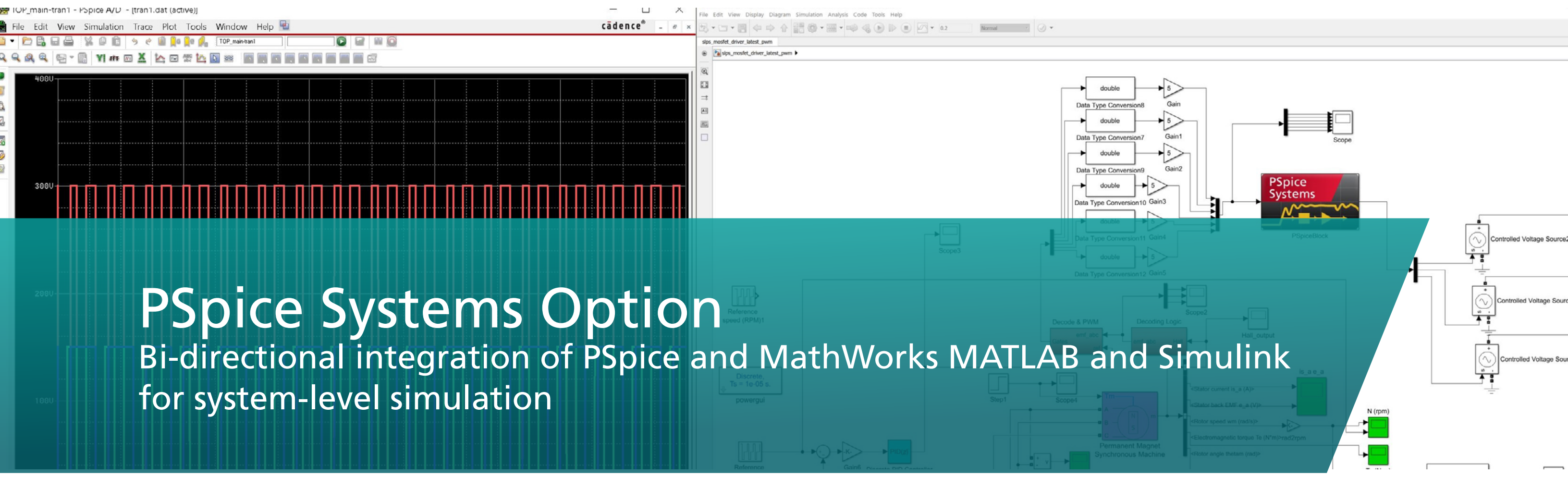

PSpice Systems Option

Bi-directional integration of PSpice and MathWorks MATLAB and Simulink for system-level simulation

Overview

A system-oriented design approach is a compelling design need in today's highly competitive market. Designers utilize Cadence® PSpice® simulation solutions for accurate analog and mixed-signal simulations, supported by a wide range of real-life electronics component models. The integration of Cadence PSpice and MathWorks MATLAB and Simulink provides a complete system-level simulation solution for PCB design and implementation. Customers can now utilize PSpice for analog/mixed-signal simulation and perform MATLAB/Simulink behavioral-level modeling, analysis, and visualization in a single, integrated system design and debug environment, improving productivity and accelerating time to market. This approach increases productivity by significantly cutting down time-to-market, and improves product reliability by enabling early identification of design and integration problems. The PSpice Systems Option brings three best-in-class simulators together to provide unmatched capability to design and optimize diverse design types at the system level.

The Simulink-PSpice (PSpice Systems Option) interface integrates:

- PSpice simulation—Cadence PSpice is a full-featured analog simulator with support for digital elements to help solve virtually any design challenge
- MATLAB—MathWorks' language and environment for technical computing
- Simulink—Mathworks' platform for multi-domain simulation and model-based design of dynamic systems

System and Circuit-Level Co-Simulation

PSpice Systems Option combines these industry-leading simulation tools into a co-simulation environment. Electro-mechanical/hydraulic systems such as control blocks, sensors, power converters, and body electronics are designed using ideal mathematical models in Simulink, forming an executable system-level specification for the design of the actual electronics.

Highlights

- Simulate electrical circuits and mechanical/hydraulic/thermal blocks together
- Simulate with ideal models for faster simulation during proof of concept
- Simulate with actual electrical designs using PSpice component models
- Electrical simulations with PSpice models exhibit nonlinearities, delay, and other real-world effects
- Large library of electrical parts for PSpice simulation and mechanical models and pre-defined blocks for Simulink available
- Full access to PSpice environment for in-depth electrical design and debugging
- Full access to MATLAB for analyzing and visualizing data, developing graphical user interfaces, and creating model data and parameters

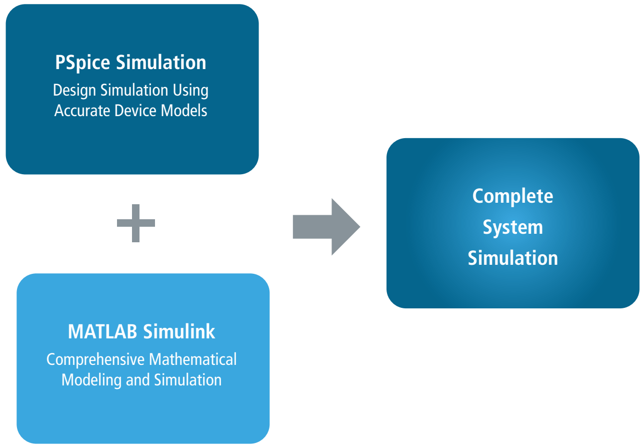

Figure 1: The PSpice Systems Option provides a complete system simulation solution.

PSpice A/D is then used to design the overall circuit based on this specification, providing simulation with more realistic models that exhibit nonlinearities, delay, and other real-life effects. Co-simulation allows system-level interfaces to be tested with actual electrical designs without the need to prototype the complete system. As a result, design problems are found much earlier, saving crucial time and money often spent in debugging trial boards within system prototype.

PSpice Systems Option Design Flow

In traditional design flow electronics and non-electronics module are designed separately and tested together at prototype stage only. This methodology requires designers to develop multiple prototypes, creating a longer design cycle. The PSpice Systems Option enables simulation between PSpice A/D and MATLAB/Simulink, allowing designers to simulate complete systems in a virtual prototype environment. In a typical flow, the system with its major blocks are designed and simulated in Simulink in their ideal form to verify the design architecture. This simulation drives exact specifications for the electronics system. The designer then switches to PSpice A/D to design the electronics circuit. Once the electronics block is designed and optimized as a standalone element, it can be integrated with the rest of the systems using the PSpice Systems Option for verification against the original system design. Engineers are able to validate the complete system design by leveraging co-simulation between Simulink and PSpice using the PSpice Systems Option.

To satisfy different design needs, there are four categories of interfaces supported by PSpice and MATLAB/Simulink integration:

Simulink/PSpice Interface

The Simulink/PSpice interface enables simulation between PSpice Designer and Simulink, allowing designers to simulate complete systems in a virtual prototype environment. It allows you to simulate with ideal models for faster simulation during proof of concept, or simulate with actual electrical designs without the need to prototype the entire system.

PSpice Device Model Interface

Cadence and MathWorks also enhance the integration by providing a bi-directional flow where the customer can import a Simulink model and co-simulate in PSpice. With PSpice Device Model Interface (DMI), which allows you to define models in C/C++ and simulate them in PSpice, you can import MATLAB software-generated code into PSpice as a DMI model and use it in a PSpice simulation.

PSpice/MATLAB Visualization Interface

The powerful waveform analysis capability of PSpice gets another boost by enabling simulation results exporting to MATLAB. PSpice users have complete and seamless access to MATLAB plotting capabilities, can view PSpice simulation results in MATLAB on click of a button, and customize waveform processing on export.

PSpice/MATLAB Functions Interface

EE designers always have a need for fast mathematical computation, and this need is felt even more when performing complex mathematical computations during the waveform analysis and behavioral modeling. The enhanced PSpice/MATLAB interface enables you to use MATLAB functions directly in measurement expression at waveform analysis stage and in behavioral modeling at circuit design stage within PSpice and Capture/DEHDL.

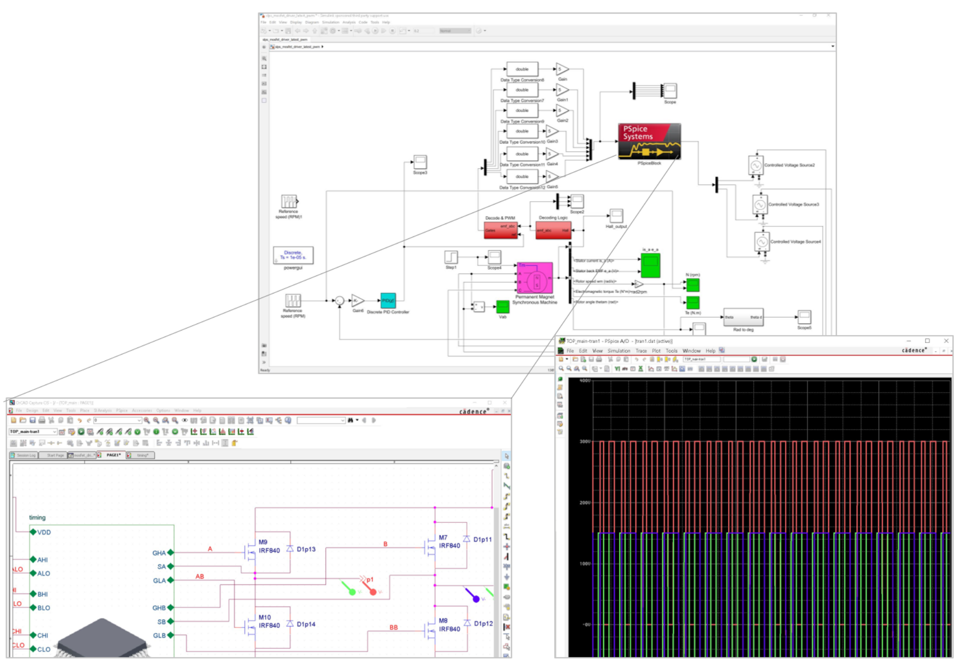

Figure 2: The PSpice Systems Option interface enables you to interface the PSpice circuit with Simulink and then observe the waveforms after Simulink-PSpice co-simulation.

Product and License Configuration

The MathWorks products

- MATLAB
- Simulink

Cadence products

- PSpice Systems Option
- OrCAD PSpice Designer or Allegro PSpice Simulator

Systems Requirements

Refer to Cadence PSpice and MATLAB system requirements.

Sales, Technical Support and Training

The OrCAD product line is owned by Cadence Design Systems, Inc., and is supported by a worldwide network of Cadence Channel Partners (VARs). For sales, technical support, or training, contact your local channel partner. For a complete list of authorized channel partners, visit www.orcad.com/CCP-Listing