Keywords: SIMPLIS, SPICE, power circuit simulation, power circuits, simulation, power simulation engines

APPLICATION NOTE 6506

WHY SIMPLIS IS BETTER THAN SPICE FOR POWER CIRCUITS

By: Brooks Robert Leman and Erin Mannas

Abstract: Power circuits are difficult to simulate; however, simulation is an essential way to help ensure that the design will work in production. In the end, simulation also saves time and money. This application note compares two power simulation engines, SIMPLIS and SPICE, and shows why SIMPLIS is ultimately the more effective option for power circuits.

Introduction

To boost the chances that a design will work properly while in production, most engineers turn to simulation. It provides an assessment of how the design will function, which can save time in the design process and, ultimately, costs. After all, redesigning and respinning PCBs is a costly endeavor. Remember the days when it was common to design boards, test them in the lab, debug, redesign, and repeat again and again until the design was right? This approach would often delay product introduction. And worse, there would be times when not all problems were caught before the product was shipped to customers.

Power management engineers, however, have been late to integrate simulation into their processes. By comparison, the semiconductor industry recognized decades ago that it could only be successful by completely simulating IC designs before fabricating the first wafers. Maxim, for instance, measures simulation success by the number of its ICs that go to production with “first silicon.” Power supplies and converters are notoriously hard to simulate. What’s more, the simulation solutions optimized for ICs are not necessarily the best tools for power conversion simulation.

That said, there are two simulation engines that have emerged for power supply circuits: SIMPLIS and SPICE. In this application note, we’ll compare the two and examine why the SIMPLIS engine is more effective for power designs.

How Healthy is Your Power Converter Design?

There’s a set of "vital sign" tests to establish the health and robustness of a power converter design that are analogous to medical vital signs that provide an overview of a human’s overall health. Based on industry practices, Maxim’s EE-Sim® design generation and simulation environment defines these tests to be:

1. Load step
2. AC loop
3. Steady state
4. Line transient
5. Startup
6. Efficiency

Like taking a patient's pulse in our human health analogy, load step is arguably the most important vital sign to assess in a power converter design. Just as a person's pulse changes with exercise, the output voltage of a power converter changes when it is exercised by a change in the load current. Load-step simulation measures how much the output voltage changes and how quickly it recovers. If the feedback circuit is not properly designed, a variety of things can happen. The converter can overshoot or undershoot too far, ring excessively, recover too slowly, or break into oscillation, for instance. An engineer with a trained eye can qualitatively judge the effectiveness of the control loop by inspecting the load-step transient response graph. For a more complete picture of the "health" of the control loop, AC loop analysis is used.

Looking at the control loop in the frequency domain, AC loop analysis enables direct measurement of the control-loop bandwidth and phase margin (going back to our human health example, this step is like taking a patient's blood pressure). AC analysis, also known as small signal, Bode, or frequency-response analysis, requires specialized equipment that's not commonly found in the lab (examples include AP Instruments AP300, Omicron Bode 100, or Agilent 4194A or 4195A). When available, a Bode analyzer injects a signal into the control loop and then measures the signal at various points in this loop to establish the gain and phase shift between two signals. After the signal is swept over a frequency range, the gain and phase response are plotted on a log scale. Simulating this analysis is particularly valuable since the analysis may not be available in the lab.

One could argue that steady-state analysis is an oxymoron for switched-mode power conversion. A better description is equilibrium analysis. With a converter in equilibrium, every switching cycle looks just like every other switching cycle (similar to a patient's respiratory rate in our human health analogy). Cycles that aren't identical could be a sign that the converter may be oscillating. Steady-state operation can actually be observed during load-step tests by zooming in for a closer inspection of the waveforms between the load steps. Using a separate steady-state analysis is actually just a convenience.

Another way to disturb the control loop and observe its recovery is via line transient. The input voltage is quickly stepped between two values while observing the output voltage. Some applications (such as audio) are particularly sensitive to line-transient performance. But, for the most part, this test is less important than load step.

Startup examines specifically what happens when input voltage is first applied (or an enable pin is asserted). As it reaches the regulated value, the output voltage should smoothly ramp up in a relatively slow manner with little or no overshoot. Typically, before the power converter is turned on for the first time, control-system health should be verified by simulating load step, AC loop, and then start up before heading to the lab.

Efficiency analysis establishes converter power losses which lead to an estimate of component temperature rise (just like taking a patient's temperature). High losses mean low efficiency and a design that produces excessive heat. If the converter is oscillating, component stresses and power losses are higher, while efficiency is lower.

Before any of these tests can provide useful direction, the converter must be stable and not oscillating. That's why the tests that directly establish the control system stability and responsiveness—load step and AC loop analysis—are the most important.

How Do SIMPLIS and SPICE Differ?

Let's now take a closer look at the SIMPLIS and SPICE engines. SIMPLIS (SIMulation of Piecewise Linear Systems) emerged in the 1980s, created initially for fast modeling of switched-mode power converters. SIMPLIS Technologies now develops and owns the engine. Today, SIMPLIS is a popular engine for power conversion circuit simulation as well as new ICs definition analysis. SPICE (Simulation Program with Integrated Circuit Emphasis) was developed in the 1970s at UC Berkeley. It's a general-purpose, open-source analog electronic circuit simulation engine. Many consider SPICE circuit simulation to be the industry-standard method to verify circuit operation at the transistor level.

The best way to compare the SIMPLIS to SPICE engines is by looking at how each handles the most critical "vital sign" power conversion tests. There are really two real differences between SIMPLIS and SPICE. First, SIMPLIS generates the load-step analysis much faster than SPICE can: you can get an accurate result 10x to 50x faster.

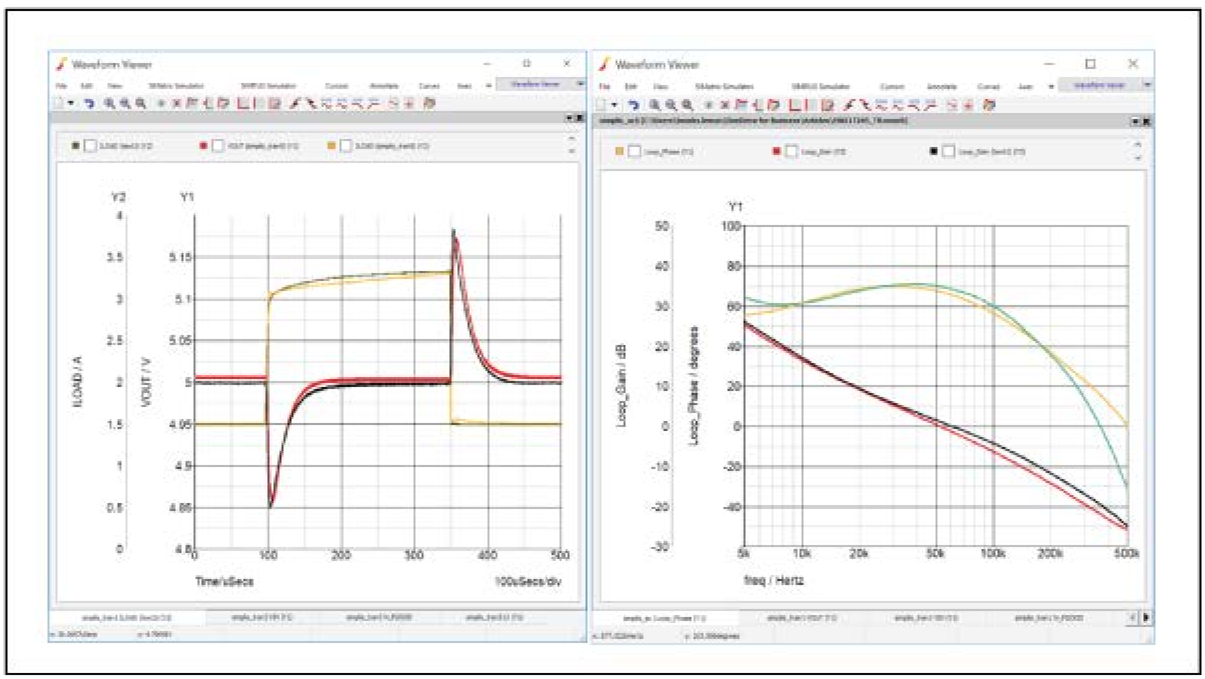

Figure 1. Both step-load transient response (left) and AC analysis Bode plots (right) show very good agreement between simulated vs. measured results for the MAX17244 synchronous buck converter. Lab data is shown in black (V_{OUT}, gain) and green (current step, phase), while simulated data is shown in redand orange.

Second, the AC loop analysis requires an order of magnitude less effort with SIMPLIS as it does with SPICE. With SPICE, AC loop analysis calls for a considerable amount of extra time and attention. SIMPLIS, on the other hand, was designed specifically to deliver the AC loop analysis as a by-product of fast time-domain simulations.

To render the Bode plot, SPICE requires a lot of special attention. In fact, some users simply skip SPICE-based AC loop analysis and instead try to deduce control loop robustness from the load step transient response alone. However, this is sort of like skipping a patient's blood pressure measurement, relying instead on the patient's pulse.

There are a variety of approaches to performing SPICE-based AC loop analysis. You can run several different SPICE transient time-domain simulations, each with a unique sinusoidal perturbation source. You could then perform fast Fourier transform (FFT) algorithms on each result and post-process all the results into a Bode plot. This is, however, a lengthy process that could take potentially hours, depending on the number of data points plotted.

Another method of performing AC loop analysis in SPICE is to create an "averaged" or "small-signal" model, without switching, which runs faster in SPICE than the switched SPICE model. This power-conversion small-signal modeling issue was recognized decades ago. Cal Tech and Virginia Tech researchers have thoroughly analyzed the matter and have succeeded in making small-signal models practical. Even so, the two different SPICE models still need to be correlated.

Finally, some engineers opt to create a second calculation engine to avoid the SPICE/Bode plot problem. This approach involves using code or Excel to solve the Bode plot problem without using SPICE. It is, however, a time-intensive and expensive method. Fortunately for power designers, there are now some better solutions.

What Makes SIMPLIS a Better Choice?

Many engineers continue to use SPICE to simulate power supply circuits, even though the engine wasn't originally developed for this purpose—and despite its drawbacks for switched-mode ICs. Because SIMPLIS models devices via a series of straight-line segments vs. solving nonlinear equations as SPICE does, SIMPLIS can perform 10x to 50x faster than SPICE with the same level of accuracy. As a result, SIMPLIS can characterize a complete circuit as a cyclical sequence of linear circuit topologies. Since SPICE resolves all voltages and currents in small increments in very accurate detail, it is also painstakingly slow. SPICE also requires the use of an average model and a switching model for simulation, while SIMPLIS uses only one model.

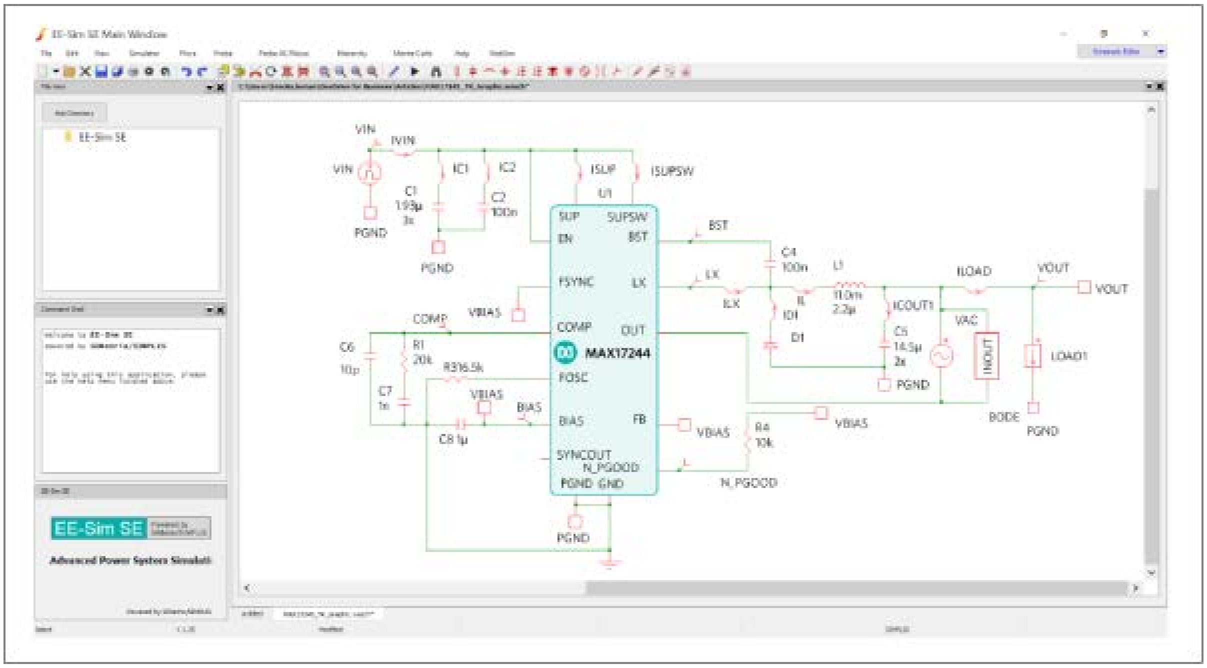

Figure 2. MAX17244 SIMPLIS synchronous buck converter schematic.

The conclusion, therefore, is simple: SIMPLIS provides SPICE-level accuracy in a much faster timeframe. You can then devote more of your resources toward your design effort, rather than getting bogged down with simulation tasks.

A similar version of this application note appeared on EDN on September 20, 2017.

EE-Sim is a registered trademark of Maxim Integrated Products, Inc.

More Information
For Technical Support: https://www.maximintegrated.com/en/support
For Samples: https://www.maximintegrated.com/en/samples
Other Questions and Comments: https://www.maximintegrated.com/en/contact

---

Application Note 6506: https://www.maximintegrated.com/en/an6506
APPLICATION NOTE 6506, AN6506, AN 6506, APP6506, Appnote6506, Appnote 6506