



2009, XVI, 399 p. 158 illus.

A product of Birkhäuser Basel

Printed book

Hardcover

- ▶ 149,99 € | £129.99 | \$179.99
- ▶ *160,49 € (D) | 164,99 € (A) | CHF 177.00

eBook

Available from your library or

- ▶ springer.com/shop

MyCopy

Printed eBook for just

- ▶ € | \$ 24.99
- ▶ springer.com/mycopy

T. Tuma, Á. Bürrmen

Circuit Simulation with SPICE OPUS

Theory and Practice

Series: Modeling and Simulation in Science, Engineering and Technology

- ▶ This is the first complete guide to analog circuit design using SPICE OPUS
- ▶ Written by the creators of the SPICE OPUS software package
- ▶ Is a unique combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual
- ▶ SPICE OPUS software and source files for the simulation examples presented in the book are freely available on the SPICE OPUS webpage: www.spiceopus.si
- ▶ May be used by novices as well as professional circuit designers
- ▶ May be used as a textbook for a course on circuit simulation and also as a self-study reference for students and researchers

This book is the first complete guide to analog circuit design using the circuit simulator software package SPICE OPUS. Developed by the authors and used by academics and industry professionals worldwide, SPICE OPUS is an improved version of the well-known University of California at Berkeley circuit simulator SPICE3 that has been freely available online since 2000.

Aimed at novices as well as professional circuit designers, the book is a unique combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual. All simulations as well as the free simulator software may be directly downloaded from the SPICE OPUS homepage: www.spiceopus.si.

The book is divided into three parts:

- * **Theory** (Chapters 1 and 6): Includes a discussion of basic mathematical notions of circuit analysis, followed by specific algorithms implemented in SPICE OPUS.
- * **Crash course** (Chapters 2 and 7): Begins with a short installation guide and then moves quickly through a typical circuit simulation scenario, based on a simple example. The reader with some fundamentals in electrical engineering may continue with a number of complete simulation sessions presented in Chapter 7.
- * **Reference guide** (Chapters 3, 4, and 5): Describes all features of SPICE OPUS in a well-structured, methodical way, starting with input file syntax, followed by circuit analysis methods and the built-in scripting language (NUTMEG).

Order online at springer.com ▶ or for the Americas call (toll free) 1-800-SPRINGER ▶ or email us at: customerservice@springer.com. ▶ For outside the Americas call +49 (0) 6221-345-4301 ▶ or email us at: customerservice@springer.com.

The first € price and the £ and \$ price are net prices, subject to local VAT. Prices indicated with * include VAT for books; the €(D) includes 7% for Germany, the €(A) includes 10% for Austria. Prices indicated with ** include VAT for electronic products; 19% for Germany, 20% for Austria. All prices exclusive of carriage charges. Prices and other details are subject to change without notice. All errors and omissions excepted.

