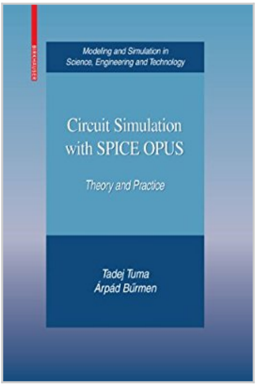

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology)

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

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

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:

Margert Lewis:

What do you think about book? It is just for students as they are still students or it for all people in the world, the particular best subject for that? Just you can be answered for that query above. Every person has distinct personality and hobby for every other. Don't to be forced someone or something that they don't want do that. You must know how great and important the book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology). All type of book is it possible to see on many solutions. You can look for the internet solutions or other social media.

Robin Millard:

Now a day those who Living in the era just where everything reachable by talk with the internet and the resources inside it can be true or not demand people to be aware of each facts they get. How people have to be smart in getting any information nowadays? Of course the answer is reading a book. Examining a book can help persons out of this uncertainty Information especially this Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) book because this book offers you rich data and knowledge. Of course the data in this book hundred per cent guarantees there is no doubt in it you probably know this.

Danny Chamberland:

As we know that book is essential thing to add our knowledge for everything. By a publication we can know everything we would like. A book is a set of written, printed, illustrated or maybe blank sheet. Every year ended up being exactly added. This reserve Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) was filled concerning science. Spend your time to add your knowledge about your research competence. Some people has diverse feel when they reading a book. If you know how big advantage of a book, you can sense enjoy to read a publication. In the modern era like now, many ways to get book that you wanted.

Charles Wright:

As a college student exactly feel bored to be able to reading. If their teacher questioned them to go to the library or make summary for some e-book, they are complained. Just minor students that has reading's heart and soul or real their passion. They just do what the instructor want, like asked to the library. They go to presently there but nothing reading significantly. Any students feel that reading through is not important, boring and can't see colorful pics on there. Yeah, it is to be complicated. Book is very important for yourself. As we know that on this era, many ways to get whatever we would like. Likewise word says, ways to reach Chinese's country. So , this Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and

Simulation in Science, Engineering and Technology) can make you truly feel more interested to read.

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 #JFW6Y53OH4M

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