Streamlined Circuit Device Model Development with *f*REEDA™ and ADOL-C

Frank P. Hart[†], Nikhil Kriplani[†], Sonali Luniya[†],
Carlos E. Christoffersen[⋆] and Michael B. Steer[†]
[†]Department of Electrical and Computer Engineering,
North Carolina State University, Raleigh, NC 27695-7914
[⋆] Department of Electrical Engineering, Lakehead University,
955 Oliver Road, Thunder Bay, Ontario P7B 5E1, Canada

Extended Abstract

April 9, 2004

Abstract

Time-marching simulation of electronic circuits, using quadrature methods, with the U.C. Berkeley program Spice and variants has been a standard practice for electronics engineers since the mid-1970s. The development time for a semiconductor device model in Spice may be lengthy because the model equations and its derivatives must be coded manually. In addition, several files in the source tree must be modified to define a new model. *f*REEDA™, an object-oriented circuit simulator under development at North Carolina State University and other universities, overcomes many limitations of the Spice model development paradigm. A key to this implementation is the ADOL-C package, which is used to automatically evaluate the derivatives of the device model equations. As a result models are more compact and the development time is shorter. The development history of selected Spice models and their *f*REEDA™ counterparts is presented to illustrate the advantages of this approach.

1 History and Motivation for Circuit Simulation

Computer-aided simulation of electronic circuits has been common since the mid-1970s, when the U.C. Berkeley program Spice [1] was made available to an electronics industry that was increasingly engaged in the development and manufacture of solid-state integrated circuits based on semiconductor devices. Spice itself is an acronym for **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis, and this longer name bespeaks the purpose which drove the development of the program: Electronic circuits based on vaccuum tube technology had existed for decades, and the cost of producing prototypes of a new vaccuum tube circuit design was not prohibitive. Once well-validated in a laboratory environment – possibly with design changes incorporated to make the design robust – the vaccuum tube design was manufactured in volume.

Solid-state integrated circuits supplanted vaccuum tube technology because of their much higher reliability and comparatively inexpensive cost of high-volume manufacturing. However, solid-state electronic circuits in integrated form are not amenable to prototype evaluation in a laboratory, for two reasons. First, manufacturing semiconductors in small quantities is always an expensive proposition since high fixed costs of manufacturing are spread over a low volume. This is still often done so that the integrated circuits can be evaluated in higher-level assemblies, but it is only economically feasible when circuit simulation in the design process minimizes the risk of faulty circuit design, thus minimizing the possibility of repeated cycles of low-volume prototype manufacturing. Second, once a circuit is fabricated in a form etched into silicon, the behavior of its internally interconnected devices cannot be easily observed with the kind of laboratory measuring equipment that electronic engineers are accustomed to using. Simulation of electronic circuits prior to manufacture in solid-state integrated form was thus not only desirable, but had become an economic necessity. Over the years, Spice has been ported successfully to generations of less expensive computers, and today it is both economical and common to simulate even circuits implemented on printed circuit boards with discrete component elements prior to building prototype models.

2 Overview

2.1 Circuit Elements and Constitutive Equations

Computer-aided analysis in its most basic form is comprised of two kinds of equations. The first are the *constitutive* element equations which usually give voltage as a function of current or vice versa for a particular element. For example, resistors are simple impedance elements described using Ohm's law, $v(t) = Ri(t)$ [2]. Other linear elements like capacitors and inductors are dynamic and described with differential equations. Nonlinear elements such as semiconductor diodes and transistors are also described with a set of constitutive equations [3]. An abstract element form called a 2-port [4] allows for the arbitrary definition of transfer function relationships between two pairs of conductors, and the generalization to an N-port allows for the arbitrary definition of transfer functions from each of N ports to $N-1$ other ports. Port transfer functions are usually provided in matrix form. Finally, circuit simulation often uses two other abstract forms known as ideal voltage and current sources. These are important in equivalent circuit modelling of real devices.

2.2 Network Equations

The second kind of equation that is essential for computer-aided circuit analysis is one that governs the interconnection of the elements. These equations are known as *network* equations, and there are two forms of them. One form is governed by Kirchoff's Current Law (KCL), which states that the sum of currents into each node (a point of interconnection) must be zero. The second form is governed by Kirchoff's Voltage Law (KVL), which states that the sum of voltages around any closed loop of interconnected elements must be zero. For any electronic circuit, the collection of KCL equations for all nodes or KVL equations for all loops can be cast in matrix form. All forms of circuit simulation require the formulation of a matrix describing the interconnection of circuit elements. The theory governing the matrix formation is well-developed and rooted in graph theory [4, 5]. Owing to the simplicity of formulation, the KCL form of the equations is preferred, and so the KVL form will not be discussed further here. The KCL form of the matrix equation is called a Modified Nodal Admittance Matrix (MNAM) [6].

2.3 Forms of Circuit Simulation

The two best-known and longest established forms of circuit simulation are known as *transient* analysis and *AC* analysis. Both forms were available in the first release of Spice. Transient analysis is time-domain simulation of the circuit using quadrature integration. Linear and nonlinear differential circuit equations are discretized using either the backward Euler or trapezoidal interpolating functions, and the integration from one time step to the next is governed by an iterative procedure using either Newton's method or the minimization of some error function. AC analysis is frequency-domain simulation of the circuit with frequency steps. An algebraic frequency-domain form, given by the Laplace transform of the constitutive equations, is used, and the network equations are also cast in MNAM form. AC analysis is not supported in *f*REEDA$^{\text{TM}}$, so it will not be discussed further. Underlying both analysis forms is a form known as *DC* analysis, which is used to establish the initial operating conditions for both both transient and AC analysis.

One other popular form of circuit simulation that is not present in Berkeley's Spice but has been made available in *f*REEDA$^{\text{TM}}$ [7] and other simulators [8, 9] is called *Harmonic Balance* analysis (HB). HB is an implementation of Galerkin's methods [10] for finding the steady-state response of a nonlinear network. The network equations governing HB are created by dividing the network into linear and nonlinear portions and equating them at the boundaries. HB will not be discussed in depth, but its method for formulating the MNAM has also been applied to *f*REEDA$^{\text{TM}}$transient analysis [11], and this will be further discussed. This MNAM construction technique differs drastically from that used by Spice, and this difference is one of the consequences of the different philosophies guiding model implementation in the two simulator environments.

3 Transient Circuit Simulation Modelling

3.1 Spice and Associated Discrete Modelling

A 1984 publication reviewing the history of circuit simulation [12] credits Rohrer and a group of graduate students with finding that nonlinear circuit elements could be modelled in the time domain by permitting an equivalent linear circuit of resistors and ideal current sources to have their model values updated not just at time steps within the simulation, but also at at different iterates of a Newton iterative loop at a given time step. (Partial disclosure of this technique was given in [13].) This equivalent circuit model was first described as the "Associated Discrete Model" in the literature [14], but more recently it has been termed simply a "companion model." Companion modelling became the standard method for implementing a nonlinear device model in Spice, and it remains unchanged to this day.

Among the advantages of companion modelling are the simplicity of reducing nonlinear behavior normally described by differential equations to a simple algebraic form. However, the algebraic model form must be obtained by mathematical analysis and inserted into the model code, and this form is not always easily obtained. Furthermore, some models which received widespread use for specific purposes were later found to have shortcomings. For example, some earlier Metal Oxide Semiconductor Field Effect Transistor (MOSFET) Spice models did not conserve charge [15, 16], thus rendering them suspect to semiconductor physicists. Moreover, an unfortunate consequence of updating model values as a simulation progresses is that the MNAM contents change at every time step and at every Newton iterate within a time step, and so the computationally expensive LU factorization performed on the MNAM gets no reuse. Philosophically, the companion model is a mathematical abstraction that risks departing from the underlying physics of the device in order to facilitate a simulation method consisting of equivalent circuit models with very few elements.

3.2 fREEDA™and State Variable Based Modelling Using ADOL-C

Time-marching simulation using state variable-based models was also initially developed in the 1960s [17]. Such simulators formed and solved systems of differential rather than algebraic equations. They fell out of favor because the Spice companion model led to purely algebraic systems of equations and because the companion model formulation also led to MNAMs that were inherently more sparse than the state variable approaches. Interest in this approach was renewed when researchers in the discipline of microwave engineering became interested in combining a device's interactions with electromagnetic fields [18]. SPICE analyses are limited to voltage and currents only, and so a new simulator environment based on state variable analysis was created to permit this form of analysis. This initial effort has evolved into *f*REEDA™.

In *f*REEDA™, device element models are faithful replicas of the physical equations describing the device. State variables become ADOL-C active variables [19] so that their derivatives can be conveniently used in the model code. Through the use of object-oriented programming techniques, all device models are derived C++ classes that inherit the characteristics of a C++ base class [20] The base class contains the interface to ADOL-C for initializing and manipulating the ADOL-C 'tapes' of active variables. Absent ADOL-C, a state variable approach would require manual coding of the derivatives and time-delayed versions of the state variables. Thus the importance of the ADOL-C package to facilitating this state variable approach in *f*REEDA™cannot be understated.

4 Spice and fREEDA™Models

4.1 Illustrative Comparison of Diode Models

The development of models for the semiconductor diode makes for an illustrative comparison. Aspects of the Spice model development are described in [21], while the *f*REEDA™model is described in [22]. Materials from both sources will be described, using both figures and equations, with detail sufficient to demonstrate the advantages and disadvantages of both approaches.

4.2 The Berkeley Short-channel IGFET Model Version 4 (BSIM4)

The Device Group at the University of California at Berkeley has been at the forefront of specifying semiconductor physics models for field effect transistors in the most advanced semiconductor process technologies. The first Field Effect Transistor (FET) models were published in 1968 and had only 41 parameters to fully describe any transistor. In the 1980s, as semiconductor process technologies continued to shrink the minimum size of their features, other physical phenomena were observed which necessitated the addition of more parameters to the models, and the BSIM models were created. In 2000, a fourth major version of the BSIM models called BSIM4 was released. This semiconductor physics model has over 200 parameters. The contemporary nature of BSIM4 thus makes it a good device upon which to do a case study comparison of a Spice model with a *f*REEDA™model. Such a study has was completed by one of the authors in 2002, who implemented the BSIM4 model in *f*REEDA™as a Master's Thesis [23].

Starting with the BSIM4 semiconductor physical model documentation, the model consisted of about 1500 lines of C++ code, fitting on 25 pages, and required 7 months to develop. Much of this time was spent implementing code structures to support the BSIM models for the first time. It is not known how much time was required to create the Spice companion model (and its developers may well have leveraged the previous BSIM implementations), but in Spice 3f5, which was also released by U.C. Berkeley, the companion model required 200 pages of C code and thousands of lines of code to implement. It should be noted that Berkeley's MOSFET models are not the only ones available, and implementing other models has proven to be less labor-intensive. For example, the EPFL-EKV model [24] was implemented by one student [25] as a semester project for a circuit simulation class at NC State University.

4.3 fREEDA™Universal Modelling Approach - BJT Example

A universal state variable-based modelling approach was proposed in [26]. The central notion of this approach is to choose as state variables those quantities most appropriate to accurately modelling the physical device, and then derive any additional required variables through a dependency hierarchy. In [26] a Bipolar Junction Transistor (BJT) model that conserves charge through the use of three ADOL-C tapes is described. The development of this model will be illustrated with figures and equations. A similar hierarchical state-variable approach is advocated for all devices.

References

[1] L.W. Nagel and R.A. Rohrer. SPICE2, A computer program to simulate semiconductor circuits. Technical Report ERL-M520, University of California at Berkeley, May 1975.

[2] D. Halliday and R. Resnick. *Fundamentals of Physics.* John Wiley and Sons, 2 edition, 1981.

[3] B.G. Streetman. *Solid-State Electronic Devices.* Prentice-Hall, 2 edition, 1980.

[4] C.A. Desoer and E.S. Kuh. *Basic Circuit Theory.* McGraw-Hill, 1969.

[5] J. Vlach and K. Singhal. *Computer Methods for Circuit Analysis and Design.* Van Nostrand Reinhold, 1994.

[6] C.W. Ho, A.E. Ruehli, and P.A. Brennan. The modified nodal approach to network analysis. *IEEE Transactions on Circuits and Systems*, CAS-22(6):504–509, June 1975.

[7] C.E. Christoffersen, M.B. Steer, and M.A. Summers. Harmonic balance analysis for systems with circuit-field iterations. In *Microwave Symposium Digest*, volume 1, pages 1131–1134. IEEE MTT-S, IEEE, 1998.

[8] K.S. Kundert, J.K. White, and A. Sangiovanni-Vincentelli. *Steady-State Methods for Simulating Analog and Microwave Circuits.* Kluwer Academic Publishers, 1990.

[9] P.J.C. Rodrigues. *Computer-Aided Analysis of Nonlinear Microwave Circuits.* Artech House, 1998.

[10] A. Quarteroni, R. Sacco, and F. Saleri. *Numerical Mathematics*, volume 37 of *Texts in Applied Mathematics*. Springer-Verlag, New York, 2000.

[11] C.E. Christoffersen. *Global Modeling Of Nonlinear Microwave Circuits*. PhD thesis, North Carolina State University, 2000.

[12] D.O. Pederson. A historical review of circuit simulation. *IEEE Transactions on Circuits And Systems*, CAS-31(1):103–111, January 1984.

[13] W.J. McCalla and W.G Howard. Bias-3 - a program for the nonlinear dc analysis of bipolar transistor circuits. *IEEE Journal of Solid-State Circuits*, SC-6(1):14–19, February 1971.

[14] L.O. Chua and P.M. Lin. *Computer-Aided Analysis of Electronic Circuits: Algorithms & Computational Techniques*. Prentice-Hall, 1975.

[15] M.A. Cirit. The Meyer model revisited: Why is charge not conserved? *IEEE Transactions on Computer-Aided Design*, 8(10):1033–1037, October 1989.

[16] P. Yang, B.D. Epler, and P.K. Chaterjee. An investigation of the charge conservation problem for mosfet circuit simulation. *IEEE Journal of Solid-State Circuits*, SC-18(1):128–139, February 1983.

[17] C. Pottle. A "textbook" computerized state-space network analysis algorithm. *IEEE Transactions on Circuit Theory*, 16(4):566–568, November 1969.

[18] F. Nusseibeh, T.W. Nuteson, J. Patwardhan, M.A. Summers, C.E. Christoffersen J. Kreskovsky, and M.B. Steer. Computer-aided engineering environment for spatial power combining systems. In *Microwave Symposium Digest*, volume 1, pages 1073–1076. IEEE MTT-S, IEEE, 1997.

[19] A. Griewank, D. Juedes, H. Mitev, J. Utke, O. Vogel, and A. Walther. ADOL-C: A package for the automatic differentiation of algorithms written in C/C++. Technical report, March 1999.

[20] C.E. Christoffersen, U.A. Mughal, and M.B. Steer. Object oriented microwave circuit simulation. *International Journal of RF and Microwave Computer-Aided Engineering*, 10(3):164–182, May 2000.

[21] L.T. Pillage, R.A. Rohrer, and C. Visweswariah. *Electronic Circuit System Simulation Methods*. McGraw-Hill, 1995.

[22] C.E. Christoffersen. State variable harmonic balance simulation of a quasi-optical power combining system. Master's thesis, North Carolina State University, 1998.

[23] N.M. Kriplani. Transistor modeling using advanced circuit simulator technology. Master's thesis, North Carolina State University, 2002.

[24] M. Bucher, C. Lallement, C. Enz, F. Theodoloz, and F. Krummenacher. The EPFL-EKV MOSFET model equation for simulation. Technical Report Model Version 2.6, Swiss Federal Institute of Technology (EPFL), Lausanne, Switzerland, June 1997.

[25] W. Jang. Ekv.doc. See the Mosnekv.cc and Mosnekv.h files in a *f*REEDA™ installation, April 2003.

[26] C.E. Christoffersen, S. Velu, and M.B. Steer. A universal parameterized nonlinear device model formulation for microwave circuit simulation. In *Microwave Symposium Digest*, volume 1, pages 2189–2192. IEEE MTT-S, IEEE, 2002.