



## 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](http://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](http://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: #737792 in Books
- Published on: 2009-07-01
- Original language: English
- Number of items: 1
- Dimensions: 9.21" h x .94" w x 6.14" l, 1.67 pounds
- Binding: Hardcover
- 399 pages

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

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

## 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](http://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:

#### Gerald James:

In this 21st one hundred year, people become competitive in each and every way. By being competitive today, people have to do something to make them survive, being in the middle of the particular crowded place and notice by surrounding. One thing that oftentimes many people have underestimated it for a while is reading. Yep, by reading a reserve your ability to survive raise then having chance to stay than other is high. For you personally who want to start reading any book, we give you this specific Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) book as beginner and daily reading book. Why, because this book is usually more than just a book.

#### Ila Robinette:

Hey guys, do you would like to find a new book you just read? May be the book with the name Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) suitable to you? The actual book was written by popular writer in this era. Typically the book untitled Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) is the one of several books in which everyone read now. This book was inspired many men and women in the world. When you read this reserve you will enter the new age that you ever know just before. The author explained their idea in the simple way, therefore all of people can easily to recognise the core of this reserve. This book will give you a great deal of information about this world now. So that you can see the represented of the world within this book.

#### Elizabeth Bello:

Do you have something that you enjoy such as book? The publication lovers usually prefer to choose book like comic, quick story and the biggest you are novel. Now, why not seeking Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) that give your enjoyment preference will be satisfied by means of reading this book. Reading habit all over the world can be said as the opportunity for people to know world considerably better than how they react to the world. It can't be stated constantly that reading behavior only for the geeky man but for all of you who wants to always be success person. So, for all you who want to start examining as your good habit, it is possible to pick Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) become your own personal starter.

#### Herbert Knight:

Can you one of the book lovers? If yes, do you ever feeling doubt when you are in the book store? Attempt to pick one book that you find out the inside because don't judge book by its protect may doesn't work at this

point is difficult job because you are frightened that the inside maybe not since fantastic as in the outside seem likes. Maybe you answer might be Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) why because the great cover that make you consider with regards to the content will not disappoint an individual. The inside or content is fantastic as the outside or maybe cover. Your reading sixth sense will directly direct you to pick up this book.

**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  
#R6PBMAOZY13**

## **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**