

SPICE Simulation History

Publish Date: May 16, 2008

Table of Contents

1. [Overview](#)
2. [SPICE Simulation History](#)
3. [References](#)

1. Overview

The National Instruments [SPICE Simulation Fundamentals](#) series is your free resource on the internet for learning about circuit simulation. The series is a set of tutorials and information on SPICE simulation, OrCAD pSPICE compatibility, SPICE modeling, and other concepts in circuit simulation.

For more information, see the [SPICE Simulation Fundamentals](#) main page.

The series is divided among a number of in-depth detailed articles that will give you HOWTO information on the important concepts and details of SPICE simulation.

Circuit simulation is an important part of any design process. By simulating your circuits, you can detect errors early in the process, and avoid costly and time consuming prototype reworking. You can also easily swap components to evaluate designs with varying bills of materials (BOMs).

SPICE simulation has been used for over thirty years to accurately predict the behavior of electronic circuits. Over the years the many revisions of SPICE have seen improvements in both accuracy and speed. In addition to these improvements, additions to the language have allowed simulation and modeling of more complex integrated circuits including MOSFETs.

2. SPICE Simulation History

Simulation Program with Integrated Circuit Emphasis, or **SPICE**, has been used for over thirty years. The original implementation of SPICE was developed at the University of California Berkeley campus in the late 1960s. SPICE was developed largely as a derivative of **CANCER** (Computer Analysis of Nonlinear Circuits, Excluding Radiation) also developed by UC Berkeley.

The first widely used version of SPICE was announced in Waterloo, Canada in 1973. Shortly thereafter SPICE was adopted by nearly all major engineering institutions throughout North America. SPICE has evolved into the academic and industry standard for analog and mixed-mode circuit simulation.

Over the years additional simulation algorithms, component models, bug fixes, and capabilities were added to the program. Even today SPICE is still the most widely used circuit simulator in the world and as of 2006 the latest version is SPICE 3F5.

XSPICE was developed at Georgia Tech as an extension to the SPICE language. XSPICE allows behavioral modeling of components which can drastically improve the speeds of mixed-mode and digital simulations. [Multisim](#) from **National Instruments** is based on SPICE 3F5 and XSPICE and provides additional convergence and speed improvements to complement these powerful simulation languages.

3. References

The SPICE Book, Andrei Vladimirescu, © 1994 John Wiley & Sons

The Life of SPICE, Laurence W. Nagel, © 1996