SIMPLIS vs. SPICE: Which is better for simulating power circuits?

Brooks Leman, Erin Mannas - September 20, 2017

One simulates up to 50× faster than the other, and both will give you comparable accuracy. Which engine would you choose for your power supply circuit? Sounds like a trick question, but it isn’t. Before answering that question, though, let’s first consider why we simulate in the first place. By establishing the desired outcome from simulation, we can objectively compare the differences between the two engines in question, SIMPLIS and SPICE.

Most engineers will agree that simulation is about increasing the chance that your design will work when it’s in production. Saving time is great, too, since you’re always under time-to-market pressures. And, ultimately, you want to save some money in the process.

Face it, redesigning and re-spinning PCBs is expensive. Back in the day, we would design boards, test them in the lab, debug, redesign, and repeat the whole cycle again and again and again, until we got it right. Often, this process delayed product introduction—and, if you were unlucky, it would allow competitors to slip their products into customers’ reach first. There would also be the unfortunate times when not all the problems were caught, and thousands (or millions) of products were built before customers found problems that needed to be corrected.

Decades ago, the semiconductor industry recognized 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.” The power management industry, however, has been late to the simulation party. Power supplies and converters are notoriously difficult to simulate, and the simulation solutions optimized for ICs are not necessarily the best tools for power conversion simulation.

Testing the health of your power converter design

Given that simulation saves rework, time, and, ultimately, money, what simulations do you need to run? To establish the health and robustness of a power converter design, either by simulation or lab measurement, there is a set of “vital sign” tests analogous to medical vital signs used to quickly establish 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. Start up
6. Efficiency

Load step is arguably the most important, akin to taking a patient’s pulse. Just like a person’s pulse will change with exercise, the output voltage of a power converter will change 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, the converter can overshoot or undershoot too far, ring excessively, recover too slowly, or break into oscillation. The 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, there’s the AC loop analysis method.

AC loop analysis looks at the control loop in the frequency domain, enabling direct measurement of the control-loop bandwidth and phase margin (going back to our medical 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 such as the AP Instruments AP300, Omicron Bode 100, or Agilent 4194A or 4195A, which are not commonly found in the lab. When available, a Bode analyzer injects a signal into the control loop and then measures the signal at various points in the control loop to establish the gain and phase shift between two signals. The signal is swept over a frequency range and the gain and phase response are plotted on a log scale. Since this analysis may not be available in the lab, being able to simulate it is especially valuable.

Steady-state analysis is arguably an oxymoron for switched-mode power conversion; equilibrium analysis would be a better description. With a converter in equilibrium, every switching cycle looks just like every other switching cycle, kind of like a patient’s respiratory rate. If the cycles are not identical, 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.

Line transient is another way to disturb the control loop and observe its recovery. The input voltage is quickly stepped between two values while observing the output voltage. There are some applications specifically sensitive to line-transient performance (audio, for example), but most of the time this test is less important compared to load step.

Start-up looks specifically at what happens when input voltage is first applied (or an enable pin is asserted). The output voltage should smoothly ramp up relatively slowly with little or no overshoot as it reaches the regulated value. Typically, before turning on the power converter 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). If losses are high, efficiency will be low and the design produces excessive heat. Note that if the converter is oscillating, component stresses will be higher, power losses will be higher, and efficiency will be lower.

Before any of these tests have meaning, the converter must be stable and not oscillating. Therefore, the tests that directly establish the control system stability and responsiveness are the most important: load step and AC loop analysis.

Key differences between SIMPLIS and SPICE
Now, back to SIMPLIS versus SPICE. SIMPLIS (SIMulation of Piecewise Linear Systems) came around in the 1980s and was developed 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 system simulation as well as new product definition analysis. Developed in the 1970s at UC Berkeley, SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose, open-source analog electronic circuit simulation engine. SPICE circuit simulation is considered the industry-standard method to verify circuit operation at the transistor level.

The best way to compare SIMPLIS to SPICE is to see how each handles the most critical “vital sign” power conversion tests. It comes down to two real differences between SIMPLIS and SPICE. First, SIMPLIS generates the load-step analysis much faster compared to SPICE. You get an accurate result in 10x to 50x faster time compared to SPICE.
Both step-load transient response (top) and AC analysis Bode plots (bottom) show very good agreement between simulated versus measured results for the MAX17244 synchronous buck converter. Lab data is shown in black ($V_{\text{OUT}}$, gain) and green (current step, phase), while simulated data is shown in red and orange.

Second, with SIMPLIS, the AC loop analysis requires an order of magnitude less effort compared to SPICE, where AC loop analysis requires a considerable amount of extra time and attention. SIMPLIS was designed specifically to deliver the AC loop analysis as a by-product of fast time-domain simulations.

SPICE requires a lot of special attention to render the Bode plot. In fact, some users skip SPICE-based AC loop analysis altogether and try to deduce control loop robustness from the load step transient response alone. But isn’t this like skipping a patient’s blood pressure measurement and relying only on taking the patient’s pulse?

If AC loop analysis is to be performed using SPICE, there are several approaches that can be tried. One way is to run several different SPICE transient time-domain simulations, each with a unique sinusoidal perturbation source, perform fast Fourier transforms (FFTs) on each result, and then post-process all the results into a Bode plot. However, this process takes a long time—potentially hours, depending on how many data points are plotted.

Another method of performing AC loop analysis in SPICE is to create an “averaged” or “small-signal” model, with no switching, which runs faster in SPICE than the switched SPICE model. This power-conversion small-signal modeling problem was recognized decades ago. The problem has been adequately analyzed by researchers from Cal Tech and Virginia Tech, who succeeded in making small-signal models practical; however, the two different SPICE models still need to be correlated.

It is also common for companies to avoid the SPICE/Bode plot problem altogether by creating a
second calculation engine, in code or Excel, specifically to solve the Bode plot problem without using SPICE. However, this approach takes a lot of time and expense.

Finally, one SPICE tactic commonly used is to avoid the Bode plot simulation and rely solely on the load-step response to gauge robustness of the control loop. This reckless approach may have been more appropriate decades ago, before SIMPLIS or small-signal models were available. Fortunately, better solutions are available today.

**Why SIMPLIS is better than SPICE for power circuits**

Although SPICE was not developed specifically for power supply circuits, many companies continue to use it for this purpose, even with its drawbacks for switched-mode ICs. SIMPLIS can perform 10x to 50x faster than SPICE with the same level of accuracy by modeling devices via a series of straight-line segments, instead of solving non-linear equations as SPICE does. Through its approach, SIMPLIS can characterize a complete system as a cyclical sequence of linear circuit topologies. SPICE, on the other hand, resolves all voltages and currents in small increments in very accurate detail. As such, the SPICE approach is painstakingly slow. SPICE also requires the use of two models for simulation: an average model and a switching model, compared to SIMPLIS, which uses only one model.

![Figure 2 Example of EE-Sim DC-DC Converter Tool in action with MAX17244 SIMPLIS synchronous buck converter schematic.](image)

So there you have it: with SIMPLIS, you simply get SPICE-level accuracy much faster. Rather than spend excess time simulating, don’t you want to devote more of your resources toward your design effort—and getting to market before the competition?

*Brooks Leman* is senior principal member of technical staff at Maxim Integrated.

*Erin Mannas* is the product manager for the EE-Sim design and simulation tools at Maxim Integrated.
Also see:

- Spice simulation, Tina-TI, LTSpice, PSpice, and more
- It's time to SPICE up your life!
- Simulation shows how real op amps can drive capacitive loads
- Testing a power supply