



# 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](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](#)

## **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:**

#### **Mark Blanding:**

The book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) give you a sense of feeling enjoy for your spare time. You can utilize to make your capable far more increase. Book can being your best friend when you getting stress or having big problem together with your subject. If you can make reading a book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) to become your habit, you can get a lot more advantages, like add your own personal capable, increase your knowledge about many or all subjects. You are able to know everything if you like open up and read a e-book Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology). Kinds of book are several. It means that, science guide or encyclopedia or other people. So , how do you think about this book?

#### **Emma Lavigne:**

This Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) is great reserve for you because the content which can be full of information for you who also always deal with world and get to make decision every minute. This particular book reveal it data accurately using great coordinate word or we can point out no rambling sentences within it. So if you are read that hurriedly you can have whole info in it. Doesn't mean it only offers you straight forward sentences but difficult core information with beautiful delivering sentences. Having Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) in your hand like finding the world in your arm, data in it is not ridiculous one. We can say that no book that offer you world throughout ten or fifteen small right but this guide already do that. So , it is good reading book. Hello Mr. and Mrs. stressful do you still doubt which?

#### **Mildred Lucas:**

The book untitled Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) contain a lot of information on the item. The writer explains the girl idea with easy technique. The language is very easy to understand all the people, so do certainly not worry, you can easy to read this. The book was compiled by famous author. The author brings you in the new time of literary works. You can actually read this book because you can read on your smart phone, or gadget, so you can read the book throughout anywhere and anytime. If you want to buy the e-book, you can wide open their official web-site as well as order it. Have a nice examine.

**Willie Dominguez:**

Beside this specific Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) in your phone, it could give you a way to get closer to the new knowledge or data. The information and the knowledge you can get here is fresh through the oven so don't possibly be worry if you feel like an previous 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 to your account readable information. Do you oftentimes have book but you rarely get what it's about. Oh come on, that will not end up to happen if you have this in your hand. The Enjoyable arrangement here cannot be questionable, similar to treasuring beautiful island. Techniques you still want to miss that? Find this book along with read it from today!

**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  
#G638QVWMR1U**

# **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**

**G638QVWMR1U: Circuit Simulation with SPICE OPUS: Theory and Practice (Modeling and Simulation in Science, Engineering and Technology) By Tadej Tuma, Árpád Buermen**