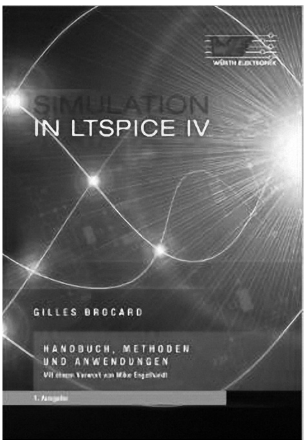
Book review – The LTSPICE IV Simulator

Bob Cordell

The LTSPICE IV Simulator – Manual, Methods and Applications; Gilles Brocard, 1st Edition. Preface by Mike Engelhardt.
Publisher: Wurth Electronik; ISBN 978-3-89929-258-9, Hardcover 2011, ISBN 10: 3899292588, ISBN 13: 978-3899292589, € 49, $ 49.99



In my view, LTspice is the best SPICE simulator out there. The fact that it is free makes it a treasure. For this reason I eagerly anticipated reading this heavy volume written by Gilles Brocard. I wish this book had been available when I was writing my power amplifier book, especially for the LTspice chapters. I have been using LTspice for about 10 years and still learned many options and details that I did not know.

Overall, this book is an excellent reference and is far better than the Help files included with LTspice. It is not an easy read, but the beginner and less experienced user should read the first several chapters and browse through all of the others. There are many good nuggets of information to be found. Even those highly experienced should take a browse through the book and then come back as needed. The depth and detail of coverage is a blessing and a curse. It makes for a difficult and very dry read, but the book is very complete. The enormity of options in LTspice makes it difficult to write a book that is an easy read or which covers all of the options in a readily accessible way. Even a casual read of this book will require a considerable investment in time.

The book goes into much more detail and depth than the average user will ever need, but this is what a reference is for. It illuminates things that can be done with LTspice that the reader may not have thought of. Many of these can be found with a modest skim of the book. If you are like me, you'll pay the most attention to the figures in a skim. However, one criticism I have is that most of the figures do not have captions.

As a reference, I would have preferred a much larger index. This shortcoming made it difficult to find things. For example, reference was made to PWL, I had forgotten what it meant, and it was not in the index. Coverage of LTspice is broad, but there is a greater amount of material pertaining to switching power supplies than to analog and audio circuit usage of the simulator. This is understandable since the author's area of expertise is in the field of switching power supplies.

The 21 Chapters

Chapter 1 is interesting but not really substantive; it is a history of SPICE and LTspice.

Chapter 2 introduces one to all of the files, file extensions and examples supplied with LTspice. Read this chapter, but don't get overwhelmed by it. You can skip it and look at it as a reference at a later time.

Chapter 3 is the most valuable in getting started, at least for a novice. It helps build familiarity and confidence quickly. Read this chapter and then read it again with LTspice open in front of you. It takes you through a simple simulation step-by-step. The chapter is generously filled with illustrations, many of which are LTspice screen shots that guide your reading. Recognizing that there is often one good-sized illustration per page, the thickness of the 700+ page book is much less daunting.
Chapter 3 is very helpful because it illustrates basic LTspice simulations without the clutter of the deep, and often unnecessary very detailed coverage of many options that are not often used. I would like to have seen this chapter even longer in covering more simple things, like noise and dc sweep, dc operating point, operating characteristics of transistors, etc. More coverage of probing and saving of plots would also have been helpful here. This is covered in later chapters, but in amongst the clutter of lots of other detail.

Chapter 4 covers the schematic editor in detail, and is worth reading or at least browsing for anyone who is not a true full-time LTspice guru (that includes me).

Chapter 5 describes the syntax and components editor, where the values and other attributes of components are chosen. Like many of the chapters, the subject matter is handled in depth, but perhaps only 10% of it is used 90% of the time by the average user. This chapter is more useful as a reference.

Chapter 6 describes the symbol editor. This comes into play when you create a new component for which there is not an existing symbol that can be used. In its simplest use, it may be used to create a box with pins for a sub-circuit that you have created. In addition to a thorough description with a generous number of illustrations, there is a very helpful example given at the end of the chapter.

Chapter 7 covers the netlist editor. It is used only infrequently because the schematic editor creates the netlist.

Chapter 8 discusses measurements, waveforms and the FFT. This chapter is especially important because it covers the use and interpretation of the results of the simulations. Read this chapter thoroughly and play with the commands with a simulation open. You'll see that most of what you need to do is easier and more straightforward than you might think based on reading the very detailed descriptions.

Chapter 9 covers simulation directives, the commands that tell the simulator what type of simulation to carry out and how sources are controlled. These are the dot commands that are placed on the schematic.

Chapter 10 describes the so-called six main simulations. In many ways this is the heart of the book for using LTspice. These six simulations are the simulations that have tabs at the top of the "Edit Simulation Command" window. They comprise DC operating point, AC analysis, Transient, DC sweep, DC transfer and Noise. Here is where the most important simulation directives are created for placement on the schematic. This chapter is a must-read. I strongly recommend experimenting with LTspice simulations while reading this chapter a second time. In some cases, it will take some time to get the hang of it.
This core chapter is about 50 pages long, and could have been double the length. In particular, I would like to have seen a minimum of 10 pages devoted to each of the six simulations, with at least one circuit example that is taken through all six of the simulations. As it is, only 8 pages are dedicated specifically to the 5 simulations of DC operating point, DC sweep, DC transfer, AC analysis and Noise. In contrast, 35 pages are dedicated to the Transient and FFT simulations, as well they should for these more complex tasks. Six pages are dedicated to a good treatment of Monte Carlo simulation. Of particular interest to audio applications is the material relating to transient simulations with FFT analysis. This section was well-done. Also important is the discussion of noise simulations, where inadequate coverage was provided.

Chapter 11 discusses numeric measurements, models and some other miscellaneous items. Exporting numeric results from simulations to Excel, is an example. Many useful ways of making and taking measurements are described, some especially useful for the more advanced user.

Chapter 12, titled "Import of Components Models" covers the importation of component models, such as for transistors and ICs. However, it also covers the important areas of model creation, sub-circuit creation and libraries. It would have better been titled "Models, Subcircuits and Libraries." With regard to organization, some material at the end of Chapter 11 on models, sub-circuits and libraries should have been placed here.

Chapter 13 deals with the definition and use of voltage and current sources. This is one of the more important chapters since virtually all simulations require sources.

Chapter 14 covers passive components in detail, including some magnetic components that may be of limited interest to those not involved in switching power supply design. Since the typical use of passive components by most users is easy and fairly obvious, this chapter is less important for most audio applications.

Chapter 15 discusses semiconductor components and their models. This chapter defines the numerous parameters in the semiconductor models, but is not adequate by itself for most to use without additional significant knowledge of SPICE semiconductor models. This is not necessarily the fault of the book, and adequate in-depth coverage of models probably cannot be expected of a reference on a simulator. The important VDMOS model is covered, but the sub-threshold parameter added to LTspice at a later point is not covered.

Chapter 16 covers so-called accessory components like switches, logic gates and transmission lines.

Chapter 17 delves deeply into the details of inductors and transformers, with emphasis on the detailed models of these elements for use in switching power supply designs. This is a reflection of the author's special area of expertise. The audio designer may find much more detail here than necessary for typical audio circuits.

Chapter 18 explains the control panel, which is quite straightforward and includes often-used features like drafting options, waveform presentations and operation and, for example, what files are deleted after a simulation. The data compression and SPICE simulator parameters are also covered, but largely with only brief descriptions of the choices available. There is little in the way of helping the user know how or why to change these. For example, nothing is mentioned here about handling compression for good FFT analysis, an area of deep interest to audio designers. Similarly, little was said about how to use SPICE configuration parameters like *abstol*, *reltol*, etc. to improve convergence.

Chapter 19 illustrates nine application examples for LTspice, each in some good detail. I found this chapter to be enjoyable and quite valuable, and recommend that one read it at an early point. This is especially recommended for the less-experienced user.

Chapter 20 presents a variety of questions and answers about LTSPICE – a sort of FAQ. I found one of the questions particularly humorous: "Is there a risk of becoming addicted to LTspice IV?" I believe that the answer is yes for some of us.

The book concludes with *Chapter 21*, which reverts back to a discussion of inductors and transformers aimed at SMPS, largely continuing the material of Chapter 17. As such, the two chapters probably should have been grouped together. This chapter is also of limited interest to the audio designer. I found this to be an odd way to conclude the book.

My wish list

No book is perfect and at times I found myself wanting more in some sections. At other times I found the organization and groupings of material to be a bit choppy. Rather than a criticism, I'd like to summarize some of the things I'd like to see in the second edition – indeed, for a book this important, there really should be a second edition.

I'd like to see Chapter 10 expanded so that at least 10 or so pages are devoted to each of the six main simulations. The treatment of noise simulation is a primary example of where more discussion and examples are needed. Elsewhere, more complete explanations of transistor modeling and the use of SPICE core parameters like *reltol* would be helpful. In some cases, the addition of only a few more pages would make a big improvement. An appendix listing all of the components available under the AND symbol in the toolbar with a one-sentence explanation would also have been helpful. Finally the index, crucial for a reference text, should be greatly expanded.

Conclusion

Given the amount of time engineers spend in SPICE simulation, this book is a must-have for LTspice users. It is well worth the price. Once you have bought it, however, do make a significant investment in time reading it, even though this involves real work. This book is valuable to both the inexperienced and the very experienced user of LTspice. I thank Gilles for taking on the enormous task of documenting LTspice so thoroughly. The book was originally written in German, and a German version exists as well, so be careful to avoid confusion about which language version you order.