Authors

Gobi K A
Applications Engineering Manager, Synopsys

Alan Courtay
Principal R&D Engineer, Synopsys

Overview

Power electronics systems are at the heart of many important and growing industries, from all-electric vehicles to renewable energy generation. Optimizing the design of these systems requires accurate modeling and simulation long before construction of physical prototypes. SPICE-level simulation has been the traditional solution and, while it still plays a key role, it cannot satisfy all requirements for contemporary design and verification flows. This white paper describes a comprehensive solution for the simulation and virtual prototyping of power electronics, ranging from early power module designs to signoff of large-scale, high-fidelity systems. The result is a more efficient development process yielding physical systems that behave as modeled and are much less likely to require fixes or redesign.

Drivers for Power Electronics

The modern world relies on electric power in countless ways. It is almost impossible to imagine life without electricity and the many applications it enables. This reliance places high demands on the reliability and efficiency of the systems that generate, distribute and consume electric power. When people think of power, they tend to think of the electric and electro-mechanical devices that have been in use for more than one hundred years: generators, motors, transformers, switches, etc. These are still very much part of the picture, but semiconductors are playing an ever-increasing role in conditioning and shaping electric power. This is the domain of power electronics.

There are numerous drivers for the growing importance of electronics to control and convert power. The wide range of applications relying on electric power has vastly different requirements, so sophisticated electronics are needed to supply power in ways that satisfy these diverse requirements. A smartphone and an electric vehicle (EV) both run from batteries yet clearly have varying needs for power control. Even within a single system, different subsystems often need power in different forms. A power electronics interface is inevitably used to supply power in a system with varied requirements.

One measure of the increasing role of power electronics is the growth in commercial products. Figure 1 shows the estimated market size for 2020 and the projected market size for 2025, with a compound annual growth rate (CAGR) of nearly five percent.

There are many applications driving innovation in power electronics. Perhaps the most visible today are EVs whose performance rivals traditional combustion-based vehicles. Driving range is a critical factor in EV's growing popularity, so energy must be consumed efficiently while satisfying the performance demands. Some of the key subsystems are the propulsion/powertrain, battery charging systems, photovoltaic power conversion and power factor correction.

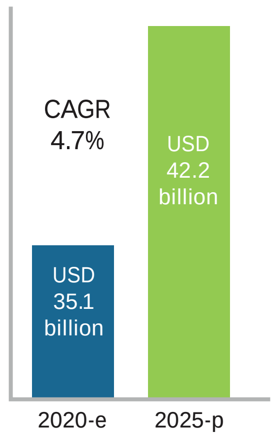

Figure 1: Power electronics market (Source: MarketsandMarkets)

Hybrid vehicles have complex connections between the battery and combustion systems, with power electronics managing every aspect of this interaction. The goals of green energy and de-carbonization are pushing the move to EVs and hybrids. Governmental incentives are also a factor, and several major European cities are heading toward a total ban on internal combustion engines in the near future. Some observers expect a similar boom in electric and hybrid aircraft over the next decade. Acceptable flying speeds and ranges will depend heavily on power electronics.

Green energy and de-carbonization are also influencing the sources of electricity. Renewable energy generation is growing in importance and, in many cases, supported by green legislation and tax incentives. These systems also need efficient power electronics equipment for extracting maximum power from the source. Energy harvested from solar and wind sources, with their constant changes, must be converted and controlled before feeding into the power grid. Power electronics help to manage the electrical grid, with the goal of maximizing efficiency while minimizing blackouts or brownouts.

Power electronics play an important role in many applications, including everyday consumer usage. All battery-powered devices, most notably smartphones, tablets and laptops, must make the best possible use of their limited battery capacity. In addition, operation must continue without interruption as devices are plugged into and unplugged from battery chargers. Power management in these devices has become quite sophisticated, able to sense and respond to changing conditions and adapt dynamically. Other applications where power electronics systems are vital include military and aerospace, industrial equipment, and medical devices.

Traditional Development Flow

Not so long ago, most electronics systems were developed primarily by building a few prototypes and validating them in the bring-up lab. There are many limitations with this approach. It is expensive and time consuming since it can be hard to find design flaws and costly to rework the hardware to fix them. The expense of building and updating prototypes limits the number that are built and therefore the number of engineers who can work on them in the lab. It is also difficult or impossible to cover all testing scenarios, especially checking for proper response to faults. Injecting some faults can damage the system, while other types of faults cannot be injected at all.

Reliance on bench testing breaks down completely if fabricated custom integrated circuits (chips) are included in the design. It can take months and cost millions of dollars to fix a flaw found in silicon. For all these reasons, simulation of the design before building prototypes or chips is critical. Historically, the simulation of circuit analog behavior has been performed using SPICE ("Simulation Program with Integrated Circuit Emphasis") and SPICE-like tools and models. SPICE works well for integrated and board-level circuits, but there are significant limitations on how and where it can be used in the development of a power electronics system.

The stiff nature of power electronics circuits can cause difficult convergence for SPICE-like tools, preventing them from reliably simulating circuits with more than a handful of diodes and power switches (e.g., IGBTs or MOSFETs). The integrated circuits for which SPICE was originally built do not present this numerical challenge because they include at almost every net a grounded capacitor that tends to smooth the transition edges of switching transistors. Due to the sharp transitions of power electronics circuits, SPICE is required to take small time steps which can result in ill-conditioned matrices and convergence errors.

The cyclical nature of pulse width modulation (PWM) control techniques ubiquitous in power electronics is also a cause of slow simulations. SPICE does not recognize the set of discrete topologies that the circuit traverses repeatedly and where certain switches are conducting while others are blocking. In addition, SPICE does not cache or reuse any of its numerical processing for repeated switching sequences.

Also, SPICE analysis can be only as accurate as its circuit models, so the design engineer must have access to high-fidelity models for the transistors and other elements within chips, any discrete devices, and interconnects such as printed circuit boards and cables. The most accurate models are provided by component manufacturers and are specific to a given part number. Thus, SPICE simulation generally cannot be performed until most design decisions have been made, including which parts from which suppliers will be used.

The result is that design flaws are discovered late in the development process, when the initial design is believed to be nearly complete. Many problems identified by SPICE can only be resolved by revising the design. Any change in the parts list requires identifying new manufacturers, selecting new parts, obtaining new SPICE models and rerunning all simulations. This extends the project timeline and delays time to market, enabling more competition and reducing profits. In the worst-case scenario, a late product would no longer be competitive, and the entire project is canceled.

As shown in Figure 2, the typical power electronics development flow is a multi-stage process. Based on the overall product requirements, the power electronics designers start with individual modules with ideal models and then develop the system-level design. Next, invely ,ue topologies an“carrifieou (t witv r)6.2enaru is s h6.7 (o(er).5 (ds and)]TJ 0 -1.444 Td s sbidelitE analysns. 719.6 (Ths SPICE simulatiothid

Rather than running early simulations, some power electronics designers rely on “rule of thumb” analysis and add huge margins to reduce the chances of surprises late in the project. Of course, this results in inefficient development and unoptimized designs. Some designers try to get around the limitations of SPICE and SPICE-like tools by using signal-flow based or other point simulation tools. Transitions between point tools or from point tools to SPICE are challenging since there's no unified flow. Point tools cannot provide a comprehensive and scalable simulation solution to efficiently simulate power electronics circuits at all levels from concept to product.

A Comprehensive Simulation Flow

The best way to improve upon the traditional approach is to use a more comprehensive and scalable simulation technique. Using simulation in the early stages of development ensures that the module-level and system-level design is well understood and analyzed. A simulation model provides a comprehensive understanding of the design and gives access to parameters that are difficult to observe and test on the hardware. Simulation helps to reduce the number of hardware iterations by weeding out design issues early. All these benefits apply to the early design phase, but a simulation method that helps to develop large-scale, multi-domain detailed designs with high-fidelity models and to perform various worst-case and reliability studies faster is also needed.

The key to earlier simulation is the use of piecewise linear (PWL) circuit models and specialized engines to solve them. The PWL approach approximates device characteristic curves with a series of line segments. Figure 3 provides some examples of different circuit elements and PWL modeling of their characteristics. Due to their abstract nature and reduced number of equations, PWL models simulate much faster than SPICE models. Power electronics designers can use generic models for ideal devices rather than part-specific models at early stages of the project when part selection has not yet taken place. PWL models require less setup and run quickly enough to enable exactly the sort of what-if scenarios that bring the most value.

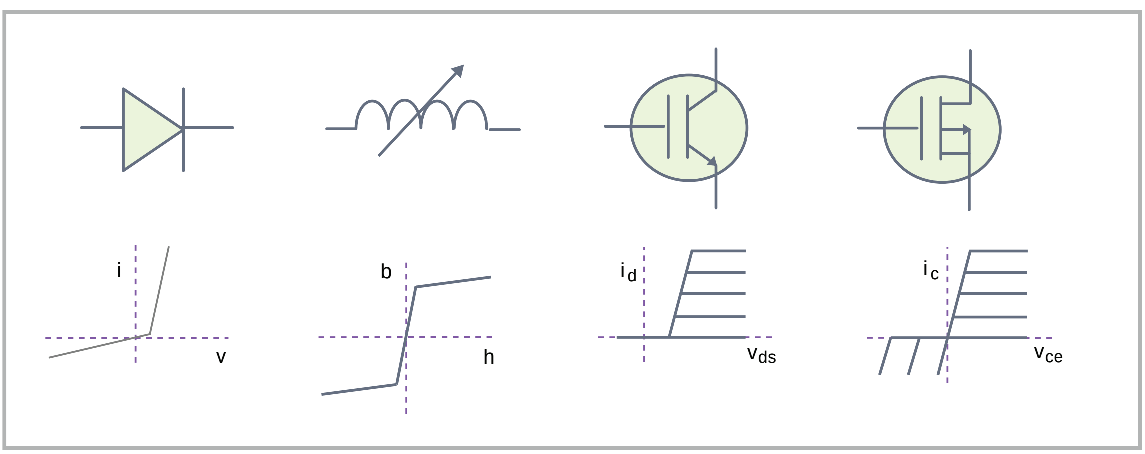

Figure 3: Examples of piecewise linear models

Linearized model curves enable high-speed simulation during proof of concept and design exploration, while high-fidelity model curves support device-specific detailed simulation. Figure 4 shows how the models compare. The early stage design simulation deals with a lower level of complexity with many teams working independently. A tool that provide quick but accurate simulation without much work on the simulation setup is ideal at this stage.

Abstract PWL models are also generally enough to design many aspects of power electronics systems. For example, the detailed switching transient of power diodes, MOSFETs and IGBTs is rarely the core problem of the engineer designing the control of switched mode power supplies. At this design stage, such devices can conveniently be represented as resistances toggling instantaneously between a high and a low value.

As the power electronics design matures, parts are selected, more advanced analysis is performed, and model and design complexity increases. At this stage, a more accurate simulation tool that can evaluate system-level performance is required. It is also necessary to analyze the detailed transients of power semiconductor devices to assess switching losses and EMI aspects.

SaberEXP combines a high-speed PWL solver with an event queue for event-driven and digital signals. It caches and reuses recurring circuit topology calculations to make runtimes even shorter. Its integrated simulation environment and easy-to-use interface ensure first-pass success. SaberEXP is tuned precisely for early design exploration and architectural what-if analysis. It provides parametric and statistical design capabilities for optimizing and verifying system robustness. Analyzing results is conveniently done with a suite of measurement and waveform calculation tools.

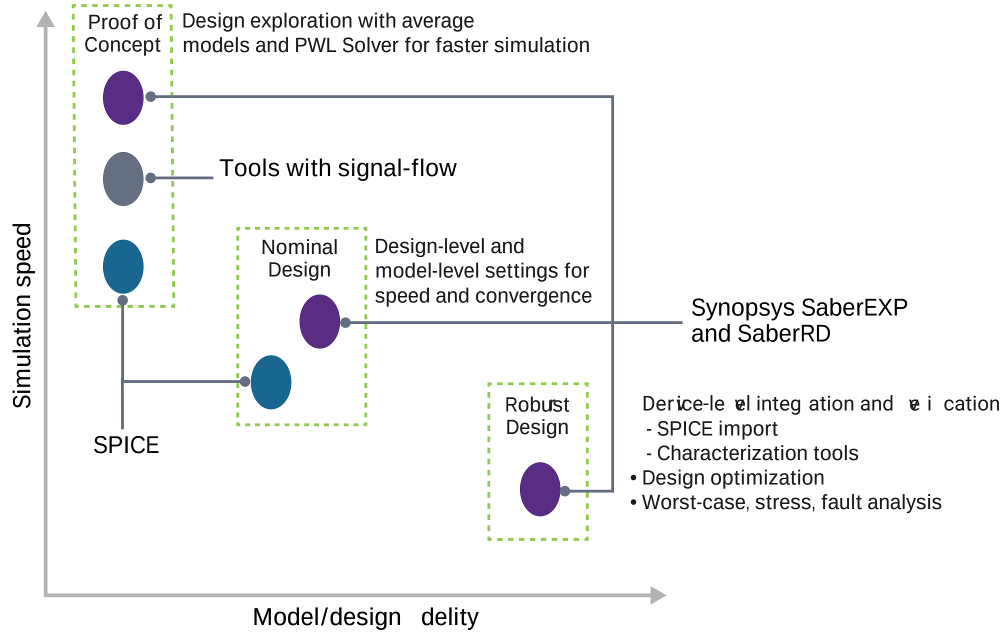

Figure 6: Comparison of alternative solutions

Summary

Power electronics systems are growing in importance and complexity, driven by a wide range of applications. Finding design flaws in lab prototypes occurs too late in the project, so simulation is required. SPICE and other point tools cannot handle the entire development flow, from architectural what-if analysis to circuit characterization and optimization. Synopsys SaberEXP and SaberRD proo-end solution. A single standard simulation and virtual prototyping environment throughout the process makes training, modeling, results sharing, support and supplier management more efficient. This solution minimizes costly physical hardware testing, saves project resources and accelerates time to market.

10/14/20.CS577651945-Comprehensive Simulation of Power Electronics Systems WP.
Pub: October 2020