

The 40-th Anniversary of the Simulation Program with Integrated Circuit Emphasis - *SPICE*

Goce L. Arsov, *Senior Member, IEEE*

SS Cyril and Methodius University, Faculty of Electrical Engineering and Information Technologies
Karpos II, b.b. P.O. Box 574
Skopje, Republic of Macedonia
g.arsov@ieee.org

Abstract—The paper is dedicated to the Simulation Program with Integrated Circuit Emphasis (SPICE) and its 40-th anniversary. Since its release in early 1970's SPICE has been serving the professional community in developing, design and production of different kind of electric and electronic circuits. It has been also very useful in educational purposes. SPICE finds its roots in CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation) developed as a student project by Laurence Nagel and his colleagues under the supervision of Prof. Ron Rohrer at the University of California at Berkeley, and is basic part of Dr. Nagel's PhD Dissertation where the improved version (SPICE2) was described in details. Since its appearance SPICE has been improved many times and today it is available for mainframes, as well as for PCs with different operating systems. In the paper a list of different versions is given. Happy birthday SPICE.

Keywords- *electronics circuits; modeling; program; simulation; SPICE.*

I. INTRODUCTION

Electronic circuit design requires accurate methods of evaluating circuit performance. Once a design for a circuit has been determined, the design must be tested to ensure that the circuit indeed conducts as required. Often this involves testing for DC operating point and performance with signals applied, over a range of DC supply voltages, input signal levels, and temperatures. One way to do this was to build a prototype of the circuit, send it to the laboratory, and invest large amounts of time and money in putting the circuit through its paces, in the hope that it would perform as desired. Even if it did, one did not know how the circuit would perform with active devices whose parameters ranged considerably from a nominal value. For example, a small signal transistor typically has a DC current gain, or beta, around 100-150. The values of beta, one might encounter in a lot of acceptable transistors shipped by a reputable manufacturer, could range from 40 to more than 250. A prudent circuit designer would have to test the circuit operation by building many test circuits, using transistors at both ends of the beta range, and putting all of them through their paces. If there was a way to simulate accurately and with confidence the performance of electronic circuits without having to build them, the development cost and time to bring a new circuit into production could be significantly reduced. This is why a great deal of effort was put into simulation of circuit performance by computers. Today, because of the enormous

complexity of modern electronic circuits, the computer aided circuit analysis is essential and can provide information about circuit performance that is almost impossible to obtain with laboratory prototype measurements.

The computer-aided analysis and simulation can be useful in several aspects, such as:

- At the very first stage of the design process, the fundamental principle of operation of the circuits can be verified. Later simulations can include all additional effects caused by any parasitic element in the circuit.
- One of the most important advantages is the possibility to check on paper design before construction of expensive prototype.
- It is much less expensive to examine the problems, such as the stresses of the devices, by simulation, at an early stage of the design. It can detect possible problems even before making the prototype of the circuit.
- Easy testing of new design solutions, as well as the evaluation of the effects of parameter variations of the used components in the circuit is possible. The improvements of the topologies can be made without additional expenses.
- Evaluation of the effects of noise and signal distortions without the need for expensive measuring instruments.
- Simulation of different destructive testing of fault conditions, that, sometimes, cannot be done, even in special laboratories, can be easily performed.
- It gives us the possibility to “look inside” the components.
- The effects of imperfections can be easily examined.
- It can be used for improving the documentation of the power electronics system performances.

The simulation may be very suitable for educational purposes and can give very useful information for good understanding of the operation of a power electronics circuit.

It is very important to point out that the simulation should correspond to the examined problems. It means that, to get an

answer, all parameters that can influence the circuit operation concerning that specific problem should be taken into consideration. Very often the answers to different problems should be solved by specific simulations.

The main disadvantages of the simulations may be listed as:

- Can be very time consuming;
- If the input data are not correct, the results of the simulation may not be correct. (“Garbage in, garbage out”);
- The inadequacy of the component models can influence the results of simulation;
- Inherent errors due to discretisation are to be considered.

One of the first programs for analysis and simulation of electronic circuits was ECAP (Electronic Circuit Analysis Program) released by IBM in 1960’s, but today one of the most popular is SPICE (Simulation Program with Integrated Circuit Emphasis) developed at the University of California at Berkeley in early 1970’s.

This paper is dedicated to the 40-th anniversary of this, today, very popular and widely used program.

II. SIMULATION PROGRAMS

At the very beginning the simulations were performed using analog computers for solving mathematical equations that were describing the operation of the circuit. Here, the devices were modeled by the mathematical expressions representing their I - V characteristics. The evolution of the digital computers made them available for solving mathematical expressions of any kind, and for simulation of any physical process. Mainly digital computers are used for these purposes today. Digital simulators, regarding the user interfaces, can be divided in two main categories: 1) Equation solver programs, 2) Circuit oriented programs..

A. Equations-Solver Programs

Equations-solver programs may be very useful in solving many electronic simulation problems. Basically, these programs are used to solve the equations describing the system under investigation or design. These programs are used when the user is interested in the system behaviour rather than in the detailed device level operation. Besides the widely used common program languages, e.g. FORTRAN or C, some special packages, such as MATLAB, SMNON, ACSL, MATRIXx, are also available on the market.

B. Circuit-Oriented Programs

The Circuit-oriented programs, or circuit simulators, are designed for analyzing the circuit topology rather than the equations. Within an input file, the circuit is specified in terms of element names and values, nodes, variable parameters and sources. Different type of analysis can be performed. The new versions of these programs are designed to accept the input file specified as a schematic diagram of a circuit to be simulated. Multilevel programming and definition of device models, or sub-circuits, by the user are incorporated in the later versions of these programs.

C. ECAP (Electronic Circuit Analysis Program)

Computer-aided circuit analysis first became popular in the mid 1960’s when the computer program ECAP (Electric Circuit Analysis Program) [1] was developed by programmers at IBM Corporation and made available on the market. This software for mainframe computers (IBM 1130 and IBM 360) made it possible for an engineer to analyze an electric or electronic circuit without having to write equations defining the circuit. The program user had only to describe the circuit using nodal notation, and the program itself would write the circuit equations and solve them. Prior to the development of ECAP, digital computers had been in use for years for solving circuit equations. But the tedium involved in writing those equations for all but the simplest circuits was substantial. And once the equations were written by an engineer and solved by a computer, any changes to the original circuit design necessitated the rewriting of the circuit equations. Its improved version, ECAP II, was released in early 1970’s [2], [3]. ECAP was followed by several similar programs, which offered a variety of improvements to ECAP. Among these were SCEPTRE [4], CIRCUS [5], TRAC [6] and CIRPAC [7].

During the early 1970’s, the number of active devices on integrated circuits (ICs) being developed grew dramatically; SSI (small-scale integration, characterized by about 20 transistors on each IC) led to MSI (medium-scale integration, with about 100 transistors per IC). (Specific quantitative definitions of SSI, MSI, LSI and VLSI vary; the transistor counts shown here are representative.) Later came LSI (large-scale integration) and VLSI (very large-scale integration), in which there were literally thousands of transistors. As a result CAD became a necessity for IC development because it was physically impossible to prototype accurately an integrated circuit using discrete components. A collection of individual transistors, diodes, resistors and capacitors connected together by wires on a circuit design board differs considerably from the same circuit implemented on an integrated circuit for the following reasons: a) the stray, or parasitic capacitances and inductances of an IC are much smaller than those of a circuit built on a circuit design board; 2) each discrete component on a circuit made on a PCB will have a different temperature, due to the differing amounts of power dissipated by each active or passive component, while on an IC, the close proximity of components ensures that they are all at essentially the same temperature; 3) propagation delay is dependent upon the geometry and physical size of the circuit, so the high-speed performance of a circuit made on an IC will bear little resemblance to the performance of the same circuit built on a PCB with discrete components [1].

One of the programs which could partly deal with ICs was ECAP II [2], [3], but it was not easy for users. It required quite long, advance, pre-education.

III. BREF HISTORY OF SPICE

The roots of SPICE can be found in the student projects led by Professor Ron Rohrer (Fig. 1). In fact, in early 1970’s, Ron Rohrer hopes to develop a simulation program for his work on optimization at the University of California Berkley. Rohrer’s students, including Laurence “Larry“ Nagel (Fig. 2), create a

program CANCER (Computer Analysis of Non-Linear Circuits Excluding Radiation) [8]-[10]. The program was able to perform DC, AC and Transient Analysis. The program has built-in models for diodes (Shockley equations) and bipolar transistors (Ebers-Moll equations). The initial conditions necessary for performing the transient analysis could be determined through proper DC analysis of the circuit.



Figure 1. Ron Rohrer IEEE Fellow - today: Professor Emeritus of Electrical and Computer Engineering at Carnegie Mellon University



Figure 2. Laurence Nagel IEEE Life Fellow - today: President of Omega Enterprises Consulting

A. SPICE1

SPICE (Simulation Program with Integrated Circuit Emphasis) represents the basis of Laurence Nagel PhD Dissertation made under supervision of Professor Donald Pederson (Fig. 3). It is an upgrade of its previous work, CANCER. Although some versions were released for internal purposes at the end of 1971, the first public version became publically available at the end of 1972 and beginning of 1973 [11].

SPICE was announced as general purpose simulation program convenient for simulation of integrated circuits as well. It was written in FORTRAN code (with more than 17000 program lines) and was running on large main frame computers. SPICE1 used nodal analysis to construct the circuit equations. Nodal analysis has limitations in representing inductors, floating voltage sources and the various forms of controlled sources. It had relatively few circuit elements available and used a fixed-timestep transient analysis. SPICE1 could perform: a) nonlinear DC analysis: the result of this analysis is commonly referred to as the DC bias or operating-point characteristic; b) nonlinear transient analysis: computes the voltages and currents with respect to time; c) linear AC analysis: linearises the circuit around the DC operating point and then calculates the output as a function of frequency. The circuit size limitations were: 400 nodes, 200 total number of

elements, of which no more than 100 could be semiconductor devices. It had included built-in models for diodes, bipolar junction transistors (BJTs), junction field-effect transistors (JFETs) and metal-oxide-semiconductor field-effect transistors (MOSFETs). Models for bipolar transistors changed to Gummel-Poon equations [11].



Figure 3. Professor Donald O. Pederson IEEE Life Fellow (1925-2004) - awarded the IEEE Medal of Honor in 1998, "For creation of the SPICE Program, universally used for the computer aided design of circuits."

B. SPICE2

In 1975 SPICE2 was released [12]-[14]. It offered significant improvements comparing to SPICE1.

Modified Nodal Analysis (MNA) (Table 1), replacing the old analysis, began to support voltage sources and inductors. Memory was dynamically allocated to accommodate growing circuit size and complexity. Implementation of the adjustable time-step control speeded the simulation process. MOSFET and bipolar models were overhauled and extended. Version SPICE2G.6 (1981) was the last FORTRAN version (still available today from Berkeley) [15].

Actually, many commercial simulators today are based on SPICE2G.6. That is the reason why SPICE became industry standard simulation tool.

TABLE I. MNA ALGORITHM

| Step | Operation |
|------|---|
| 1 | Apply KCL to each node of the circuit |
| 2 | Apply branch equations to eliminate as many branch currents as possible, leaving branch voltages and some branch currents |
| 3 | Write down the unused branch equations |
| 4 | Apply KVL to replace branch voltages with node voltages |
| 5 | Write equations in partitioned-matrix form |
| 6 | Solve simultaneous equations to obtain the node voltages and branch currents |
| 7 | Back-substitute to obtain the remaining branch currents |

C. SPICE3

In 1985 SPICE3 version has been released. SPICE code was rewritten in the C programming language (1985) [16], [17]. It includes polynomial capacitors, inductors and voltage controlled sources. New version eliminates many convergence problems. It features a graphical interface for viewing results.

The new models added to this version were: MESFET, lossy transmission line and non-ideal switch. Improved semiconductor models accommodate smaller transistor geometries. But SPICE3 was not backward compatible with SPICE2.

D. 1980's and Beyond

In 1980's several commercial versions based on SPICE were released. They include: HSPICE, IS_SPICE and MICROCAP. MicroSim released PSPICE, the first PC version of SPICE. As SPICE began to attract many more users in industry and academia, some companies started to integrate SPICE versions to their schematic entry and layout packages. They have also started to develop and release to public SPICE models for their devices.

IV. SPICE - PRESENT STATE

Today's commercial versions of SPICE satisfy all basic requirements, which should be satisfied by a modern circuit simulator, such as:

- User-friendly interface, which allows the user to describe the circuit, the elements and the simulation conditions as text or by schematics.
- Circuit checking: does the circuit comply with the circuit laws and simulator conventions?
- Equation formulation, which turns the input description of the circuit into a set of equations.
- Operating point determination ("DC analysis"): what is the state of the circuit at $t = 0$?
- "Transient analysis": finding the circuit waveforms from $t = 0$ to $t = t_{END}$.
- The output user interface for presenting the results of the simulation run.
- Capability for multilevel programming;
- Improved power semiconductor device models that can predict accurately the functionality and reliability of a specific electronic circuit.
- Capability for simulation of switch-mode behaviour;
- Variable step during the simulation. To shorten the overall simulation time the possibility of variable time-step is, usually, incorporated in the simulation program.
- Capability to accept the initial condition set by the user – During the analysis of electronics systems with non-linear or switching behaviour (e.g. power electronics circuits) the time of simulation can be reasonably shortened by setting appropriate initial conditions.

Each new improved version has expanded list of built-in components. This enables faster and more accurate operation of the simulation program when dealing with large complicated circuits. The list of built-in components in some of the newest commercial versions of SPICE is given in Table 2.

TABLE II. SYMBOLS OF CIRCUIT ELEMENTS AND SOURCES

| First Letter | Circuit Elements and Sources |
|--------------|-----------------------------------|
| B | GaAs MES field-effect transistor |
| C | Capacitor |
| D | Diode |
| E | Voltage-controlled voltage source |
| F | Current-controlled current source |
| G | Voltage-controlled current source |
| H | Current-controlled voltage source |
| I | Independent current source |
| J | Junction field-effect transistor |
| K | Mutual inductors (transformer) |
| L | Inductor |
| M | MOS field-effect transistor |
| Q | Bipolar junction transistor |
| R | Resistor |
| S | Voltage-controlled switch |
| T | Transmission line |
| V | Independent voltage source |
| W | Current-controlled switch |
| Z | IGBT |

A. SPICE's Basic Characteristics

SPICE can do several types of circuit analyses. The most important are:

- Non-linear DC analysis: calculates the DC transfer curve;
- Non-linear transient and Fourier analysis: calculates the voltage and current as a function of time when a large signal is applied; Fourier analysis gives the frequency spectrum;
- Linear AC Analysis: calculates the output as a function of frequency. A Bode plot is generated;
- Noise analysis;
- Parametric analysis;
- Monte Carlo Analysis.

In addition, PSpice has analog and digital libraries of standard components (such as NAND, NOR, flip-flops, MUXes, FPGA, PLDs and many more digital components). This makes it a useful tool for a wide range of analog, digital and mixed applications.

All analyses can be done at different temperatures. The default is 300K.

Before one can simulate a circuit one needs to specify the circuit configuration. This can be done in a variety of ways. One way is to enter the circuit description as a text file in terms of the elements, connections, the models of the elements and the type of analysis. This file is called the SPICE input file or source file. An alternative way is to use a schematic entry program such as OrCAD CAPTURE or LTSpice. Capture is a user-friendly program that allows you to capture the schematic of the circuits and to specify the type of simulation. Capture is not only intended to generate the input for PSpice but also for PCB layout design programs.

The Simplified block diagram of the main SPICE program flow is shown in Fig. 4.

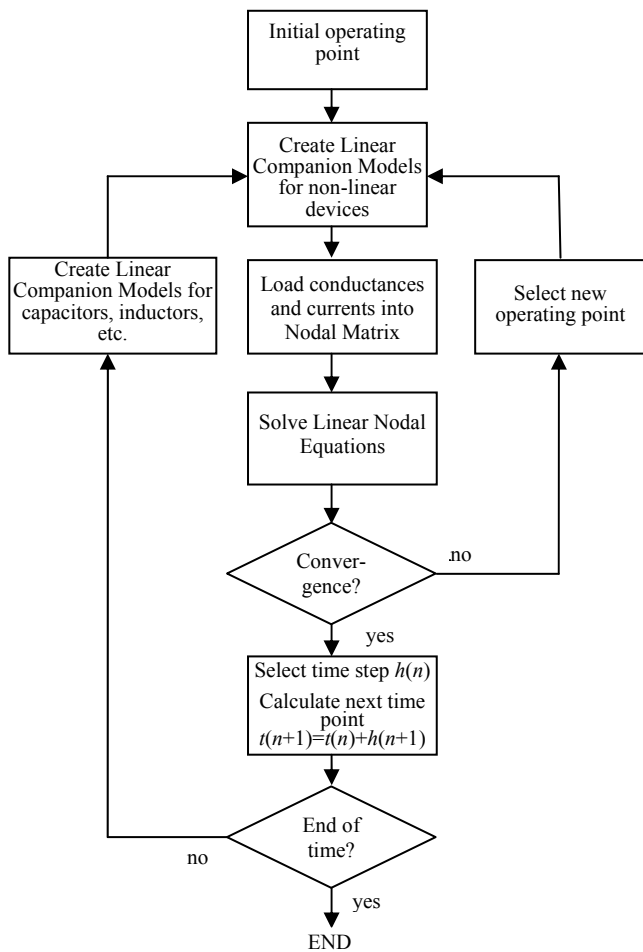


Figure 4. Simplified block diagram of the main SPICE program flow [18]

The three key points about the algorithm are:

- At the heart of the SPICE is Modified Nodal Analysis accomplished by formulating the Nodal Matrix and solving the nodal equations for the circuit voltages.
- The inner loop finds the solution for Non-Linear circuits. Non-Linear devices are replaced by equivalent linear models. It may take many iterations before the calculations converge to a solution.
- The outer loop, together with the inner loop, performs a Transient Analysis creating equivalent linear models for energy-storage components for capacitors, inductors, etc. and selecting the best time points.

Figure 5 summarizes the different steps involved in simulating a circuit with Capture and PSpice.

The output results can be obtained in the form of line printer plot using the .PLOT statement, in the form of tables (print tables) using .PRINT statement, and as a graphic presentation, using .PROBE statement.

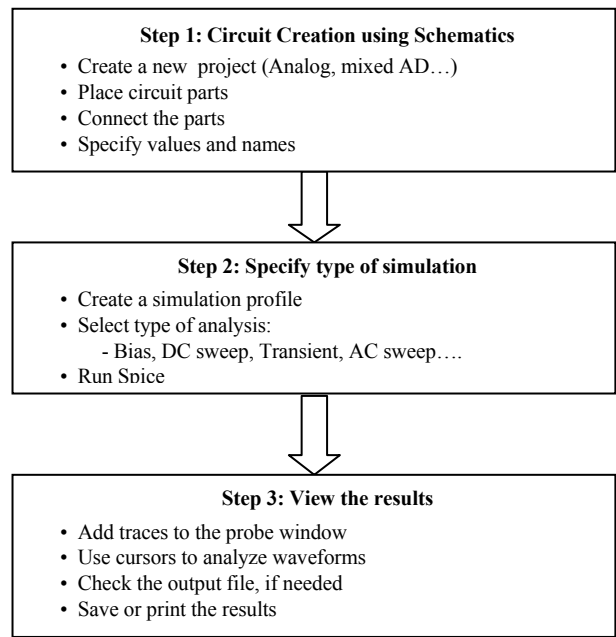


Figure 5. Steps involved in simulating a circuit with PSpice

B. Commercial Versions of SPICE today

Today several companies are offering to the market their own versions of SPICE. Some versions are available for mainframe computers, others for PCs while some companies are releasing versions for mainframes as well as for PCs.

The most popular SPICE versions available on the market for mainframes, as well as the corresponding producers, are shown in Table 3.

TABLE III. THE SPICE FAMILY (MAINFRAME)

| Version | Producer + comment |
|------------|---|
| HSPICE | Meta-Software |
| I-SPICE | NCSS Time Sharing |
| IG-SPICE | A.B. Associates |
| PSpice | Orcad/Cadence |
| RAD-SPICE | Meta-Software - special version for simulation of circuits subjected to ionizing radiation |
| SPICE-Plus | Analog Design Tools |

The number of current SPICE versions available for personal computers is much higher. The first versions were released for DOS operating system. Today, SPICE versions for Windows and only few for Linux, operating systems are available on the market. Some companies are offering free versions for students with limited capabilities. One commercial (complete package without limitations) version, available for Windows and Linux, as well, (LTSpice from Linear Technology) can be purchased free of charge.

A list of most popular commercial SPICE versions for PCs as well as the corresponding companies which are producing them, are shown in Table 3.

TABLE IV. THE SPICE FAMILY (PC)

| Version | Producer + comment |
|--------------|--|
| AIM Spice | AIM Software (for Windows + Linux) |
| All Spice | Acotech |
| B2 Spice v.5 | Beige Bag Software - Emag Technologies |
| DSPICE | Daisy Systems |
| IG-SPICE | A.B. Associates |
| IS-SPICE | Intusoft |
| LTSpice | Linear Technology - free (for Windows + Linux) |
| PSPICE | Orcad/Cadence |
| Spectre | Cadence Design |
| SPICE-Plus | Analog Design Tools |
| SPICE-plus | Valid Logic |
| T Spice | Tanner EDA |
| TOPSPICE | Penzar |
| Z-SPICE | Z-Tech |

V. CONCLUSIONS

The main purpose of this paper is to honor the 40-th anniversary of the Simulation Program with Integrated Circuit Emphasis (SPICE). Since its release in early 1970's SPICE has been serving the professional community in developing, design and production of different kind of electric and electronic circuits. It has been, also, very useful in educational purposes. The author of this simulation program is Laurence "Larry" Nagel. Actually this program represents the basis of his PhD dissertation made at the University of California at Berkeley under the supervision of Prof. Don Pederson. There, Dr. Nagel, on more than 400 pages, describes in details the SPICE improved version (SPICE2). Since its appearance SPICE has been improved many times and today it is available for mainframes, as well as for PCs, with different operating systems. It can simulate different type of circuits working in linear, nonlinear, or pulse mode, as well as digital and mixed circuits. It is capable of performing very different types of circuit analyses such as: non-linear DC analysis, non-linear transient and Fourier analysis, linear AC analysis, noise analysis, parametric analysis, Monte Carlo analysis, etc.

That's the reason why it became an industry standard simulation tool indispensable for design of electric and electronic circuits, and is used in academia and industry.

Today's SPICE is available in several versions produced by different companies. It is improving and updating continuously since its appearance. There are different open source codes available free of charge which can be modified by the users themselves. Besides SPICE is fairly general purpose and is not tied to any particular technology or component.

Knowing all that, it is easy to conclude that SPICE will play significant role in the design of integrated circuits for a long time. According to Larry Nagel, SPICE will not be used to simulate billion transistor circuits, but it will play a key role in developing the devices, device models, building blocks, and behavioral models for the building blocks for the billion transistor chips. SPICE will continue to play important role of the foundation of the integrated circuit design [19].

Happy 40th anniversary and successful future SPICE.

REFERENCES

- [1] Banzhaf, "Computer-Aided Circuit Analysis Using SPICE", Prentice-Hall, Inc, 1989
- [2] ECAP II Application Description Manual (GH20-0983),
- [3] F. H. Branin, JR., G. R. Hogsett, R. L. Lunde, and L. E. Kugel, "E2AP II—A New Electronic Circuit Analysis Program" IEEE Journal of Solid-State Circuits, Vol. SC-6, No. 4, August 1971, pp. 146–166.
- [4] "Automated digital computer program for determining the response of electronic systems for transient nuclear radiation (SCEPTRE)", Air Force Weapons Lab., Res. Tech. Div., Air Force Systems Command, Kirtland, N. Mex, 87117, AFWL-TR-69-77.
- [5] L. D. Milliman, W. A. Massena, R. H. Dickhaut, "CIRCUS—A digital computer program for transient analysis of electronic circuits—User's guide" Boeing Co., Harry Diamond Lab., Rep. AD-34-1, Jan. 1967.
- [6] E. D. Johnson, C. T. Kleiner, L. R. McMurray, E. L. Steele, and F. A. Vassallo, "Transient radiation analysis by computer program (TRAC)", Automatics Div., North American Rockwell Corp., Anaheim, Calif., Harry Diamond Lab., Tech. Rep., June 1968.
- [7] H. Schichman, "Integration system of a nonlinear network analysis program", IEEE Trans. Circuit Theory, vol. CT-17, Aug. 1970, 378-386.
- [8] L. W. Nagel, "CANCER: Computer Analysis of Nonlinear Circuits Excluding Radiation," Masters Report, Dept of EECS, Univ. of California, Berkeley, CA, December 11, 1970.
- [9] R. A. Rohrer, L. W. Nagel, R. Meyer, and L. Weber, "CANCER: Computer Analysis of Nonlinear Circuits Excluding Radiation," 1971 IEEE Intl Solid-State Circuit Conference, Philadelphia, PA, February 18, 1971, pp. 124-125, ieeexplore.ieee.org/search/srchabstract.jsp
- [10] L. Nagel and R. Rohrer, "Computer Analysis of Nonlinear Circuits, Excluding Radiation (CANCER)," IEEE J Solid-State Circuits, Vol SC-6, No 4, August 1971, pp. 166-182, ieeexplore.ieee.org/xpl/freeabs_all.jsp
- [11] L. W. Nagel and D. O. Pederson, "Simulation Program with Integrated Circuit Emphasis (SPICE)," presented at 16th Midwest Symp. on Circuit Theory, Ontario, Canada, April 12, 1973 and available as Memorandum No ERL-M382, Electronics Research Laboratory, College of Engineering, University of California, Berkeley, CA, www.eecs.berkeley.edu/Pubs/TechRpts/1973/ERL-382.pdf
- [12] L. W. Nagel, "SPICE2: A Computer Program to Simulate Semiconductor Circuits," PhD dissertation, Univ. of California, Berkeley, CA, May 9 1975 and available as Memorandum No, ERL-M520, Electronics Research Lab., College of Engg, Univ. of California, Berkeley, CA, www.eecs.berkeley.edu/Pubs/TechRpts/1975/ERL-520.pdf
- [13] E. Cohen, "Program Reference for SPICE2," University of California, Berkeley, ERL Memo UCB/ERL M75/520, May 1975, www.eecs.berkeley.edu/Pubs/TechRpts/1976/ERL-592.pdf
- [14] A. Vladimirescu and S. Liu, "The Simulation of MOS Integrated Circuits Using SPICE2," University of California, Berkeley, UCB/ERL M80/7, October 1980, www.eecs.berkeley.edu/Pubs/TechRpts/1980/ERL-80-7.pdf
- [15] A. Vladimirescu, K. Zhang, A. R. Newton, D. O. Pederson and A. L. Sangiovanni-Vincentelli, "SPICE Version 2G User's Guide," Dept. of EECS, University of California, Berkeley, August 1981.
- [16] T. L. Quarles, "SPICE3 Version 3C1 User's Guide," University of California, Berkeley, ERL Memo UCB/ERL M89/46, April 1989, www.eecs.berkeley.edu/Pubs/TechRpts/1989/ERL-89-46.pdf
- [17] <http://www.ecircuitcenter.com/spicetopics/history.htm>
- [18] <http://www.ecircuitcenter.com/SpiceTopics/Overview/Overview>
- [19] L.W. Nagel, C.C. McAndrew, "Is SPICE good enough for tomorrow's analog?", Proc IEEE 2010 Bipolar/BiCMOS Circuits and Technology Meeting (BCTM), 2010, 106 - 112