The Simulink-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. This forms an executable system-level specification for the design of the actual electronics. PSpice 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 then 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 prototypes.

**SYSTEM AND CIRCUIT LEVEL CO-SIMULATION**

The Simulink-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. This forms an executable system-level specification for the design of the actual electronics. PSpice 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 then 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 prototypes.

**INTEGRATION HIGHLIGHTS:**

- Simulate electrical, mechanical, and system-level systems
- 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 and mechanical models and pre-defined blocks for Simulink
- 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

**SIMULINK-PSPICE INTEGRATION REQUIREMENTS:**

- PSpice Simulator
- MATLAB and Simulink
- SLPS — the integration component

**PSpice integration with MATLAB/Simulink**

**ELECTRO-MECHANICAL SYSTEM SIMULATION WITH PSPICE ACCURACY FOR ACTUAL ELECTRONICS**

Designers utilize PSpice® simulation solutions for accurate analog and mixed-signal simulations supported by a wide range of board-level models. Simulink® is a platform for multi-domain simulation and model-based design of dynamic systems. Used together, they give designers the ability to perform system-level simulations that include realistic electrical models of actual components. Design and integration problems can now be found much earlier in the design process.

SLPS will allow you to interface the PSpice circuit with Simulink and then observe the waveforms after Simulink-PSpice co-simulation.
SLPS INTEGRATION

SLPS Integration allows an ideal block in Simulink to be replaced with a circuit design from PSpice, thus enabling co-simulation between PSpice and Simulink. In a typical flow, the system with its major blocks are designed and simulated in Simulink in their ideal form. This results in an executable or verifiable integration specification for the electrical block to be designed with PSpice. Once the block is designed and debugged as a standalone element, it can replace the ideal block in Simulink for verification against the original system design. Engineers are able to validate the system components due to co-simulation which enables system specifications to be passed between Simulink and PSpice.

SYSTEM REQUIREMENTS

- Pentium 4 (32-bit) equivalent or faster
- Minimum 256MB RAM (512MB recommended)
- 300MB swap space (or more)
- CD-ROM drive
- 32,768 color Windows display with minimum 1024 x 768 (1280 x 1024 recommended)

SALES, TECHNICAL SUPPORT, AND TRAINING

The OrCAD product line is owned by Cadence Design Systems, Inc. and supported by a worldwide network of value-added resellers (VARs) or distributors. For sales, technical support, and training, contact your local VAR. For a complete list of authorized VARs, visit www.orcad.com.

PRICING INFORMATION

For product pricing and availability, contact the OrCAD VAR nearest you. To locate a VAR in your area, visit www.orcad.com.

Simulink-PSpice integration design flow validates system interfaces with an electrical block prior to the prototype phase.