HOW2POWER TODAY
Your Power Design Newsletter

Book Review

ISSUE: June 2026

Author Explains And Demonstrates Methods For Achieving Successful Power Supply Simulations In SPICE

Simulating Switching Converters with LTspice, Christophe Basso, 132 numbered pages, glossy soft cover, 8.5 × 11 inch, Faraday Press, Ken Coffman, Editor at Stairway Press, ISBN 978-1-960405-64-7, 2026.

Reviewed by Dennis Feucht, Innovatia Laboratories, Cayo, Belize

Author Christophe Basso has designed converter control ICs for ON Semiconductor in Toulouse, France and has written related books reviewed in How2Power Today.[1] His books characteristically mix analytic derivations of circuit equations with a large amount of simulation. This book is almost entirely simulation-oriented, providing the reader with ready-made simulation templates in the form of circuit diagrams that execute on a free and popular version of SPICE issued years ago by Linear Technology.

Basso approaches simulation logically, by beginning with simple versions of circuits, to verify that simulation is convergent and free of errors at a basic level, then expands the simple model to be more comprehensive. Accordingly, the book begins with five pages—one per page—of simple op-amp models, beginning with a simple "generic model" with input shunt R and C and voltage clamping. The output has a dependent voltage source followed by an RC integrator. He then leads the reader through simulation testing of its static and dynamic behaviors. He adds additional circuit elements to the output, to make it more like a UC384x PWM controller.

Next, a two-winding transformer with magnetizing inductance referred to the primary side and with primary and secondary leakage inductances has a Norton equivalent circuit (shunt current source with equivalent magnetizing-inductance resistance) on the primary side and a Thevenin circuit (voltage source in series with resistance) on the secondary side. The dependent voltage source gain is the turns ratio. It is applied in the role of a current transformer, then expanded with an additional secondary winding followed by a transformer with a center-tapped primary winding.

The author has power circuits as the underlying theme running through these simulations, demonstrating how LT SPICE can be applied to their simulation. This is not a trivial feat because SPICE simulators historically showed convergence problems on circuits that are partially digital, partially analog.

Digital waveforms, being square-waves and having zero transition times, constitute mathematical discontinuities. In Fourier-series math, the Dirichlet conditions (found in most books on differential equations) show that at the point of discontinuity the Fourier series converges to the average between the two levels. In math, "analytic" means continuous functions with infinite resolution, and the convergence of the Dirichlet conditions is analytic.

However, simulators run on the computer mathematics of numerical analysis and are themselves inherently digital—that is, discontinuous and iterative in discrete steps of the circuit waveform variables. When a waveform discontinuity occurs, in some cases, the simulator, which solves the entire circuit as a giant matrix equation, can fail to converge to a solution with fixed numerical values. While seeking to home in on the final value, it might oscillate between two discrete values, never able to decide which one to resolve to.

To dive directly into this complication, the next pages of the book simulate an SR latch and D flip-flop. The D flop is built from the SR latch. Then that most basic of ADCs—the comparator, with analog input and digital output—is followed by a dead-time enhancement. This is followed by a "small-pulse generator," a couple of NAND gates connected to provide a "differentiated" pulse of gate-delay width. This path of circuit simulation ends with a "digital delay," a one switching cycle delay of a function, or in the z-domain, a z^{-1} function.

The next couple of templates are of optocouplers followed by the TL431 IC. (I also have given a large amount of explanation of this IC in my power-converters book and articles.) Here is yet another simulation of a part that I have found among manufacturers is not the same, not in its dynamics. The Bode plots differ among manufacturers. Basso has worked for ON Semi in the past, which was previously split off from Motorola. I suspect that his template follows the Motorola IC in its frequency response.

The next templates are of a VCO, a voltage-mode controller and a current-mode controller. In voltage-mode control, the controlled variable, which is the converter output voltage, is fed back to control the forward-path variable, the duty-ratio δ of the power-transfer circuit in the central subsystem of the converter. The circuit is designed to operate around a point of some fixed duty-ratio D and vary around it by some differential amount d so that $\delta = D + d$. The feedback configuration is that of a single loop and has in it the transfer-circuit inductor and output capacitor. They form an LC resonance within the feedback loop. This leads to a nontrivial feedback-control design problem.

Current-mode control is popular in design because it causes the switching power-transfer circuit inductor to effectively be in series with a current source and hence the inductor is removed from forming a resonance with the output capacitor. Nonetheless, it is even more complicated to analyze than voltage-mode control because it has an inner peak (or valley) control loop and an outer voltage-control loop.

The hint of a smile formed on my face as I read Basso’s line about current-mode control: “… slope compensation is necessary to calm down subharmonic poles located at $F_s/2$, but it is a well-understood process these days.” The statement is true; slope compensation is required for that purpose, but I would say that current-mode control from a modeling standpoint is only *relatively* well-understood in that a fourth-generation truly-unified model of it only emerged in the late 2010s (see my series on the topic[2]).

That model followed the previous so-called “unified” model of Tan and Middlebrook of the 1990s which followed the second-generation model in the late 1980s by Ray Ridley. Before that, the original Middlebrook model of the 1970s was quasistatic only and could not model the subharmonic oscillation that can occur at the Nyquist frequency $f_s/2$.

The model often used, as promulgated by Ray Ridley in his courses, goes about as deep into this topical abyss as most power-electronics engineers desire to descend and is quite adequate under those conditions. Yet it is not in itself a refined model in that it has theoretical errors, nor is the one by Tan and Middlebrook which is better than Ridley’s model in some ways (its sampled loop has a transfer function over the Nyquist interval; Ridley’s does not) and is worse in others (in how the transfer function is configured and the PWM is modeled).

Moving along into what is a more pleasant though also complicated topic nowadays, Basso gives a template for a quasi-resonant power-transfer circuit controller and for a resonant “asymmetrical half-bridge” controller. The name refers to the switch sequencing of the bridge and its use with a flyback power-transfer circuit.

The next six pages of the book cover similar LLC-related control circuits for various transfer circuits. Different labels are placed on them than what I use, so beware that there is not a standardized vocabulary for some of power electronics. The variations in circuits include differing ways to sequence the power switches.

After this, the topic turns to the beloved UC384X family of PWM controllers, with a page of explanation of the preceding variables to finalize this section of the book. Its organization is optimal for the kind of book this is. Each template has its own page and somehow conveniently fits it.

The topic now turns to control, and Basso categorizes control schemes by how many poles they have at the origin. In control language, for *n* poles this constitutes a type *n* control scheme. Basso follows this standard labeling. Types 1 through 3 compensators each have a page, then types 1 and 2 with a voltage-controlled current source, or operational transconductance amplifier (OTA), which Basso notes is a nifty IC circuit.

The next five pages are these control schemes with a TL431 followed by a couple of PID schemes. These proportional, integral, differential (PID) schemes are commonly found in motor-drives because of the inherent electromechanical resonance of motors, a resonance between the winding inductance on the electrical ~~a~~side with the electrically-referred inertia from the mechanical side as a capacitance.

Having such an LC resonance within a feedback loop is not to be taken lightly and the topology for a PID compensator involves parallel paths for the P, I, and D with outputs summed as the PID output. This kind of circuit can actually have more zeros than poles, unlike the condition placed on passive (and most active) circuits.

The book then moves along to digital compensators, suitable for µC-controlled converters. Basso shows in the next few pages how to quickly go from familiar *s*-domain circuit concepts of continuous control to the *z*-domain block diagrams of digital compensation. With this the first part of the book ends with subcircuit simulation, and turns to working with converter simulation in LTspice: importing models of subcircuits, simulating voltage- and current-mode buck (common-passive PWM-switch) power-transfer circuits, and using the frequency-response analyzer (FRA) instrument built into LTspice.

A page contrasts the ease of use of SIMPLIS with LTspice for frequency-response simulation. The next few pages illustrate the FRA use with buck converters with different control schemes, including constant on-time, and descriptions of their waveforms.

The simulations begin looking like whole converters, with a UC3845 control IC in a couple of the templates. Variations on buck or forward and boost transfer circuits appear with various control schemes. Monte Carlo analysis has a couple of pages. A PFC circuit with an onsemi controller has its page.

Because the LTspice simulator, typical of simulators, solves the circuit with total-variable (large-signal) equations, both the static and incremental results are simulated in a combined way. Several pages describe this and other PFC-related topics, then go to three-phase rectifiers for PFCs.

Multiple pages (97 to 115) then pertain to the common-inductor PWM-switch configuration of power-transfer circuits, as buck-boost and flyback, with variations on each to be modeled. Ćuk-class circuits (SEPIC, zeta) appear followed again by LLC circuits, including a bike charger. (Basso bikes in the Pyrenees of southern France, where he lives.) The book ends with a description of some of Basso's other books, references to papers and articles, and a page of Internet links.

References

1. "Text Is Comprehensive Guide To Converter Circuit Transfer Functions," by Dennis Feucht, review of *Transfer Functions of Switching Converters: Fast Analytical Techniques at Work with Small-Signal Analysis* by Christophe Basso, How2Power Today, August 2021.

2. "Special Series On Current-Mode Control" by Dennis Feucht, How2Power.com.

About The Author

Dennis Feucht has been involved in power electronics for 40 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 working on projects in theoretical magnetics and power converter research.

To read Dennis' reviews of other texts on power supply design, magnetics design and related topics, see How2Power's Power Electronics Book Reviews.