

The book was found

Semiconductor Device Modeling With Spice



Synopsis

SPICE (Software Program with Integrated Circuit Emphasis) is a powerful design aid that electronics engineers learn and is the world standard for circuit simulation. And when circuit designers are using the various versions of SPICE to simulate circuits prior to fabrication and accurately predict future performance, this guide could be a useful reference. This revised version explains the ins and outs of SPICE, plus gives new data on modelling advanced devices such as MESFETs, ISFETs, and thyristors. The book should help readers gain maximum value from SPICE, also covering both MOS and FET models. Step by step, it takes the reader through the modelling process, providing complete information on a variety of semiconductor devices for designing specific circuit applications. These include: Pn junction and Schottky diodes; bipolar junction transistor (BJT); junction field effect transistor (JFET); metal oxide semiconductor field effect transistor (MESFET); ion sensitive field effect transistor (ISFET); and semiconductor controlled rectifier (SCR-thyristor). Given the immense and growing acceptance of SPICE, this revision should appeal to the audience of engineers, technicians, and students who use this design program.

Book Information

Hardcover: 479 pages

Publisher: McGraw-Hill Professional Publishing; 2 Sub edition (April 1, 1993)

Language: English

ISBN-10: 0070024693

ISBN-13: 978-0070024694

Product Dimensions: 9.3 x 6.3 x 1.3 inches

Shipping Weight: 1.9 pounds (View shipping rates and policies)

Average Customer Review: 3.4 out of 5 stars 2 customer reviews

Best Sellers Rank: #1,753,427 in Books (See Top 100 in Books) #48 in Books > Engineering & Transportation > Engineering > Electrical & Electronics > Electronics > Solid State #283 in Books > Engineering & Transportation > Engineering > Electrical & Electronics > Electronics >

Semiconductors #319 in Books > Computers & Technology > Programming > Software Design, Testing & Engineering > Logic

Customer Reviews

Eh, it's a decent-enough book. The author is obviously well-versed. But, there's a lot of background theorems and proofs that make this feel more like a math-text than a "tweak this to get this" solution for SPICE modeling. I thought Chap7 (noise) plus Appendix-A and Appendix-B were solid, the rest

would be useful if you plan to design your own SPICE-sim, not use something already out there.

Spice is a software package that become a de facto standar in circuit simulation. But as in any other circuit simulator we need to define the accurate input model to get the accurate results. And this book describe in more detail the components that is used in almost any electronic circuit, i.e. semiconductor devices. The value that is a credit point for this book is not just how detail the description of the semiconductor model is in order to get the accurate parameters but also how to use those parameters in Spice. So, for you who new to this circuit simulator there is no need to worry because this book explain how the circuit model is built using Spice. And for you who need the basic theory of semiconductor model the book gives two nice appendices on that one. One chapter that is need to be explored in more detail is about noise and distortion (hopefully in the next edition the editor could have more on that topic). Finally, this is a "must be read" book for you who want to get excellent results in modelling electronic circuit. (For you who understand more clearly in Indonesian) Spice adalah sebuah paket aplikasi yang secara defacto merupakan standar untuk simulasi dan analisa rangkaian elektronik. Namun secara umum berlaku pula bahwa untuk menghasilkan simulasi rangkaian elektronik secara akurat dibutuhkan parameter-parameter yang akurat pula untuk setiap modelnya. Buku ini memberikan ulasan mendalam tentang parameter-parameter bagi pemodelan device semikonduktor. Kelebihan yang dimiliki buku ini tidak hanya pada bagaimana penghitungan dan pengukuran parameter untuk model tersebut tetapi bagaimana menggunakannya pada simulasi rangkaian menggunakan SPICE. Bagi Anda yang baru mendalami spice jangan khawatir karena buku ini juga memberikan ulasan tentang sistem pemodelan dalam spice. Sebaliknya bagi anda yang baru dalam pemodelan semikonduktor buku ini juga memberikan apendiks tentang teori dasar semikonduktor. Satu hal yang mungkin perlu dibahas lebih mendalam adalah tentang Noise dan Distorsi (mudah-mudahan dalam edisi berikutnya materi ini dapat lebih dikembangkan). Kesimpulannya, buku ini merupakan suatu buku yang wajib dibaca bila anda ingin lebih akurat dalam menjalankan simulasi rangkaian elektronik yang menggunakan semikonduktor.

[Download to continue reading...](#)

Semiconductor Device Modeling with Spice Introduction to Semiconductor Device Yield Modeling (Artech House Materials Science Library) Semiconductor Physics and Applications (Series on Semiconductor Science and Technology) Semiconductor Device Fundamentals Semiconductor Material and Device Characterization The Monte Carlo Method for Semiconductor Device Simulation (Computational Microelectronics) How to Add a Device to Account: How to add a device

to my account - 3 easy steps in few minutes Spice It Up: Spice Up Your Sex Life, Explore Your Fantasies and Kinks, and Blow Your Partner's Mind The Spice Merchant's Daughter: Recipes and Simple Spice Blends for the American Kitchen Spice Mix Recipes: Top 50 Most Delicious Spice Mix Recipes [A Seasoning Cookbook] (Recipe Top 50's Book 104) Spice Mix Recipes: Top 50 Most Delicious Dry Spice Mixes [A Seasoning Cookbook] Computational Electronics: Semiclassical and Quantum Device Modeling and Simulation High-Speed Heterostructure Devices: From Device Concepts to Circuit Modeling The Model's Bible & Global Modeling Agency Contact List - An Insider's Guide on How to Break into the Fashion Modeling Industry Modeling Agency Tips: Get Listed with Fashion Modeling Agencies and Find Your Dream Job 3ds Max Modeling for Games: Insider's Guide to Game Character, Vehicle, and Environment Modeling: Volume I Atmospheric and Space Flight Dynamics: Modeling and Simulation with MATLAB® and Simulink® (Modeling and Simulation in Science, Engineering and Technology) Introduction to the Numerical Modeling of Groundwater and Geothermal Systems: Fundamentals of Mass, Energy and Solute Transport in Poroelastic Rocks (Multiphysics Modeling) Modeling Dynamic Biological Systems (Modeling Dynamic Systems) Dynamic Modeling in the Health Sciences (Modeling Dynamic Systems)

[Contact Us](#)

[DMCA](#)

[Privacy](#)

[FAQ & Help](#)