Books for SPICE-Programs

Bücher für SPICE-Programme

January 23, 2003

Hans Gall

BAUSCH-GALL GmbH

www.Bausch-Gall.de

A lot of literature for SPICE-based simulators was written since about 1970. With this overview, we want to inform you about technical books in english or german which proved to be useful for the simulation of electronic circuits. The books are briefly described in the language in which they are written. Books with a given price in EUR (without V.A.T.) are available directly from BAUSCH-GALL GmbH.

Im Lauf der Jahre ist sehr viel Literatur zu SPICE-basierten Simulatoren entstanden. Mit dieser Übersicht wollen wir Sie auf deutsch- und englischsprachige Fachbücher hinweisen, die sich für die Elektroniksimulation als nützlich erwiesen haben. Die folgenden Kurzbeschreibungen sind in der Sprache verfaßt, in der das jeweilige Buch geschrieben wurde. Bücher bei denen ein Preis (ohne MwSt.) angegeben ist, können Sie von der BAUSCH-GALL GmbH beziehen.

SPICE2: A Computer Program to Simulate Semiconductor Circuits (ERL-M520) [1]

This report is the most often cited reference about SPICE in the technical and scientific literature for circuit simulation. The report describes the numerical methods which are the necessary ingredients of a circuit analysis program. The theoretical and practical aspects of SPICE2 are also documented.
Price: EUR 27.00 for the fotocopy

SPICE - Analyseprogramm für elektronische Schaltungen [2]
Untertitel: Benutzerhandbuch mit Beispielen

Dieses Buch ist ein Standardwerk zur Einführung in die Anwendung von SPICE. Der gesamte Sprachumfang der Standardversion SPICE 2G.6 wird ausführlich erläutert und mit vielen Beispielen veranschaulicht. Inhalt: Einführung, Schaltelemente, Analysearten, Beschreibung der internen Halbleitermodelle, Teilschaltungen, Optionen, Literaturhinweise. Das Buch ist auch für PSpice- und SPICE3-Anwender zur Einarbeitung geeignet.
Dieses Buch ist leider vergriffen!

Semiconductor Device Modeling with SPICE [3]

This book features updated and expanded coverage on using SPICE-based programs to simulate and accurately predict the performance of electronic circuits. As the only book on SPICE to describe the device models themselves rather than how to run SPICE software, this reference book explains a step-by-step analysis of the modeling process and provides complete information on the various semiconductor devices, including: pn-junction and Schottky diodes, bipolar junction transistor (BJT), junction field-effect transistor (JFET), metal-oxide-semiconductor transistor (MOST), metal-semiconductor field-effect transistor (MESFET), ion-sensitive field-effect transistor (ISFET), semiconductor-controlled rectifier (SCR-thyristor). The book also describes the models implemented in the widely used SPICE-based programs such as SPICE3, HSPICE and PSpice plus a new appendix on metal-semiconductor junction theory.
Price: EUR 72.00

Inside SPICE: Overcoming the Obstacles of Circuit Simulation [4]

Written by a leading SPICE expert, this hands-on reference tool goes beyond the basics to tackle the actual problems you face while using SPICE or any of the other SPICE-like simulator programs such as HSPICE, PSPICE, IsSpice and MICROCAP IV. Packed with sample circuit simulations that illustrate common problems and their solutions, this in-depth guide shows you how to make simulations faster and more efficient by setting the .OPTION-parameters to appropriate values. The book also provides step-by-step coverage of how to overcome such stumbling blocks as nonconvergence, numeric integration instabilities and timestep control errors. With the book comes a disk with copies of RSPICE (PC-32bit-extended-memory Version of SPICE 2G.6) and RGRAPH, a graphical postprocessor designed to work with RSPICE. The disk also contains simulation examples from the book so that the reader can reproduce them with RSPICE and RGRAPH.
Price: EUR 67.00

Fundamentals of Computer-Aided Circuit Simulation [5]

This book is intended as both an introduction to and a quick summary of those numerical techniques which have been found to be relevant to circuit simulation. As such it is suitable for use by advanced undergraduate students, graduate students and practicing engineers. At least a rudimentary understanding of calculus (derivatives and Taylor series), linear algebra (systems of linear equations), numerical analysis (nonlinear equation solution and numerical integration) and basic circuit and semiconductor device theory are desirable for but not essential to an understanding of the material presented. This book describes the formulation and implementations of most of the various numerical techniques for circuit simulation programs. Methods are compared where possible. Some of the methods have been used extensively while others have not been used at all. The intent is to cover this material at an intuitive and understandable level.
Price: EUR 88.00

SPICE, A Guide to Circuit Simulation and Analysis using PSpice [6]

Introductory-level textbook for new users explaing all analysis types of PSpice. Contents: Introduction, Getting Started, DC Operation, SC Sensitivity, DC Sweep, Transfer Function, Frequency Response, Feedback Control Analysis, Noise Analysis, Transient Response, Distortion and Spectral Analysis, Device Models, Active Devices, Appendix.

Simulation integrierter Schaltungen [7]

Dieses Buch zeigt Verfahren und Praxis der rechnergestützten Simulation nichtlinearer Schaltungen auf der Ebene der Spannungen und Ströme unter Berücksichtigung von diskreten Elementen, der Hybridintegration und der monolithischen Höchstintegration. Der Leser erfährt, was machbar ist und wie, warum die Simulationgerade in dieser und keiner anderen Weise durchgeführt wird, was besonders zu beachten ist, um effizient und ökonomisch arbeiten zu können. Dargestellt sind insbesondere die in modernen Programmen verwendeten numerischen Verfahren und Algorithmen.
Preis: EUR 35,00

Entwurf und Simulation von Halbleiterschaltungen mit PSpice [8]

Inhalt: Physik und Technologie der Mikroelektronik, PSpice, Transistormodelle, Vierpol- und Signalflußmethode, rechnergestützer Entwurf von Elektronikschaltungen mit PSpice. Das Buch stellt die Handhabung von PSpice dar und erläutert den Umgang mit dem Programm an Hand praktischer Übungen. Es beschreibt weiterhin Entwurf und Simulation von breitbandigen, rauscharmen und klirrarmen Verstärkern.
Preis: EUR 67,00

Elektronische Schaltungen und Systeme [9]
Untertitel: simulieren, analysieren und optimieren mit SPICE

Dieses Buch wendet sich an Entwicklungsingenieure und Studierende, die sich mit der Schaltungspraxis vertraut machen wollen. Inhalt: Einführung, Schaltungsbeispiele Analogtechnik, Schaltungsbeispiele Digitaltechnik, analoge Rechenschaltungen, kontinuierliche und diskontinuierliche Systeme, Hochfrequenz- und Mikrowellenschaltungen. Das Buch enthält neben den Simulationsergebnissen auch theoretische Erläuterungen und Berechnungsformeln.
Preis: EUR 31,00

Simulating with SPICE [10]

This book is for those interested in learning about analog simulators based on Berkeley SPICE 2G.6. It includes a tutorial for novice users, example problems with helpful hints and an advanced techniques section with application notes. A complete SPICE syntax guide with examples is also included. Contents: Introduction, SPICE Syntax, Understanding SPICE, Advanced Techniques, Application Notes, Benchmark Circuits, Bibliography.
Preis: EUR 66.00

Spice Applications Handbook, 2nd Edition [11]

Since June 1986, *intusoft* has been providing free technical information on SPICE via the *intusoft*-Newsletter. A compilation of all the back issues of the Newsletter is available as the Spice Applications Handbook, 2nd Edition. It contains 34 newsletters dating from 6/86 to 2/94. Over 60 technical articles are included covering a wide range of applications. The Handbook also contains valuable SPICE simulation tips and techniques, device modeling and actual SPICE models for a wide variety of components. A floppy disk is included with the Handbook and it contains all of the models, SPICE netlists and associated SpiceNet schematics from all of the newsletters.
Price: EUR 51.00

A Spice Cookbook [12]

This book contains over 100 practical circuit examples encompassing a wide array of topics (RF, power, filters, digital, microwave). Each example includes a technical explanation, SPICE simulation tips, models, netlists, schematics and output graphs. The SPICE models, schematics and netlists are included on a floppy disk. A Spice Cookbook is perfect for engineers, novice SPICE users, teachers and students. Descriptions of basic SPICE simulations are presented throughout the book providing a guide for future simulations.
Price: EUR 51.00

Handbuch der Electronic Design Automation [13]

Dieses Buch ist ein unentbehrlicher Ratgeber für Praxis und Ausbildung und sollte an keinem CAE-Arbeitsplatz fehlen. Es dient all denen, die in der industriellen Praxis mit der Entwicklung moderner Elektronik betraut sind. Dazu enthält es eine Fülle von aktuellen, kompakten und praxisnahen Informationen über das Gebiet. Behandelt werden: der Systementwurf von ASIC mit den aktuellen CAE-Methoden, höhere Entwurfssprachen wie VHDL, Modellierung im analogen wie digitalen Bereich, programmierbare Logikschaltungen, Aspekte der Plazierung und Verdrahtung, Leiterplattenentwurf und MCM-Technik sowie Testen von Schaltungen während der Entstehungsphasen. Das Handbuch gliedert sich in Entwurfseingabe, Verifikation und Physikalische Implementation und enthält ein EDA-Tutorium in dem ein Beispielentwurf auf unterschiedliche Art bearbeitet und gelöst wird. Ein umfangreicher Anhang gibt Auskunft zu Institutionen, Verbänden Abkürzungen und Begriffen. Das EDA-Handbuch schafft eine vereinheitlichte aktualisierte Wissenbasis für die EDA-Anwendung. Es bleibt bei aller Komplexität des Themas verständlich und durch viele Illustrationen anschaulich ohne auf wissenschaftliche Exaktheit zu verzichten. Die einzelnen Autorenbeiträge sind in sich konsistent und mit eigenen Literaturhinweisen versehen. Zahlreiche Beispiele auch mit Zahlenwerten sowie realistische Simulationsergebnisse von Entwürfen der Autoren vermitteln Praxisnähe und Authentizität und eignen sich auch als Vorlage für ähnliche Aufgabenstellungen des Lesers.

THE SPICE BOOK [14]

This new book, written by Andrei Vladimirescu, who was instrumental in the development of SPICE at the University of California Berkeley, introduces computer simulation of electrical and electronics circuits based on the SPICE standard. Relying on the functionality first supported in SPICE 2 that is now supported in all SPICE programs, this text is addressed to all users of electrical simulation. The approach to learning circuit simulation is to interpret simulation results in relation to electrical engineering fundamentals; the book asks the student to solve most circuit examples by hand before verifying the results with SPICE.

Addressed to both the SPICE novice and the experienced user, the first six chapters provide the relevant information on SPICE functionality for the analysis of linear as well as nonlinear circuits. Each of these chapters starts out with a linear example accessible to any new user of SPICE and proceeds with nonlinear transistor circuits. The latter part of the book goes into more detail on such issues as functional and hierarchical models, distortion analysis, basic algorithms in SPICE and related options parameters, and, how to direct SPICE to find a solution when it does not converge to a solution.

- The approach emphasizes that SPICE is not a substitute for knowledge of circuit operation but a complement. The SPICE book is different from previously published books in the approach of solving circuit problems with a computer. The solution to most circuit examples is sketched out by hand first and followed by a SPICE verification. For more complex circuits it is not feasable to find a solution by hand but the approach stresses the need for the SPICE user to understand the results.
- Readers gain a better comprehension of SPICE thanks to the importance placed on the relation between EE fundamentals and computer simulation. The tutorial approach advances from the hand solution of a circuit to SPICE verification and simulation results interpretation.
- This book teaches the approach to electrical circuit simulation rather than a specific simulation program. Examples are simulated alternatively with SPICE 2, SPICE 3 or PSPICE.
- Accurate descriptions, simulation rationale and cogent explanations make this an invaluable reference.

Analog Integrated Circuits for Communication [15]

This book deals with the analysis and design of analog integrated circuits that form the basis of present-day communication systems. The material is intended to be a textbook for class use but is also a valuable source of information for the practicing engineer. Both bipolar and MOS transistor circuits are analyzed and many numerical examples are used to illustrate the analysis and design techniques developed in the book. A set of problems is presented at the end of the book which covers the subject matter of the whole book.

The approach take is as follows: first-order analysis techniques are developed first, using basic principles and simple device models. Then circuit simulation is used to corroborate the analysis techniques. This procedure provides insight into the operation of circuits and a systematic way of getting an initial design of a circuit. The circuit simulation program SPICE has been extensively used to verify the results of first-order analyses, and for detailed simulations with complex device models. In this manner the student can appreciate the shortcomings of the hand analysis and can resort to simulations when necessary. Simulation results can only be interpreted once one has an understanding of how a circuit operates and this is reflected by the manner in which the material is presented. SPICE input files are given for all the circuits that have been analyzed so that the students can quickly duplicate the input file and verify the results.

References

[1] Laurence W. Nagel,
"SPICE2: A Computer Program to Simulate Semiconductor Circuits", Memorandum No. ERL-M520, 9 May 1975,
Electronics Research Laboratory, College of Engineering,
University of California, Berkeley, CA 94720,
web: `www.eecs.berkeley.edu`

[2] E.E.E. Hoefer, H. Nielinger,
"SPICE, Analyseprogramm für elektronische Schaltungen",
Benutzerhandbuch mit Beispielen,
Springer-Verlag Berlin, Heidelberg, 1985,
ISBN 3-540-15160-5

[3] Paolo Antognetti, Giuseppe Massobrio,
"Semiconductor Device Modeling with SPICE", Second Edition, 1993,
McGraw-Hill Book Company, Inc., New York,
ISBN 0-07-002469-3

[4] Ron Kielkowski,
"Inside SPICE, Overcoming the Obstacles of Circuit Simulation",
Second Edition,
McGraw-Hill Book Company, Inc., New York, 1998,
ISBN 0-07-913712-1

[5] William J. McCalla,
"Fundamentals of Computer-Aided Circuit Simulation",
Kluwer Academic Publishers, Boston, 1988,
ISBN 0-89838-248-3

[6] Paul W. Tuinenga, MicroSim Corporation,
"SPICE, A Guide to Circuit Simulation and Analysis using PSpice",
Prentice-Hall, Inc., Englewood Cliffs, NJ 07632, U.S.A., 1988,
ISBN 0-13-834607-0

[7] Hans Spiro,
"Simulation integrierter Schaltungen",
Verfahren und Praxis der rechnergestützten Simulation nichtlinearer Schaltungen,
R. Oldenbourg Verlag GmbH, München, 1990, 2. Auflage,
ISBN 3-486-21660-0

[8] H. Khakzar, A. Mayer, R. Oetinger, G. Kampe, R. Friedrich,
"Entwurf und Simulation von Halbleiterschaltungen mit PSPICE",
Physik und Technologie der Mikroelektronik, PSPICE, Transistormodelle, Vierpol- und Signalflußmethode, rechnergestützer Entwurf von Elektronikschaltungen mit PSPICE,
Kontakt & Studium, Band 321,
expert verlag, 71272 Renningen-Malmsheim,
3., völlig neubearbeitete und erweiterte Auflage 1997,
ISBN 3-8169-1262-1

[9] Karl Heinz Müller,
”Elektronische Schaltungen und Systeme”,
simulieren, analysieren, optimieren mit SPICE,
VOGEL Buchverlag Würzburg, 1. Auflage,
ISBN 3-8023-0292-3

[10] Lawrence G. Meares, Charles E. Hymowitz,
”Simulating with SPICE”,
intusoft, San Pedro, CA, 1988,
web: `www.intusoft.com`

[11] Lawrence G. Meares, Charles E. Hymowitz,
”SPICE Applications Handbook”, 2nd Edition,
intusoft, San Pedro, CA, 1994,
ISBN 0-923345-03-5,
web: `www.intusoft.com`

[12] Karl Heinz Müller,
”A SPICE Cookbook”,
Edited by Charles E. Hymowitz and Jeff Robson,
intusoft, San Pedro, CA, 1991,
ISBN 0-923345-02-7,
web: `www.intusoft.com`

[13] Dirk Jansen (Hrsg.),
”Handbuch der Electronic Design Automation”,
Carl Hanser Verlag München Wien, 2001,
ISBN 3-446-21288-4
web: `www.hanser.de`

[14] Andrei Vladimirescu,
”THE SPICE BOOK”,
John Wiley and Sons, Inc., New York, 1994,
ISBN 0-471-60926-9

[15] Donald O. Pederson (Univ. of California, Berkeley),
Kartikeya Mayaram (Texas Instruments),
”Analog Integrated Circuits for Communication”,
Principles, Simulation and Design, Second Printing, 1991,
Kluwer Academic Publishers Group, Distribution Centre, P.O.Box 322,
The Netherlands,
ISBN 0-7923-9089-X