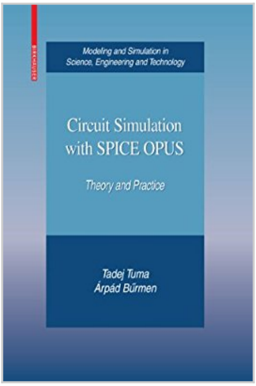

Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology)

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

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

---

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:

Joe Vizcarra:

Playing with family inside a park, coming to see the sea world or hanging out with friends is thing that usually you could have done when you have spare time, then why you don't try matter that really opposite from that. A single activity that make you not feeling tired but still relaxing, trilling like on roller coaster you already been ride on and with addition of knowledge. Even you love Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology), you may enjoy both. It is very good combination right, you still would like to miss it? What kind of hang type is it? Oh come on its mind hangout fellas. What? Still don't understand it, oh come on its known as reading friends.

Laura Burnham:

Your reading 6th sense will not betray you actually, why because this Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) guide written by well-known writer who knows well how to make book that can be understand by anyone who also read the book. Written within good manner for you, still dripping wet every ideas and creating skill only for eliminate your personal hunger then you still question Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) as good book not merely by the cover but also by content. This is one reserve that can break don't assess book by its include, so do you still needing one more sixth sense to pick this!? Oh come on your looking at sixth sense already told you so why you have to listening to one more sixth sense.

Cesar Benedetto:

Beside this particular Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) in your phone, it may give you a way to get closer to the new knowledge or facts. The information and the knowledge you will got here is fresh through the oven so don't be worry if you feel like an old people live in narrow commune. It is good thing to have Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) because this book offers for your requirements readable information. Do you oftentimes have book but you would not get what it's all about. Oh come on, that would not happen if you have this in the hand. The Enjoyable agreement here cannot be questionable, such as treasuring beautiful island. So do you still want to miss the item? Find this book along with read it from right now!

Kirk Nutter:

Is it you who having spare time and then spend it whole day through watching television programs or just resting on the bed? Do you need something totally new? This Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) can be the respond to, oh how comes? The new book you know. You are so out of date, spending your free time by reading in this brand-new era is common not a geek activity. So what these books have than the others?

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 #8FE04B5KDCR

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

8FE04B5KDCR: Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen