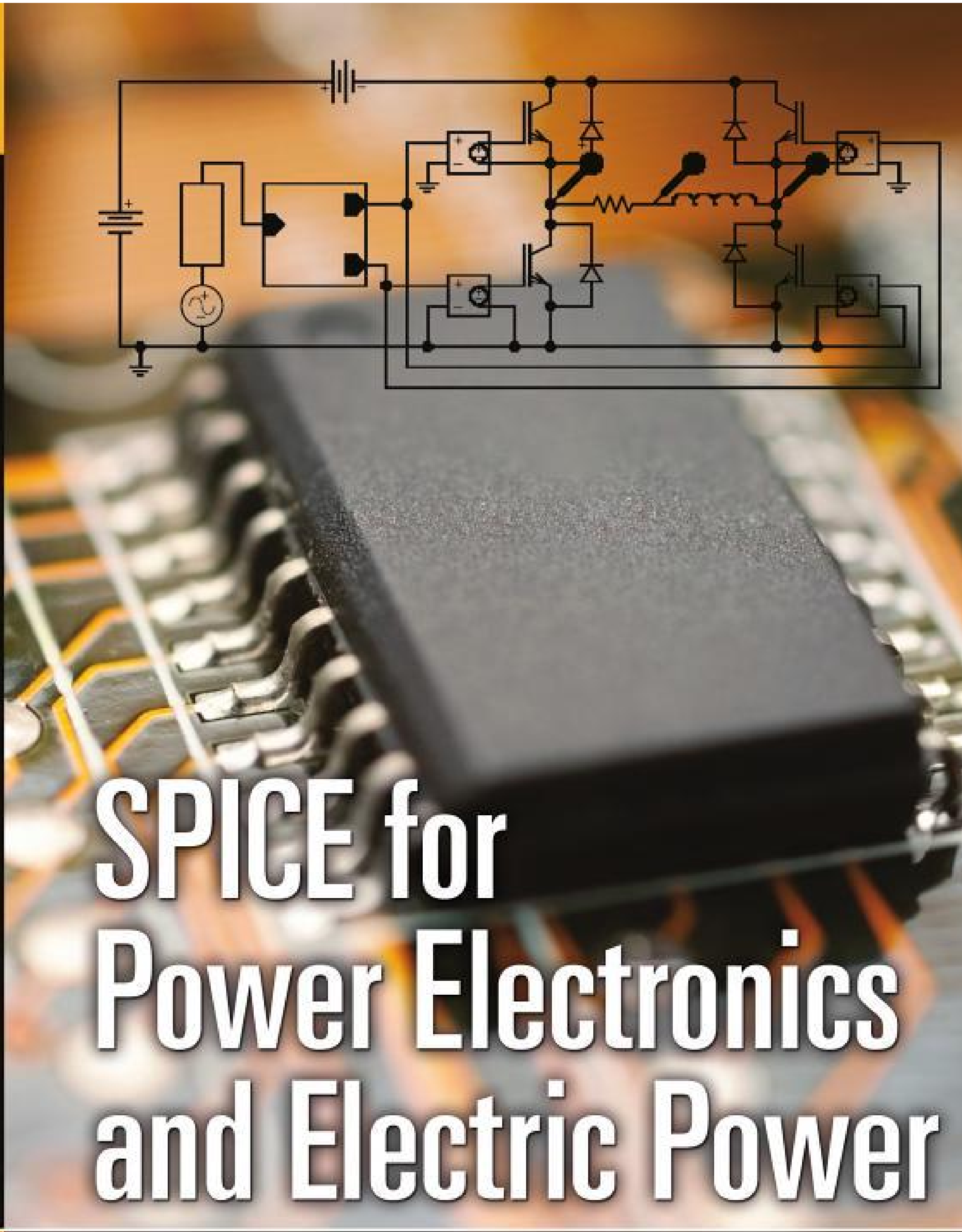

SPICE for Power Electronics and Electric Power

Third Edition

CRC Press

Muhammad H. Rashid

SPICE for Power Electronics and Electric Power, Third Edition, Muhammad H. Rashid, CRC Press, 2012, 1439860467, 9781439860465, 559 pages. 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. RashidÐ²Ð‚â„¢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..

DOWNLOAD HERE

PSPICE and MATLAB for Electronics An Integrated Approach, John Okyere Attia, May 15, 2002, Technology & Engineering, 360 pages. PSPICE has circuit simulation features unmatched by any other scientific software. MATLAB's capabilities for matrix computations, plotting, data processing, and analysis are ....

Fundamentals of Power Electronics , Robert W. Erickson, Dragan Maksimovic, Jan 31, 2001, Technology & Engineering, 883 pages. Fundamentals of Power Electronics, Second Edition, is an up-to-date and authoritative text and reference book on power electronics. This new edition retains the original ....

Power Electronics Circuits, Devices, and Applications, Muhammad Harunur Rashid, 2004, Power electronics, 880 pages. .

SPICE for circuits and electronics using PSpice , M. H. Rashid, 1995, Technology & Engineering, 364 pages. Computer-aided analysis and design is fast becoming a required skill for today's electronic engineers/technicians. SPICE Ð²Ð‚â€• a very popular software for analyzing electrical and ....

Fundamentals of Power Electronics , Muhammad H. Rashid, 1996, Technology & Engineering, 203 pages. This comprehensive introduction to power semiconductor devices, their characteristics, and their ratings will take you step-by-step through the most important topics in the ....

PSpice Simulation of Power Electronics Circuits An Introductory Guide, E. Ramshaw, D.C. Schuurman, Dec 31, 1996, Technology & Engineering, 404 pages. This book is aimed at advanced students and practising engineers. It provides step by step instructions in the use of MicroSim PSpice, industry-standard software that simulates ....

Power-Switching Converters, Second Edition , Simon Ang, Alejandro Oliva, Mar 17, 2005, Technology & Engineering, 568 pages. After nearly a decade of success owing to its thorough coverage, abundance of problems and examples, and practical use of simulation and design, Power-Switching Converters ....

Spice A Guide to Circuit Simulation and Analysis Using Pspice, Paul W. Tuinenga, 1995, Computers, 288 pages. Revision of best-selling guide to the PSpice circuit simulator by an authoritative author. Provides a "tutorial approach" to using PSpice through graduated examples ....

Introduction to PSpice using OrCAD for circuits and electronics , M. H. Rashid, 2004, Computers, 456 pages. For second and third year Electrical Engineering courses in Electronics, Circuit Analysis, and Circuit Simulation. Implementing the industry-standard software, this book can be ....

PSpice and circuit analysis , John L. Keown, 1991, Technology & Engineering, 333 pages. .

Microelectric Circuits: Analysis and Design Analysis and Design, Muhammad H. Rashid, Apr 1, 2010, Technology & Engineering, 1100 pages. The objectives are the book are to provide an understanding of the characteristics of semiconductor devices and commonly used integrated circuits; to develop skills in analysis ....

http://kgarch.org/1163.pdf
http://kgarch.org/9cf.pdf
http://kgarch.org/16a7.pdf
http://kgarch.org/mn2.pdf
http://kgarch.org/bjn.pdf
http://kgarch.org/fh2.pdf
http://kgarch.org/6b.pdf
http://kgarch.org/l3m.pdf
http://kgarch.org/28e.pdf
http://kgarch.org/c3l.pdf
http://kgarch.org/12m2.pdf