

PSPICE INTEGRATION WITH MATLAB/SIMULINK

PSpice® integration with The MathWorks’ MATLAB Simulink (SLPS) combines two industry-leading simulation tools in a co-simulation environment. PSpice is a simulation solution for analog and mixed-signal environments, and Simulink is a platform for multi-domain simulation and model-based design of dynamic systems. Used together, these two products give designers the ability to perform system-level simulations that include realistic electrical models of actual components. They allow engineers to discover design and integration problems much earlier in the design process, reducing the number of prototypes needed to execute the design.

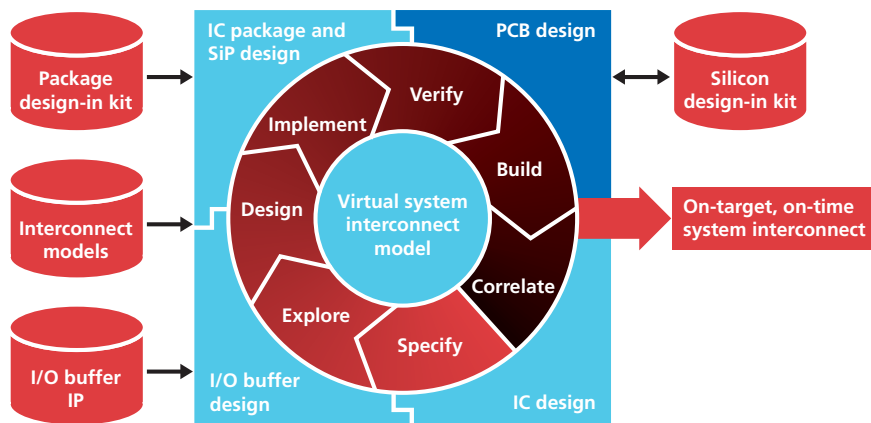


Figure 1: The affordable, high-performance OrCAD product line is easily scalable with the full complement of Cadence Allegro PCB design solutions

ORCAD PCB DESIGN TECHNOLOGIES

OrCAD products have a proven track record of innovation in the PCB personal productivity market. Available as stand-alone tools or in comprehensive suites, they allow designers to realize products from conception to manufacturing output. Easy-to-use and intuitive, they offer exceptional value. OrCAD technology also provides easy migration to the Cadence® Allegro® platform (see Figure 1).

SYSTEM-LEVEL SIMULATION WITH ACTUAL ELECTRONICS

SLPS integration enables designers of electro-mechanical systems—such as control blocks, sensors, and power converters—to perform integrated system and circuit simulation. Generally, system design and circuit design are separate processes that employ separate simulators. Because the simulators are not linked, there is no way for the electrical engineer to plug

the actual circuit data back into the system design. As a result, designers don't really know what influence the actual circuit module will have on the system and vice versa. In contrast, SLPS gives the designer the ability to perform system-level simulations that include realistic electrical PSpice models of actual components.

BENEFITS

- Allows system-level interfaces to be tested with actual electrical designs
- Reduces prototype re-designs and re-spins
- Includes a large library of electrical parts for PSpice and mechanical models and pre-defined blocks for Simulink
- Allows full access to PSpice for in-depth electrical design and debugging
- Allows full access to MATLAB for analyzing and visualizing data, and creating model data and parameters

DESIGN METHODOLOGY USING SLPS INTEGRATION

In the SLPS flow (see Figure 2), the system with its major blocks is designed and simulated in Simulink in ideal mathematical form, resulting in an executable system-level specification for design of the actual electronics. PSpice is then used to design a circuit module based on this data. 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. The PSpice model is a more realistic model than the black box in the Simulink design as it exhibits nonlinearities, delay, and other real-life effects (see Figure 3). As a result, design problems are found much earlier, saving crucial time and money often spent in debugging trial boards within system prototypes.

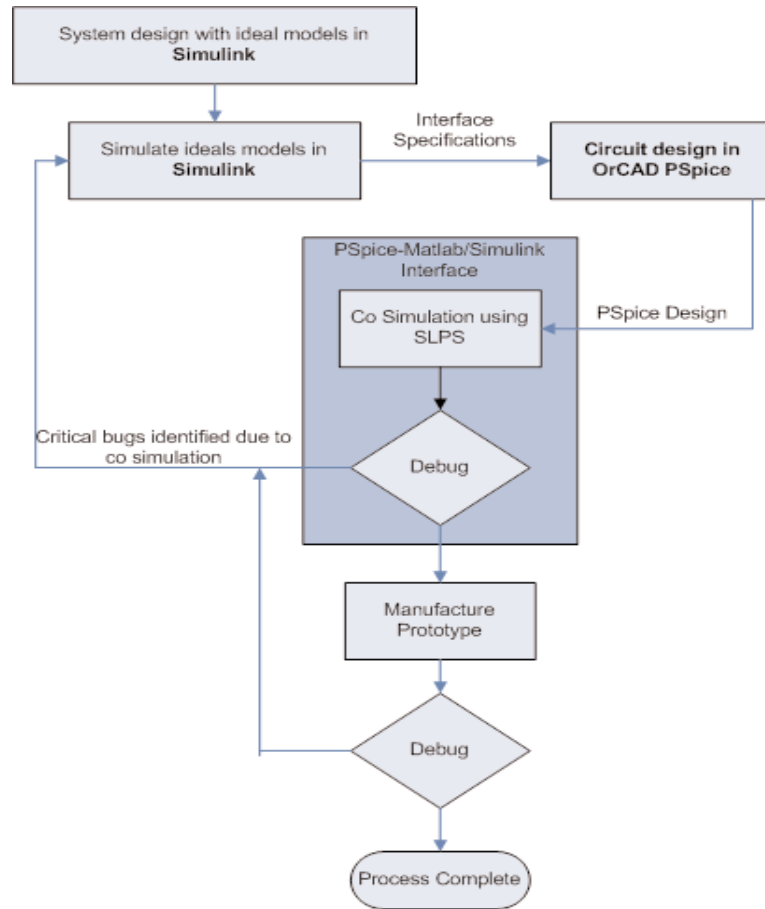


Figure 2: Simulink-PSpice integration design flow validates system interfaces with an electrical block prior to the prototype phase

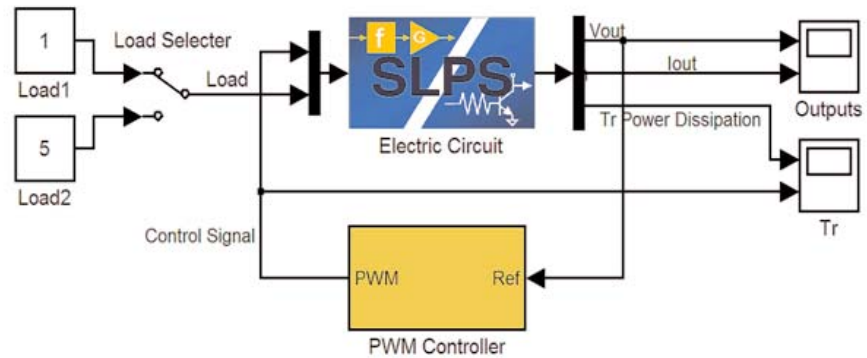


Figure 3: SLPS integration allows designers to interface the PSpice circuit with Simulink and then observe the waveforms after Simulink-PSpice co-simulation

SYSTEM REQUIREMENTS

- Pentium 4 (32-bit) equivalent or faster
- Windows XP Professional, Windows XP Home Edition, Windows 2000 (SP4), or Windows Server 2003
- 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 Cadence Channel Partners. For sales, technical support, or training, contact your local Cadence Channel Partner. For a complete list of authorized Cadence Channel Partners, visit http://www.cadence.com/partners/channel_partner/index.aspx.

PRICING INFORMATION

For product pricing and availability, contact the Cadence Channel Partner nearest you. For a complete list of authorized Cadence Channel Partners, visit http://www.cadence.com/partners/channel_partner/index.aspx.