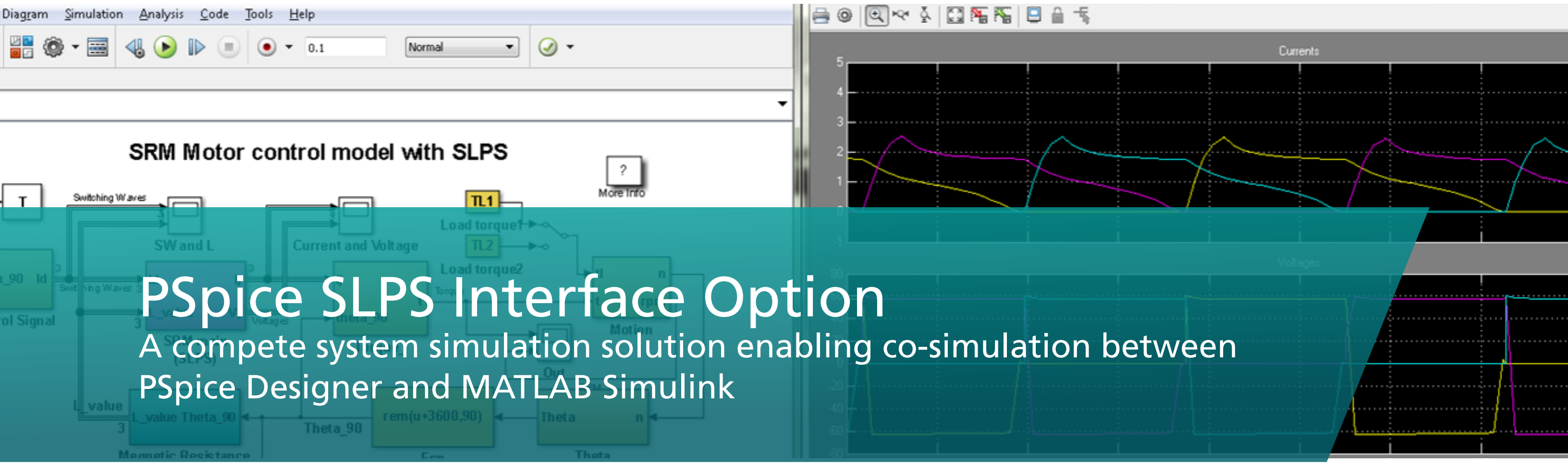
OrCAD™
CADENCE PCB SOLUTIONS



PSpice SLPS Interface Option

A compete system simulation solution enabling co-simulation between PSpice Designer and MATLAB Simulink

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 components models. The PSpice SLPS interface enables designers to perform the conceptual design and what-if analysis for a complete system, enabling co-simulation between PSpice Designer and MATLAB Simulink. 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 SLPS interface brings two best-in-class tools together to provide unmatched capability to design and optimize diverse design types at the system level.

The Simulink-PSpice (SLPS) interface integrates:

- PSpice simulation—Cadence PSpice Designer 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 SLPS integration combines two 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. PSpice Designer is then used to design the 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

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

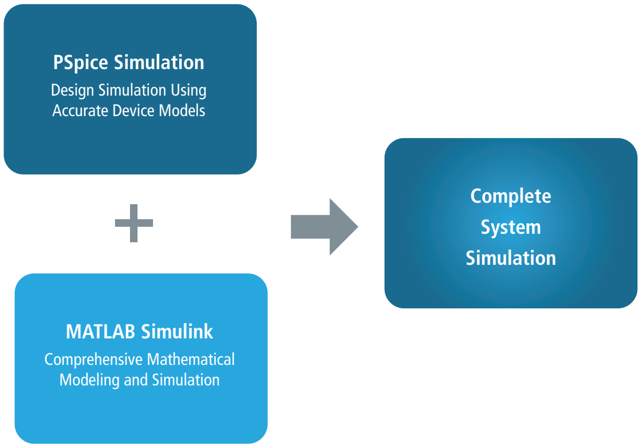


Figure 1: The PSpice SLPS interface provides a complete system simulation solution.

complete system. As a result, design problems are found much earlier, saving crucial time and money often spent in debugging trial boards within system prototypes.

PSpice SLPS Integration

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 SLPS interface enables simulation between PSpice Designer and 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 Designer 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 SLPS interface 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 SLPS interface.

Product and License Configuration

The Mathworks products

- MATLAB
- Simulink

Cadence OrCAD® products or Cadence Allegro® products

- PSpice SLPS license
- OrCAD Capture/CIS or Allegro Design Entry CIS license
- PSpice A/D or Allegro AMS Simulator license

System 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

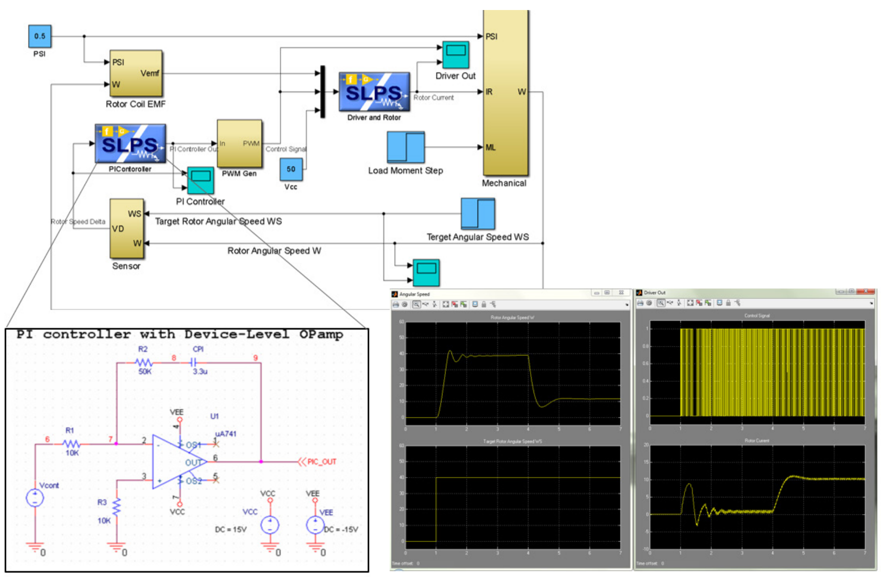


Figure 2: The PSpice SLPS interface enables you to interface the PSpice circuit with Simulink and then observe the waveforms after Simulink-PSpice co-simulation.