



Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology)

By Tadej Tuma, Árpád Buermen

Download now

Read Online ➔

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen

This book is a unique combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual, which may be used as a textbook or self-study reference. The book is divided into three parts: mathematical theory of circuit analysis, a crash course on SPICE OPUS, and a complete SPICE OPUS reference guide. All simulations as well as the free simulator software may be directly downloaded from the SPICE OPUS homepage: www.spiceopus.si.

Circuit Simulation with SPICE OPUS is intended for a wide audience of undergraduate and graduate students, researchers, and practitioners in electrical and systems engineering, circuit design, and simulation development.

↓ [Download Circuit Simulation with SPICE OPUS: Theory and Pra ...pdf](#)

📖 [Read Online Circuit Simulation with SPICE OPUS: Theory and P ...pdf](#)

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology)

By Tadej Tuma, Árpád Buermen

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen

This book is a unique combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual, which may be used as a textbook or self-study reference. The book is divided into three parts: mathematical theory of circuit analysis, a crash course on SPICE OPUS, and a complete SPICE OPUS reference guide. All simulations as well as the free simulator software may be directly downloaded from the SPICE OPUS homepage: www.spiceopus.si.

Circuit Simulation with SPICE OPUS is intended for a wide audience of undergraduate and graduate students, researchers, and practitioners in electrical and systems engineering, circuit design, and simulation development.

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen Bibliography

- Sales Rank: #2416841 in eBooks
- Published on: 2009-06-23
- Released on: 2009-06-23
- Format: Kindle eBook

 [Download Circuit Simulation with SPICE OPUS: Theory and Pra ...pdf](#)

 [Read Online Circuit Simulation with SPICE OPUS: Theory and P...pdf](#)

Editorial Review

Review

From the reviews:

“The book deals with circuit simulation with SPICE in theory and practice. ... The book enjoys an excellent graphics (formulae, various representations of results). The references give only the cornerstone contributions in the area (55 entries). The examples cover a large area of applications: modeling a nonlinear transistor; logic gates; phase-locked loops; etc. Finally, we conclude that the book represents an excellent ‘instrument de travail’ for all those working in the area of circuits design and simulation.” (Dumitru Stanomir, Zentralblatt MATH, Vol. 1219, 2011)

“This book is a combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual, which may be used as a textbook or self-study reference. ... Circuit simulation with SPICE OPUS is a book intended to a wide audience ranging from undergraduate students to IC designers, researchers and simulator developers. ... This book can be also useful to other commercial SPICE users for additional insight into SPICE internals and may help explain certain issues with other version of SPICE.” (Danut Burdia, IASI Polytechnic Magazine, Vol. 22 (1/4), March-December, 2010)

From the Back Cover

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).

Circuit Simulation with SPICE OPUS is intended for a wide audience of undergraduate and graduate students, researchers, and practitioners in electrical and systems engineering, circuit design, and simulation

development. The book may be used as a textbook for an advanced undergraduate or graduate course on circuit simulation as well as a self-study reference guide for students and researchers alike.

Users Review

From reader reviews:

Mable Garza:

Often the book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) will bring you to definitely the new experience of reading a book. The author style to describe the idea is very unique. When you try to find new book to study, this book very suited to you. The book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) is much recommended to you to learn. You can also get the e-book from your official web site, so you can more easily to read the book.

Margaret Clayton:

Exactly why? Because this Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) is an unordinary book that the inside of the publication waiting for you to snap the idea but latter it will jolt you with the secret this inside. Reading this book beside it was fantastic author who also write the book in such awesome way makes the content within easier to understand, entertaining technique but still convey the meaning entirely. So , it is good for you because of not hesitating having this any longer or you going to regret it. This excellent book will give you a lot of advantages than the other book have such as help improving your skill and your critical thinking means. So , still want to hesitate having that book? If I had been you I will go to the guide store hurriedly.

Terry Kline:

This Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) is completely new way for you who has attention to look for some information because it relief your hunger associated with. Getting deeper you into it getting knowledge more you know or you who still having little bit of digest in reading this Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) can be the light food to suit your needs because the information inside that book is easy to get by means of anyone. These books build itself in the form and that is reachable by anyone, sure I mean in the e-book form. People who think that in publication form make them feel tired even dizzy this guide is the answer. So you cannot find any in reading a reserve especially this one. You can find what you are looking for. It should be here for anyone. So , don't miss this! Just read this e-book variety for your better life along with knowledge.

Tommy Bowles:

Don't be worry for anyone who is afraid that this book will probably filled the space in your house, you might have it in e-book approach, more simple and reachable. This particular Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) can give

you a lot of close friends because by you taking a look at this one book you have matter that they don't and make anyone more like an interesting person. That book can be one of one step for you to get success. This publication offer you information that maybe your friend doesn't learn, by knowing more than other make you to be great individuals. So , why hesitate? Let's have Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology).

**Download and Read Online Circuit Simulation with SPICE OPUS:
Theory and Practice (Modeling and Simulation in Science,
Engineering and Technology) By Tadej Tuma, Árpád Buermen
#IJPR4Z7EQ0Y**

Read Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen for online ebook

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen Free PDF d0wnl0ad, audio books, books to read, good books to read, cheap books, good books, online books, books online, book reviews epub, read books online, books to read online, online library, greatbooks to read, PDF best books to read, top books to read Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen books to read online.

Online Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen ebook PDF download

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen Doc

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen Mobipocket

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen EPub

IJPR4Z7EQ0Y: Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen