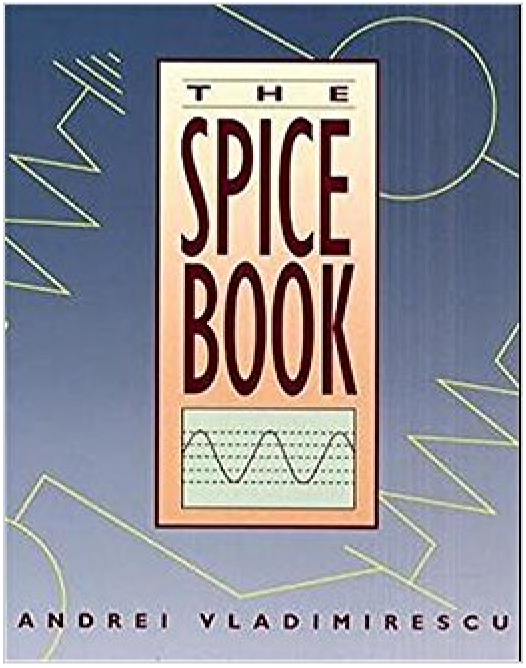
The book was found

The SPICE Book



Synopsis

Extremely easy-to-follow due to its natural progression tutorial approach on how to advance from the solution of typical electrical and electronic circuit examples by hand, followed by a SPICE verification through the discussion of simulation results. The first part contains relevant data about SPICE in order to analyze both linear passive and electronic circuits. The latter half provides more detail on such topics as distortion models and analysis, basic algorithms in SPICE, analysis option parameters and how to direct SPICE to find a solution when it fails.

Book Information

Paperback: 432 pages

Publisher: Wiley; 1 edition (January 6, 1994)

Language: English

ISBN-10: 0471609269

ISBN-13: 978-0471609261

Product Dimensions: 7.5 x 0.8 x 9.8 inches

Shipping Weight: 1.8 pounds (View shipping rates and policies)

Average Customer Review: 3.2 out of 5 stars   7 customer reviews

Best Sellers Rank: #532,060 in Books (See Top 100 in Books)   #164 inÂ Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Design   #423 inÂ Books > Computers & Technology > Graphics & Design > CAD   #485 inÂ Books > Computers & Technology > Databases & Big Data > Data Processing

Customer Reviews

Extremely easy-to-follow due to its natural progression tutorial approach on how to advance from the solution of typical electrical and electronic circuit examples by hand, followed by a SPICE verification through the discussion of simulation results. The first part contains relevant data about SPICE in order to analyze both linear passive and electronic circuits. The latter half provides more detail on such topics as distortion models and analysis, basic algorithms in SPICE, analysis option parameters and how to direct SPICE to find a solution when it fails.

This new book, written by Andre 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 SPICE2 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 feasible to find the solution by hand but the approach stresses the need for the SPICE user tounderstand 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 SPICE2, SPICE3 or PSPICE. Accurate descriptions, simulation rationale and cogent explanations make this an invaluable reference.

I bought the book to give to a new engineer to get him started in Spice modeling, but it isn’t really about that. It’s more about developing spice models for new components than it is about using Spice. It’s for really senior-level people wanting to take Spice to the next level than it is for a newbie who just downloaded LTSpice or Tina and who’s trying to figure out how to make it work.

This gives a good and detailed coverage of the Spice language and applications, but I was looking for LTspice IV and XVII guidance and this predates them - my mistake.

This book will take you from using SPICE to using SPICE with a decent knowledge of what’s under-the-hood. It’s well organized, uses a good consistent vocabulary, provides many examples and code snippets, does a good job decomposing some of the constructs (model innards, etc.) that you might use and wish you knew more about. I hope this helps somebody checking out this book.

The table of contents is viewable so check that out. There’s also a section on converge which is really helpful.

I was expecting a book that presented in-depth simulation techniques using SPICE, but I got a book that is pretty much the LTSpice help manual.

This book answers many of the questions I couldn’t find answers to in help files or other books. It also contains much which is found in help files, but I find having it all in one book is easier than wading through the multiple .pdf’s that help files have become.

This book has just about everthing you want to know about spice from it’s history in early 60’s, to solving convergance in array of circuits,which are difficult to simulate.I found the book to be useful for it’s tables to construct better and more models than I had ever imagined possible.The author does use alot of mathematical expressions to explain the spice syntax, but the format is great, and well thought out.This book is the "toolbox" for spice. Others fall short, but this book I’m gald to have on my shelf.

This book incloses most of the mathematical equations of the models used by SPICE and helps good by solving problems with the convergence, written by one of the designer of SPICE A. Vladimirescu of Spice Vegan Fire & Spice: 200 Sultry and Savory Global Recipes An Aphrodisiac Cookbook: What to cook to charm for one evening. Complete Guide, Tips & Tricks, Essential TOP recipes to Spice Up Your Sex Life ... recipes, easy recipes, cookbooks) Nathaniel's Nutmeg: Or the True and Incredible Adventures of the Spice Trader Who Changed the Course of History New Drugs: Bath Salts, Spice, Salvia, & Designer Drugs The Taste of Conquest: The Rise and Fall of the Three Great Cities of Spice Spice: The History of a Temptation