BY DAVID DIVINS • INTERNATIONAL RECTIFIER

# Use alternative approaches to evaluate dc/dc synchronous buck converters

LINEAR SYSTEMS, ONLINE TOOLS, AND FULL SIMULATIONS HELP YOU ANALYZE YOUR DESIGNS.

A number of methods exist for dc/dc-converter analysis. The one you choose depends on your proficiency as an engineer, the circuit requirements, and the availability of software tools. Before going into the analysis, however, you need to understand the makeup of these circuits, which are linear systems with feedback (**Figure 1**). You model them as classical feedback systems with transfer functions relating the output voltage to the reference voltage. You start by representing a generalized dc/dc regulator:

$$\frac{V_{OUT}}{V_{REF}} = \frac{G(s)}{1-H(s)\times G(s)},$$

where $V_{OUT}$ is the output voltage, $V_{REF}$ is the reference voltage, G(s) is feedforward, and H(s) is the direct feedback.

Taking a typical buck regulator circuit (**Figure 2**), you solve for G(s): $G(s)=D(s)\times G_P(s)$. You then derive the circuit-component values for optimum dynamic performance by using standard equation-manipulation and control-system theory. You can then rearrange **Figure 1** and rewrite the transfer function. This step allows you to determine the output voltage's and duty cycle's response to a load-current step. You can easily vary component values to determine their effect on the circuit's output. Tools that facilitate such mathematical manipulation include The MathWorks' (www.mathworks.com) Matlab and Simulink, open-source Maxima (http://wxmaxima.sourceforge.net), Gnu's Octive (www.gnu.org/software/octave/download.html), Scilab's (www.scilab.org) Scilab/Scicos, and PTC's Mathcad (www.ptc.com/products/mathcad).

You can design the compensation network and dynamic re-

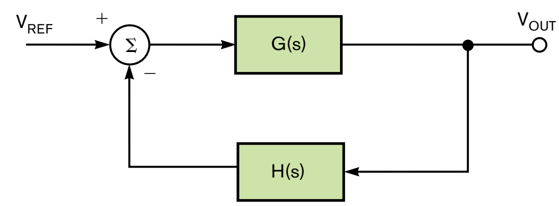


Figure 1 A dc/dc regulator is a linear system with feedback. You model it as a classical feedback system with a transfer function relating the output voltage to the reference voltage.

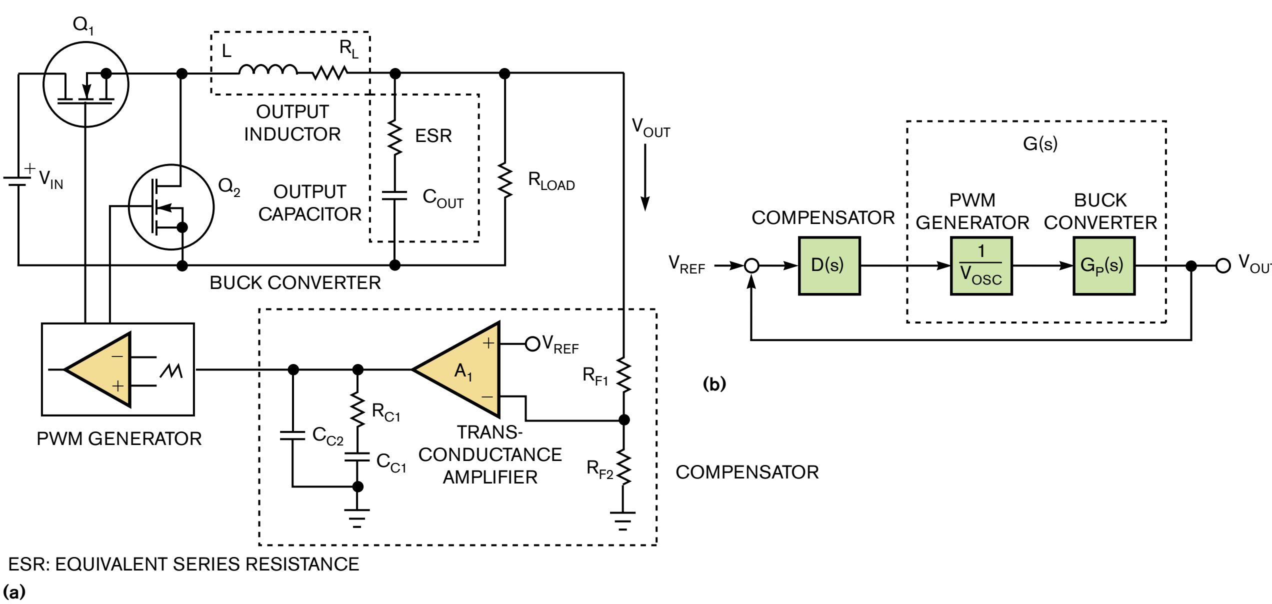


Figure 2 Taking a typical buck regulator circuit—shown here in circuit (a) and block-diagram (b) form—you solve for G(s): $G(s)=D(s)\times G_P(s)$.

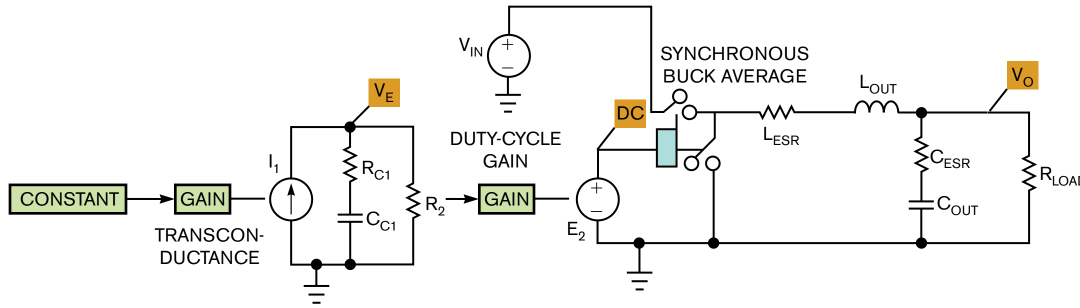


Figure 3 Simulation software allows you to represent a dc/dc converter as an open-loop state-average model.

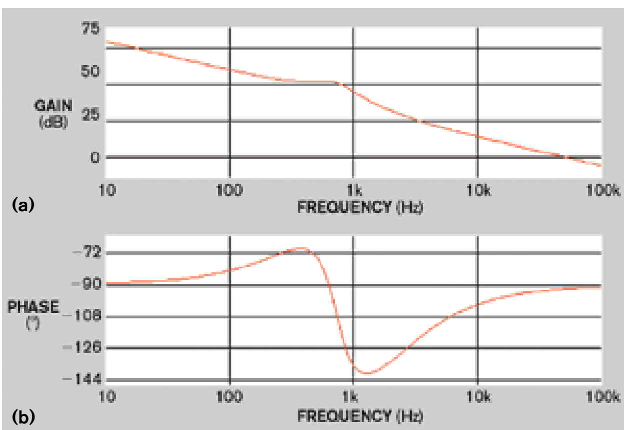


Figure 4 Using the design equations in the vendor's data sheets, you calculate the component values for gain (a) and phase (b); the simulator then validates the performance of the circuit, presenting it as an open-loop bode plot.

sponse of a dc/dc converter using a linear-system approach. This approach requires an understanding of control-system analysis and design and of Laplace-transform techniques. The most mathematically rigorous approach, linear analysis provides a good system model but no detailed circuit-performance information. Using the linear-system-analysis approach allows you to quickly simulate the open-loop response from the transfer function or block diagram to determine system stability—that is, phase and gain margins. The approach uses controller vendor provided transfer functions and block diagrams and data sheets or application notes. On the other hand, this approach does not allow you to analyze circuit behavior, such as inductor ripple current, output-voltage ripple, and efficiency. It also forces you to rewrite transfer functions or reconfigure block diagrams for each response type.

## ONLINE TOOLS

As an alternative to the linear-system approach, vendors of dc/dc-controller ICs provide online design tools targeting their controller offerings and providing various types of analysis. Because these tools target specific components, vendors have optimized the tools to provide fast and accurate results for the various analyses for designing and implementing a dc/dc regulator. You can use these tools to visualize control-loop compensation and dynamic performance, such as step response, ripple voltage, ripple current, and efficiency. Vendors also help you choose the accompanying MOSFETs the converter uses. These tools use custom routines and Spice as their mathematical engines.

This approach requires little mathematical manipulation on the user's part. Usually, the user needs only to derive the compensation-component values and enter data into an online form. Some vendors' online tools aid you in deriving the compensation-component values. Each vendor formats its tools in a unique way, providing a different view of the same prob-

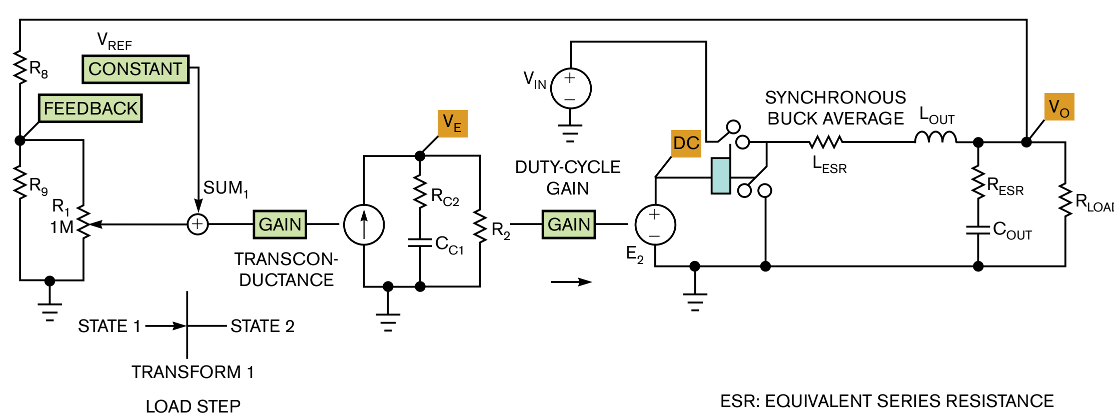


Figure 5 Software allows you to model a dc/dc converter in a closed loop using a state-average model.

lem. Companies including Fairchild Semiconductor (www.fairchildsemi.com), International Rectifier (www.irf.com), Intersil (www.intersil.com), National Semiconductor (www.national.com), and Texas Instruments (www.ti.com) provide such tools as offline Spice simulators, MOSFET-selection guides, reference-board designs and support, online simulation, and Spice models. Each vendor has a method for providing design aid to the engineering community. Vendors that use online tools control the analysis types and provide accurate simulation results. This approach provides an alternative to manual calculations and offline simulation. As with linear-system analysis, online tools have both pros and cons.

On the plus side, the vendors of these tools validate them, they are easy to use, they typically have associated demo boards, and they require no third-party software. On the other hand, they provide limited what-if analysis and minimal parameter-variation capability. They also don't allow you to change the circuit topology from the one that the vendor presents.

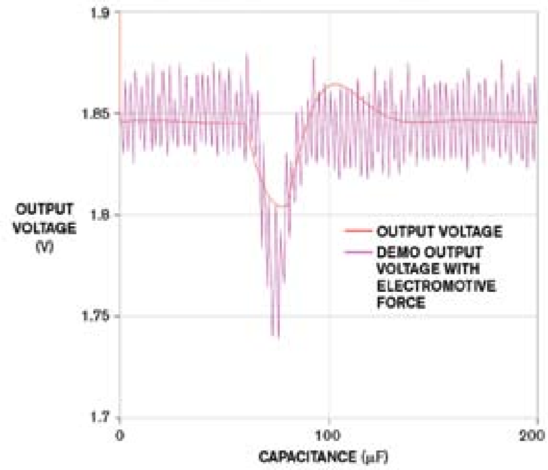

Figure 6 The results of a state-average simulation correlate well with the measurements from a demo board. The simulation does not show the converter output ripple, however.

## USING FULL-CIRCUIT SIMULATION

Engineers have used circuit simulation for more than four decades for IC design and for 25 years for power-electronics design. The tool of choice has long been Spice, which is still the de facto standard in circuit simulation and analysis. Other high-power circuit simulators are available, however, including Synopsys' (www.synopsys.com) Saber, Mentor Graphics' (www.mentor.com) Eldo and SystemVision, Ansoft's (www.ansoft.com) Simplorer, and Cadence's (www.cadence.com) Spectre. The value of offline simulation and analysis is its flexibility in circuit topology and postprocessing analysis. Simulation allows for the same types of analysis as linear-system analysis and vendors' online tools. However, this approach also allows you to perform a number of other

valuable analyses, such as state-average modeling. This type of modeling combines the design techniques and speedy results of control-system analysis. State-average modeling allows for both open-loop and control analysis in the circuit domain instead of the linear-system domain.

You can create a model of a dc/dc-POL (point-of-load) converter, which includes the gain of the transconductance feedback amplifier, the compensation components, and the fixed duty-cycle-conversion gain. You can then create a state-average model of the PWM (pulse-width-modulation)/power stage, the output filter, and the load (**Figure 3**). The state-average model for the PWM/power stage simplifies and linearizes the circuit for fast simulation and allows for linear-system analysis.

You can start with the transfer function for the state-average model: $V_{OUT} = D \times V_{IN}$. Then, using the design equations in the vendor's data sheets, you calculate the component values. The simulator can then validate the performance of the circuit by running an ac analysis (**Figure 4**). Full-circuit simulations also allow you to create a macro model of a closed-loop dc/dc-POL circuit using a state-average model (**Figure 5**).

The simulation results from a state-average model can produce good steady-state and dynamic results when you compare them with those from an actual circuit (**Figure 6**). The results show dynamic information but not ripple information. State-average models provide 20-times-faster simulation than does a PWM simulation. This method of dc/dc-converter analysis also has benefits and drawbacks. For example, it offers freedom from circuit-topology constraints and postprocessing analysis. It also offers both state-average and full-PWM modeling. On the other hand, not all vendors' models are available, and the state-average model has simulation-convergence issues and requires a knowledge of macro modeling.

With the maturity of simulation as a design and validation tool and the emergence over the past few years of online tools, you have a plethora of methods for analyzing dc/dc converters. Both system- and circuit-simulator tools can also play important roles because of their inherent flexibility and their history of use. In addition, engineers have a good understanding of dc/dc-POL converters, and there is much available information on them. So, no matter which design tack you take, a suitable tool is available to accomplish the job. **EDN**

## AUTHOR'S BIOGRAPHY

*David Divins is a senior applications engineer at International Rectifier, where he has worked for eight years. In his current position, he designs discrete power components and analog ICs into industrial and automotive applications. He has a master's degree in electrical engineering from the State University of New York—Binghamton. His personal interests include circuit simulation and analysis.*