

Book Will Help You Get More, Out of LTSPICE IV

The LTSPICE IV Simulator: Manual, Methods, and Applications, Gilles Brocard, Würth Elektronik, 2011, 744 pages, hardback, Print ISBN: 978-3-89929-258-9. www.we-online.com

Reviewed by Dennis Feucht, Innovatia Laboratories, Cayo, Belize

This book is in the genre of books that supplement product manuals, such as *Windows Annoyances* or *Engineering with MathCAD*, which supports and explains what the MathCAD manual did not. With the prevalence of circuit simulation in SPICE, Linear Technology Corp. (LTC) has offered engineers a free version of their own SPICE program. This book, written by an author from France in English and published in Germany about a program from Silicon Valley, is illustrated largely with computer display screens of LTSPICE IV. In his preface to the book, Mike Engelhardt, who manages simulation development at LTC, comments on his experience in writing this latest version of the program, saying it "let me implement a number of numerical methods that make LTspice better than traditional SPICE programs."

This book will appeal to both beginners in LTSPICE and experienced, serious users seeking explanations for those nuanced behaviors that writers of manuals never include—or do not even know about. The first section covers LTSPICE history, beginning with computerized circuit simulation generally, and working through the different eras of SPICE to arrive at LTSPICE. The next section goes through the inventory of what you get: a survey of the files that comprise LTSPICE IV. Chapter 3 is for beginners, how to bring it up on Windows, a walk-through of the menu items and what they do, and the various measurement functions. Chapter 4, also for beginners, is the editor for circuit (schematic) diagrams. The author emphasizes subtleties that lead the user around needless frustrations.

The next group of chapters explains LTSPICE, its syntax and the components editor (chapter 5), the symbol editor and hierarchy usage rules (chapter 6), netlist editor (chapter 7), and the measurement tools (chapter 8): the virtual oscilloscope and FFT editors.

Chapter 9 starts a new group of chapters on how to "drive" the simulator. It is about configuration directives. SPICE has several kinds of simulation modes and chapter 10 covers "The Six Main Simulations" (DC, AC, TRAN, etc.). Chapter 11 covers advanced features: how to download measurements, backup, and the creation and modification of subcircuit models. Chapter 12 is about how to import them. While this review is attempting to give you a sense of the book in words only, the book itself is loaded with illustrations. Many pages are almost all illustrations of circuits, SPICE code, or mainly screen shots.

The next group of chapters is component-intensive: voltage and current sources (chapter 13), passive components (14), semiconductors (15), transmission lines and other special functions (such as Schmitt trigger circuits, phase detectors, and samplers), and of interest to power-circuits modelers, magnetic components (17) with their various nuances such as hysteresis and saturation.

Chapter 18 gives advanced driving instructions about the control panel and keyboard shortcuts. Example simulations and component modeling (19) are followed by a Q&A chapter.

The last chapter—and power circuits engineers will like this—is a return to the nuances of inductors and transformers in the context of switching converter simulation. Discontinuous functions have been a numerical math problem in SPICE historically, and various power-electronics experts (such as Professor Richard Tymerski at Portland State U., a developer of the original PWM-switch model, and his Ph.D. student, Duwang "Don" Li, now of CUI) have written their own simulators with a more-advanced understanding of how to handle the switching behavior. This chapter does not offer superficial explanations but confronts some of the more-detailed problems involving power circuits. It even makes a plug for *WE-FLEX Transformer Designer* software.

The Appendix has tables of magnetic core properties. And Internet links are given, including "if you only remember one address where you can find information about LTspice IV, its applications, models, ..." Of course, a few webpages in French are included. I credit the author or publisher (or both) for a book well-written.

This book is a must-have for anyone using LTSPICE. It immensely expands coverage of its use.

About The Author



Dennis Feucht has been involved in power electronics for 25 years, designing motor-drives and power converters. He has an instrument background from Tektronix, where he designed test and measurement equipment and did research in Tek Labs. He has lately been doing current-loop converter modeling and converter optimization.