ISSUE: September 2016

Power Electronics Simulation Platform Adds SPICE Engine

Powersim has added the SPICE engine to its core simulation tool, PSIM. The addition will provide the PSIM simulation platform, optimized for topology verification and control design, a new SPICE engine to support the vast library of industry-standard SPICE models for a more in-depth simulation (see the figure.) PSIM users will have the ability to easily import their existing SPICE libraries or circuits and simulate them in the PSIM environment. Additional features of the PSIM SPICE engine include the ability to support most LTspice netlists and the majority of PSIM library elements, including s-domain control blocks.

"The SPICE engine is a significant addition to PSIM's toolset. While PSIM praises itself for expert solution in system and control simulation, SPICE is well recognized in the industry for detailed device models," says Dr. Hua Jin, president of Powersim. "The new platform allows users to switch between PSIM and SPICE simulations effortlessly with minimum or no changes to a circuit. By combining the two powerful simulation engines, we are providing users with the best of both worlds".

PSIM and SPICE are complementary simulation engines with different strengths. Jin elaborates on the similarities and distinctions between the two simulators.

"While PSIM and SPICE are similar in the underlining numerical algorithm (they all use nodal analysis), PSIM was designed specifically for power electronics applications while SPICE was for general electronic circuits, "says Jin. "Because of its focus, PSIM can handle power electronic circuits, power converter systems, and controls very efficiently with fast speed and virtually no convergence problem. PSIM's component models are mostly behavior models, although we do offer more detailed models for RLC, PWM ICs, and MOSFET and diode to simulate the switching transients and diode reverse recovery."

Jin also notes how PSIM offers control over the level of complexity in simulations.

"A big difference between PSIM and SPICE is that complexity is not forced into a simulation with PSIM. With the multi-level model approach in PSIM, users can decide the level of details and complexity of a circuit depending on the simulation objective," says Jin.

According to the vendor, this combination of simulation engines represents an industry first among commercially available toolsets for power electronics, motor drives, and converter systems and control."

"PSIM is ideally suited for simulations that can be painful to perform with a SPICE-based solver. On the other hand, users are still interested in using industry-standard SPICE models to understand the high-frequency interactions between their components. With the PSIM/SPICE integration, users will be able to perform both detailed device simulation as well as system and control simulation in one integral environment with no or minimum change in the schematic," says Jin.

"For example, users will not have to redraw their schematic in another package in order to study the needed gate drive or turn-on/turn-off transients after they design their control with PSIM. The combination of one schematic with two solver choices is unique in the industry, and will lead to much faster design cycles and more efficient simulation process."

This new release of PSIM is the first on this design path and will have many additional functionality improvements coming up. According to the vendor, the new PSIM/SPICE platform will provide the most complete and integrated simulation environment available today in terms of speed, workflow, and overall design path for both switch-mode power supplies and motor drives as well as other applications.

Powersim will exhibit at the upcoming IEEE-ECCE Conference in Milwaukee, WI on September 18-22, 2016 at booth #504. They will also be facilitating their own PSIM training session at the conference on Sept 20th. To register & for more information, you can visit them at https://powersimtech.com/news-events/event/join-us-psim-training-session-sept-20-2016-milwaukee-wi/.

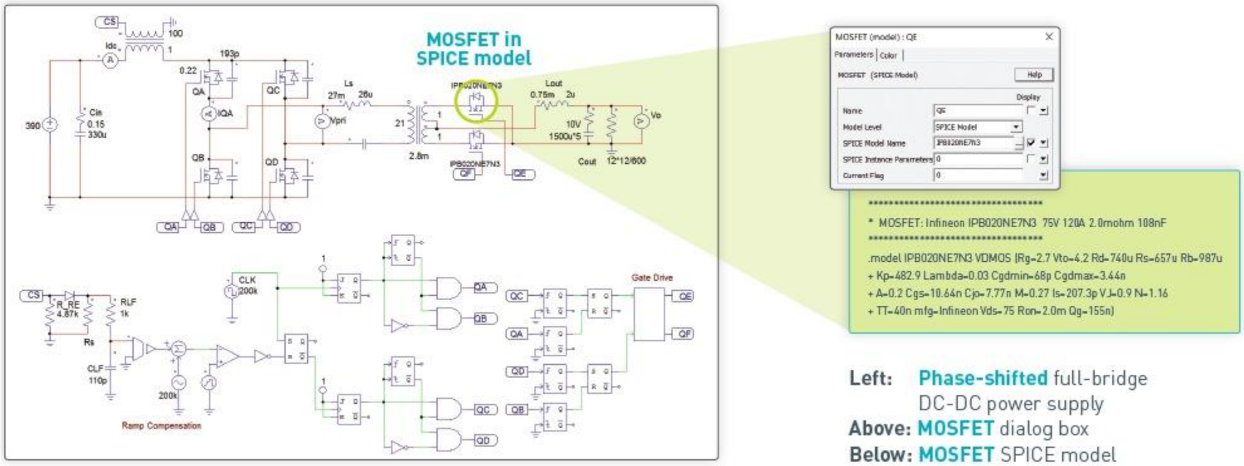

Figure. The PSIM simulation platform, which is optimized for power supply topology verification and control design, now includes a SPICE simulation engine. With the PSIM/SPICE integration, users will be able to perform both detailed device simulation as well as system and control simulation in one integral environment with no or minimum change in the schematic.

.