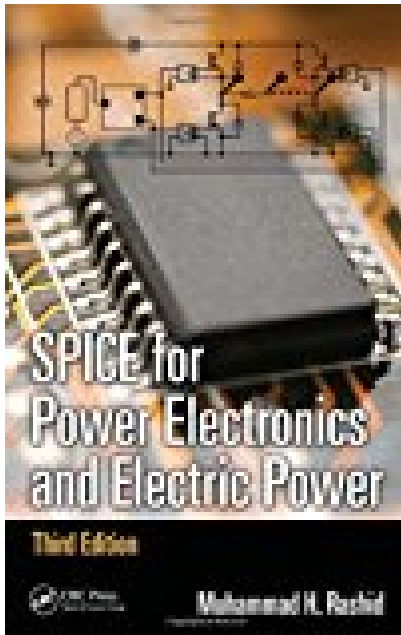


SPICE for Power Electronics and Electric Power Third Edition Electrical and Computer Engineering



BOOK DETAILS

- Author : Muhammad H. Rashid
- Pages : 559 Pages
- Publisher : CRC Press
- Language : English
- ISBN : 1439860467



BOOK SYNOPSIS

Power electronics can be a difficult course for students to understand and for professors to teach. Simplifying the process for both, SPICE for Power Electronics and Electric Power, Third Edition illustrates methods of integrating industry standard SPICE software for design verification and as a theoretical laboratory bench. Helpful PSpice Software and Program Files Available for Download Based on the author Muhammad H. Rashid's considerable experience merging design content and SPICE into a power electronics course, this vastly improved and updated edition focuses on helping readers integrate the SPICE simulator with a minimum amount of time and effort. Giving users a better understanding of the operation of a power electronics circuit, the author explores the transient behavior of current and voltage waveforms for each and every circuit element at every stage. The book also includes examples of all types of power converters, as well as circuits with linear and nonlinear inductors. New in this edition: Student learning outcomes (SLOs) listed at the start of each chapter Changes to run on OrCAD version 9.2 Added VPRINT1 and IPRINT1 commands and examples Notes that identify important concepts Examples illustrating EVALUE, GVALUE, ETABLE, GTABLE, ELAPLACE, GLAPLACE, EFREQ, and GFREQ Mathematical relations for expected outcomes, where appropriate The Fourier series of the output voltages for rectifiers and inverters PSpice simulations of DC link inverters and AC voltage controllers with PWM control This book demonstrates techniques of executing power conversions and ensuring the quality of the output waveforms rather than the accurate modeling of power semiconductor devices. This approach benefits students, enabling them to compare classroom results obtained with simple switch models of devices. In addition, a new chapter covers multi-level converters. Assuming no prior knowledge of SPICE or PSpice simulation, the text provides detailed step-by-step instructions on how to draw a schematic of a circuit, execute simulations, and view or plot the output results. It also includes suggestions for laboratory experiments and design problems that can be used for student homework assignments.

SPICE FOR POWER ELECTRONICS AND ELECTRIC POWER THIRD EDITION ELECTRICAL AND COMPUTER ENGINEERING

- Are you looking for Ebook SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering ? You will be glad to know that right now SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering is available on our online library. With our online resources, you can find Applied Numerical Methods With Matlab Solution Manual 3rd Edition or just about any type of ebooks, for any type of product.

Best of all, they are entirely free to find, use and download, so there is no cost or stress at all. SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering may not make exciting reading, but Applied Numerical Methods With Matlab Solution Manual 3rd Edition is packed with valuable instructions, information and warnings. We also have many ebooks and user guide is also related with SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering and many other ebooks. We have made it easy for you to find a PDF Ebooks without any digging. And by having access to our ebooks online or by storing it on your computer, you have convenient answers with SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering . To get started finding SPICE For Power Electronics And Electric Power Third Edition Electrical And Computer Engineering , you are right to find our website which has a comprehensive collection of manuals listed.