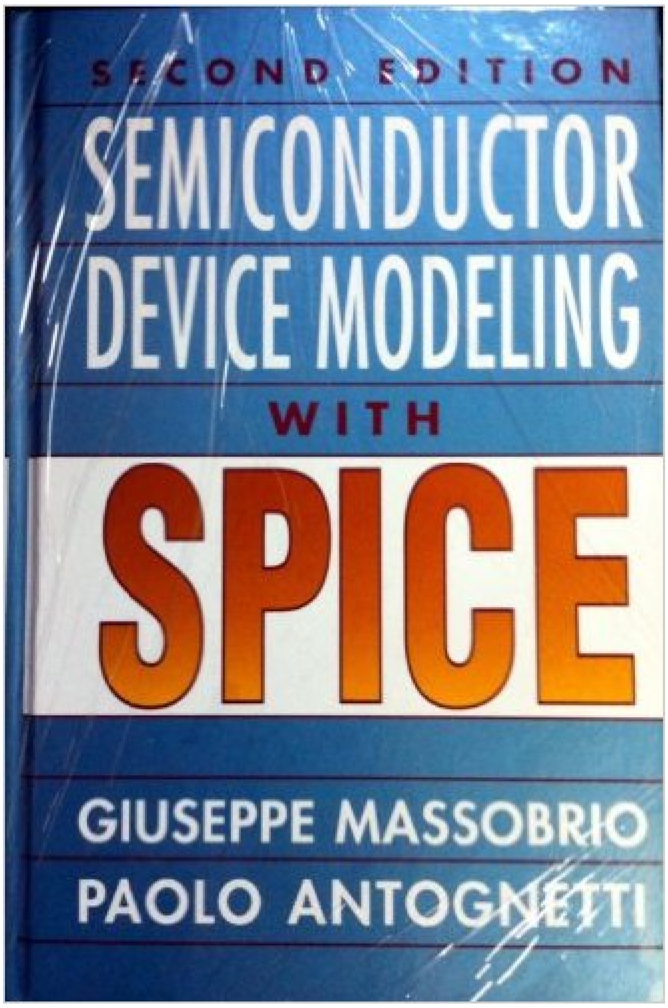
Semiconductor Device Modeling With Spice



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

Average Customer Review: 3.5 out of 5 starsÂ Â See all reviewsÂ (2 customer reviews)

Best Sellers Rank: #1,143,854 in Books (See Top 100 in Books) #24 inÂ Books > Engineering & Transportation > Engineering > Electrical & Electronics > Electronics > Solid State #196 inÂ Books > Engineering & Transportation > Engineering > Electrical & Electronics > Electronics > Semiconductors #198 inÂ Books > Computers & Technology > Programming > Software Design, Testing & Engineering > Logic

Customer Reviews

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 simulai 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.

Modeling and Differential Equations and Boundary Value Problems: Computing and Modeling Fault-Tolerance and Reliability Techniques for High-Density Random-Access Memories (Prentice Hall Modern Semiconductor Design Series) Understanding Semiconductor Devices (The Oxford Series in Electrical and Computer Engineering) Microchip Fabrication, Sixth Edition: A Practical Guide to Semiconductor Processing