages: Complexity, Dependence on SPICE, Slower code performance, Difficult to install and set up for beginners
Simulink	Newton's laws of motion Ohm's law and Kirchhoff's laws Laws of thermodynamics Maxwell's equations Hooke's Law Navier-Stokes equations Differential equations Linear systems and control laws	Numerical solution of differential equations using methods such as Euler's method, Runge-Kutta methods, etc. Integral calculations Scalar and vector operations Solving systems of linear equations Transformations (Fourier, Laplace) Statistical analysis Optimization methods (Gradient methods, Stochastic gradient methods, Genetic algorithms, etc.) Various principles and methodologies from control systems theory	Advantages: Visualization of the models Integration with MATLAB Real-time modelling Rich set of predefined blocks Support for various standards (AUTOSAR, DO-178, ISO 26262, etc.) Disadvantages: Cost to purchase Requires significant understanding of systems and control theory Requires powerful hardware Incompatibility between different versions Requires time and effort to learn, especially for those unfamiliar with MATLAB

Conclusion

In conclusion, the question of which software is "better" - LTspice, PySpice or Simulink - depends largely on the specific needs of the user and the specific application area.
LTspice is a powerful and free electronic circuit simulation tool that is particularly useful for analog simulations. It is Python scripts, PySpice might be best. If you want to model complex multi-domain systems and have a MATLAB license, Simulink may be best for these purposes.

Acknowledgment

This research was carried out within the framework of the project “Artificial Intelligence-Based modeling, design, control and operation of power electronic devices and systems”, КП-06-Н57/7/16.11.2021, Bulgarian National Scientific Fund.

Thank you very much for your attention