CADENCE PSPICE A/D & PSPICE ADVANCED ANALYSIS
ADVANCED CIRCUIT SIMULATION AND ANALYSIS FOR ANALOG AND MIXED-SIGNAL CIRCUITS

Cadence® PSpice® simulation technology combines industry-leading, native analog and mixed-signal engines to deliver a complete circuit simulation and verification solution. It meets the changing simulation needs of designers as they progress through the design cycle, from circuit exploration to design development and verification. Designed for use in conjunction with PSpice A/D, PSpice Advanced Analysis helps designers improve yield and reliability.

Many electrical engineers rely on setting up their circuits on a bread board to verify operation and function of their designs. While this technique is a perfectly valid method for proof of concept, it has inherent limitations. The length of time it takes an engineer to collect parts, wire them up on a bread board, come up with a stimulus, and then manually measure the results can be significantly reduced by using PSpice A/D and Advanced Analysis to simulate design circuits. Also, endless component variations and combinations along with circuit stimulus make large-scale repetition of any manual process time and cost prohibitive.

Manufacturing variations of the electronic components that make up the circuit can affect the function and performance of the circuit. These manufacturing variations are generally expressed as tolerances on the key specifications of the components. It is not usually possible or practical to test a circuit to the complete range of tolerances of all the components. The tolerances of the components can also affect some of the circuit properties such that the operating limits of some components could be exceeded. Exceeding the operating limits can result in damage to a component, causing the circuit to cease to function.
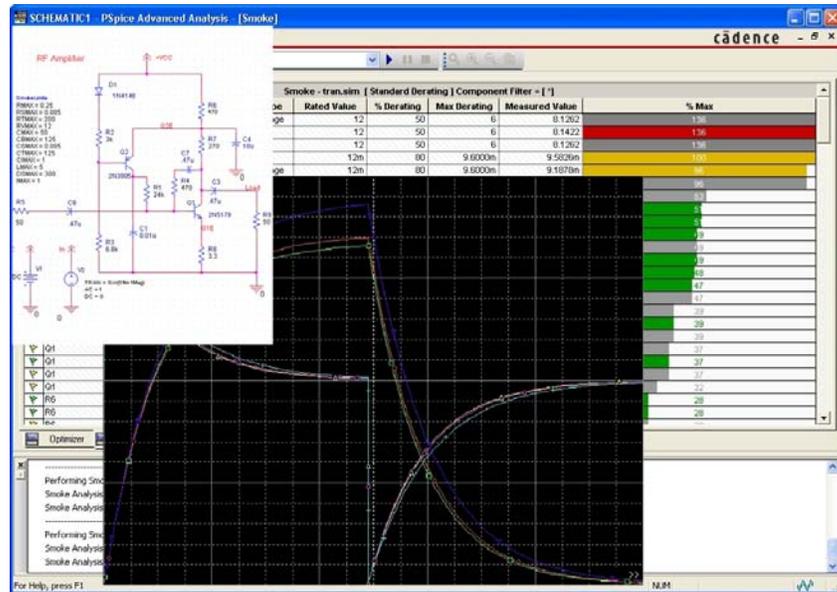
cādence®

Figure 1: Cadence PSpice provides a complete circuit simulation and analysis environment

Dealing with the implications of these issues can be expensive for a manufacturer. When a product is released without a thorough tolerance analysis the manufacturer doesn't really know how the circuit is going to function when it is produced in large volumes. The result can be that some percentage of products does not function as expected. In many cases, a test procedure is performed before the product is boxed and shipped. This test procedure sorts out defective products, which are then either repaired and retested or scrapped.

Many manufacturers consider test, rework and scrap a necessary part of doing business, but a robust design allows a manufacturer to reduce rework and scrap. A robust design can be produced by taking component tolerances into account and designing such that the circuit functions properly throughout the range of all the possible combinations of component values. An engineer can make a circuit more robust by specifying tighter tolerances on all the components, but tighter tolerances mean higher priced components. Over-specifying a design increases the cost to produce a product, and reduces the profit a company can realize.

Some issues, however, slip past final test and as a result defective products are shipped to customers. Manufacturers then need to deal with returns from the field, sending technicians to the customer to troubleshoot and repair, or conduct recalls. Field failures are the most expensive problems a manufacturer has to deal with, and can also result in safety hazards, injuries, lawsuits, class actions, judgments and fines, to say nothing of the damage to a company's reputation.

Simulation with PSpice provides a proven, quick, and easy way to perform circuit calculations an electrical engineer would otherwise perform on paper providing a significant productivity enhancement. With this powerful environment and capability, an engineer can be confident that circuits function as intended and the tolerances specified are correct—not so tight as to be overly costly, not so loose as to be producing inordinate numbers of defective products. This leads to increased manufacturing yield, fewer prototypes, less time spent in the lab, and ultimately a reduced cost of the product; thereby increasing the potential profit.

PSpice Simulation Technology is an advanced, industry-proven, mixed-signal simulator for electrical engineers. With widely available models, it is capable of simulating designs from power supplies to high-frequency systems to simpler IC designs. It enables engineer to:

• Understand and explore circuit performance and functional relationships with "what if" scenarios and design analysis

- Simulate complex mixed-signal designs, containing both analog and digital parts to supporting models like IGBTs, pulse width modulators, DACs and ADCs

It also simplifies viewing of simulation results—both analog and digital—by having a single display for the mixed-signal analysis results while retaining the same time axis. PSpice simulation technology is easy to use and highly integrated with one of the industry's most widely used schematic capture tools: Cadence OrCAD® Capture.

Advanced Analysis combines Sensitivity, Monte Carlo, Smoke (stress) analysis, and an Optimizer to provide an expended environment to take design analysis beyond simulation. Used in conjunction with core PSpice simulation Advanced Analysis maximizes design performance, yield, cost-effectiveness, and reliability.

BENEFITS

- Fast, accurate simulation technology and widespread model availability saves time, improves reliability, and aids convergence on larger designs
- MATLAB Simulink interface allows system-level interfaces to be tested with actual electrical designs emulating real-world applications
- Determining which components are over-stressed using Smoke analysis or observing component yields using Monte Carlo analysis helps prevent board failures
- Availability of resources such as multi-vendor models, built-in mathematical func tions, and behavioral modeling techniques enable an efficient simulation process
- Magnetic Parts Editor saves time and reduces errors by automating the design of inductors and transformers
- Single-button simulation, cross-probing, and full integration with OrCAD Capture improves productivity and data integrity
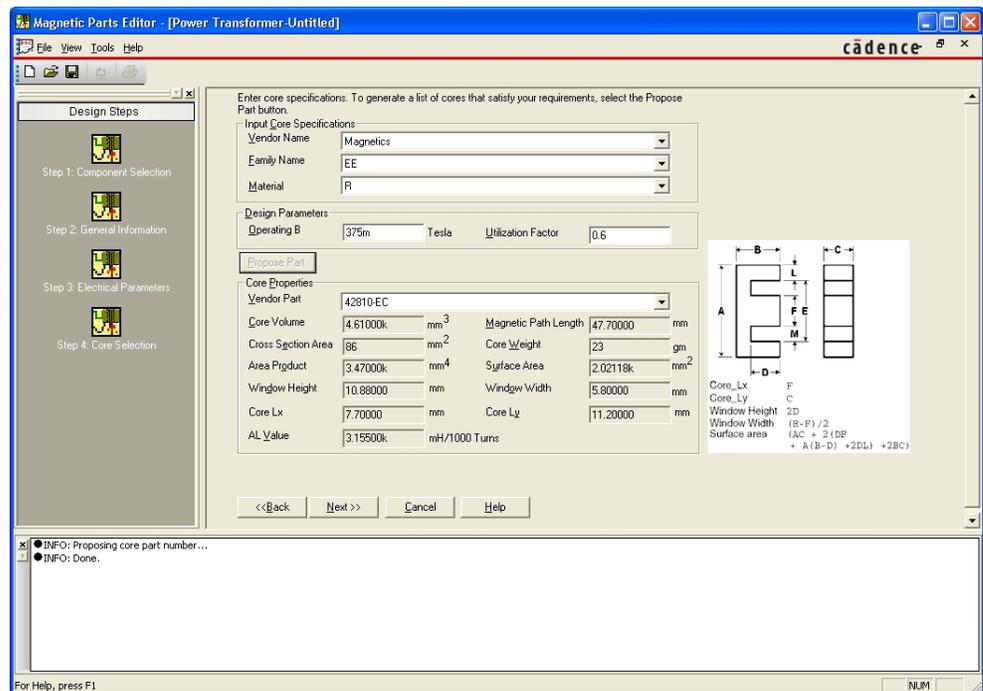
Figure 2: Magnetic Parts Editor automates the process of designing magnetic transformers / DC inductors and generating simulation models

PSPICE PRODUCT / TECHNICAL FEATURES

DESIGN ENTRY AND EDITING

- Use the advanced capabilities of OrCAD Capture schematic or Capture CIS, the world's most popular schematic entry system, to enter your designs
- Select from a library of over 20,000 parts for simulation, or choose from the large library of parts within OrCAD Capture for general schematic entry
- Easily import existing PSpice designs, created from Schematics, into the OrCAD Capture / PSpice environment
- Navigate through complex designs quickly with the hierarchical browser
- Create hierarchical block diagrams with automatic pin placement on hierarchical blocks
- Connect analog and digital components to reflect physical connections. The simulator automatically manages the transitions between analog and digital domains
- Cadence Allegro® Design Entry HDL is also fully integrated with PSpice, including one-button simulation and cross-probing

STIMULUS CREATION

- Invoke the interactive, graphical PSpice Stimulus Editor from within OrCAD Capture to define and preview stimulus characteristics
- Access built-in functions that can be described parametrically, or draw piecewise linear (PWL) signals freehand with the mouse to create any shape stimulus
- Create digital stimuli for signals, clocks, and buses; click-and-drag to introduce and move transitions

ORCAD CAPTURE / PSPICE INTEGRATION

- Set up and run simulations, and cross-probe simulation results from OrCAD Capture
- Use the hierarchical netlister with parametric sub-circuits for faster netlisting of complex hierarchical designs
- Expanded simulations can be run in the background while design editing continues
- Create multiple simulation profiles and save them in the OrCAD Capture Project Manager, allowing previous simulations to be recalled and run
- View simulation bias results directly on the schematic, including node voltages, pin and sub-circuit currents, and device power calculations

SIMULATION CONTROL

- Perform and monitor simulations, view simulation messages and graphical results, view and edit text files — all from a unified simulation environment
- Utilize analog analysis capabilities such as user-defined accuracy, automatic time-step control, and proprietary convergence algorithms to control the simulation process
- Interactively trade off accuracy and simulation time by loosening tolerances and time steps during non-critical periods of transient analyses, or by extending a transient analysis beyond pre-specified end time
- Preempt the current simulation to immediately run another one, then return to complete the preempted simulation later; control the queue of simulations waiting to be performed

MIXED ANALOG / DIGITAL SIMULATION

- PSpice automatically recognizes A-to-D and D-to-A connections, and properly processes them by inserting interface sub-circuits and power supplies
- Integrated analog and event-driven digital simulation engines improve simulation speed without loss of accuracy
- Single graphical waveform analyzer displays mixed analog and digital simulation results on the same time axis
- Digital functions support five logic levels and 64 strengths, load-dependent delays, and hazard / race checking

ANALOG ANALYSIS

- Explore circuit behavior using basic DC, AC, noise, and transient analysis
- View node voltages, pin currents, and power consumption or noise contributions of individual devices
- Include specific local temperature effects on individual devices for more accurate analysis
- Show circuit behavior variations as components change, via parametric, Monte Carlo, and worst-case analysis

DEBUGGING AND CONVERGENCE

- CHECKPOINT RESTART: Store simulation states at various time-points and then restart simulations from any of the simulation state. This saves time. Designer can modify simulation settings and design parameters before starting a simulation from pre-recorded time-state.
- AUTO-CONVERGENCE: Automatically change tolerances limits of convergence to make the design converge. Designer can use this option to get convergence and then fine tune simulations by further modifying simulator options. This option is recommended for Power Electronic Designs.
- ASSERTIONS: Use special parts to insert error and warning conditions for simulator to flag the conditions as the simulation progresses.
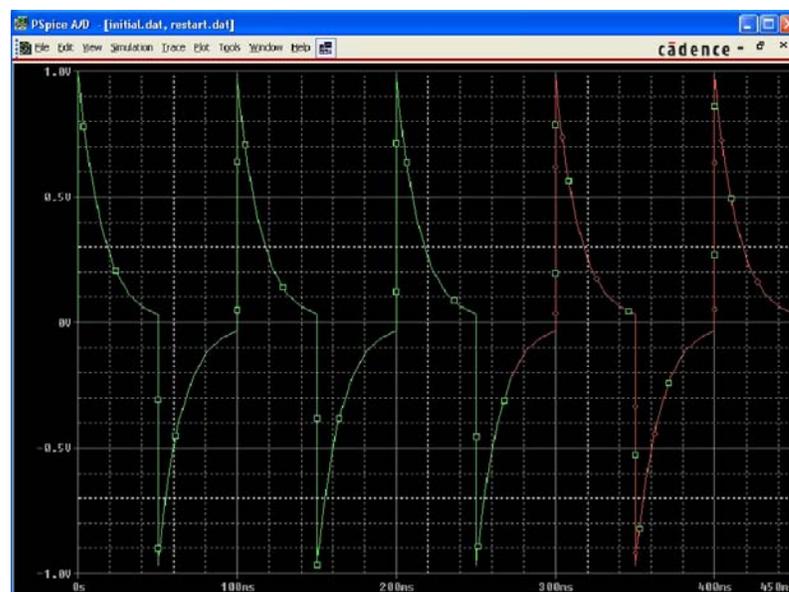
Figure 3: Checkpoint Restart can be used to accelerate verification of late stabilizing circuits

PROBE WINDOWS

- Choose from an expanded set of mathematical functions to apply to simulation output variables
- View simulation results in multiple waveform windows
- Select waveforms by name or by marking a net, pin, or part in the schematic
- Utilize cross-probing markers once and they stay with the analysis. As you change and re-simulate the design, the marked waveforms appear after each simulation
- View continuous, real-time "marching waveforms" as simulation progresses
- Copy and paste high-resolution, scalable waveforms into other applications for producing documentation

- Create plot window templates and use them to easily plot complex functions of signals, just by placing markers on desired pins, nets, and parts in the schematic
- Measure performance characteristics of your circuit using built-in measurement functions or create your own measurements

DATA DISPLAY

- Plot both real and complex functions of circuit voltage, current, and power consumption including Bodé plots for gain and phase margin and derivatives for small-signal characteristics
- Display Fourier transforms of time domain signals or inverse Fourier transforms of frequency domain signals
- Vary component values over multiple runs and quickly view results as a family of waveforms with parametric, Monte Carlo, and worst-case analysis
- Plot waveform characteristics, such as rise time versus temperature or supply voltage, using parametric analysis
- Create histograms after Monte Carlo analysis to display the distribution of a characteristic, such as overshoot

ACCURATE INTERNAL MODELS

- Large variety of built-in models adds flexibility to simulations, most include temperature effects
- Shipped models include R, L, C, plus:
  - Built-in IGBTs
  - Seven MOSFET models, including industry standard BSIM4, BSIM3v3.2 and the new EKV 2.6 model
  - Five GaAsFET models, including Parker-Skellern and TriQuint TOM-3 models
  - Two Bipolar models, including Gummel Poon and Mextram
  - Nonlinear magnetic models complete with saturation and hysteresis
  - Transmission line models that incorporate delay, reflection, loss, dispersion, and crosstalk
  - Digital primitives, including bidirectional transfer gates with analog I/O models

- Device Equations Developer's Kit (DEDK) allows implementation of new internal model equations which can be used with PSpice*

*DEDK is only available with a separate legal agreement through OrCAD Marketing. DEDK is intended for use by experienced device physicists and requires knowledge of C programming.

MODEL LIBRARY

- Select from more than 20,000 analog and mixed-signal models of industry devices
- More than 4,500 parameterized models for BJTs, JFETs, MOSFETs, IGBTs, SCRs, magnetic cores and toroids, power diodes and bridges, operational amplifiers, optocouplers, regulators, PWM controllers, Multipliers, timers, and sample-and-holds. These models allow passing simulation parameters as properties from the Schematic Editor
- Access basic components plus a variety of macro-models for more complex devices, including operational amplifiers, comparators, regulators, optocouplers, ADCs, and DACs
- Use state space average models to do fast feasibility simulations and control loop analysis for Switched mode power supplies

SYMBOLS FROM MODELS:

- Automatically generate OrCAD Capture parts for the models created by the Model Editor
- Automatically generate OrCAD Capture part libraries from simulation model libraries obtained from part vendors or colleagues
- Base the symbol generation on the PSpice symbol set, or your own
- Generate symbols for analog, digital, or mixed-signal devices (both primitives and macro-models)

PSPICE MODEL EDITOR

- Accessible from a part in OrCAD Capture, the intuitive user interface of the PSpice Model Editor can be used to view or edit its simulation model
- Encrypt model data to protect IP
- Add tolerance, distributions and maximum operating conditions (stress data) besides simulation data to the model for advanced analysis
- Extract a model of a supported device type by simply entering required data from the device datasheet
- Proceed quickly through the extraction process using fully interactive features as a guide. Device characteristic curves give you quick graphical feedback

BEHAVIORAL MODELING

- Describe functional blocks using mathematical expressions and functions
- Leverage a full set of mathematical operators, nonlinear functions, and filters
- Implement any transfer function via controlled voltage and current sources
- Define circuit behavior in the time or frequency domain, by formula (including Laplace transforms), or by look-up tables
- Select parameters which have been passed to sub-circuits in a hierarchy and insert them into transfer functions
- Create Boolean expressions that reference internal states and pin-to-pin delays using digital behavioral modeling

ADVANCED ANALYSIS PRODUCT / TECHNICAL FEATURES

Advanced Analysis includes:

- **Sensitivity:** Identifies critical circuit components
- **Optimizer:** Optimizes key circuit components
- **Monte Carlo:** Analyzes statistical circuit behavior and yield
- **Smoke:** Detects component stress
- **Parametric Plotter:** Examine solution space through nested sweeps

SENSITIVITY

Sensitivity identifies which component parameters are critical to the design goals of circuit performance. It examines how much each component affects circuit behavior by itself and in comparison to the other components. It also varies all tolerances to create worst-case (minimum and maximum) values. This information can be used to evaluate yield versus cost trade-offs, thereby maximizing cost-effectiveness.

Use Sensitivity for:

- Identifying the sensitive components within the circuit; then export the components to Optimizer to fine-tune the circuit behavior.
- Identifying which components affect yield the most; then tighten tolerances of sensitive components and loosen tolerances of non-sensitive components.

OPTIMIZER

Optimizer analyzes analog circuits and systems, fine-tuning designs faster than trial and error prototypes or bench testing. Optimizer helps find the best component values to meet design performance goals and constraints. Multiple goals and constraints can be analyzed to account for competing specifications.

Use Optimizer for:

- Improving design performance
- Updating designs to meet new specifications
- Optimizing behavioral models for top-down design and model generation

Optimizer includes four engines:

- Modified LSQ engine—Uses both constrained and unconstrained minimization algorithms to optimize goals subject to nonlinear constraints
- Random engine—Randomly picks values within a specified range and displays misfit error and parameter history
- Discrete engine—Used at the end of the optimization cycle to round off component values to match commercially available components

MONTE CARLO

Monte Carlo predicts the behavior of a circuit statistically when part values are varied within their tolerance range. Monte Carlo also calculates yield, which can be used for mass manufacturing predictions.

Use Monte Carlo for:

- Calculating yield based on your specifications
- Calculating statistical data

- Displaying results in a probability density histogram
- Displaying results in a cumulative distribution graph
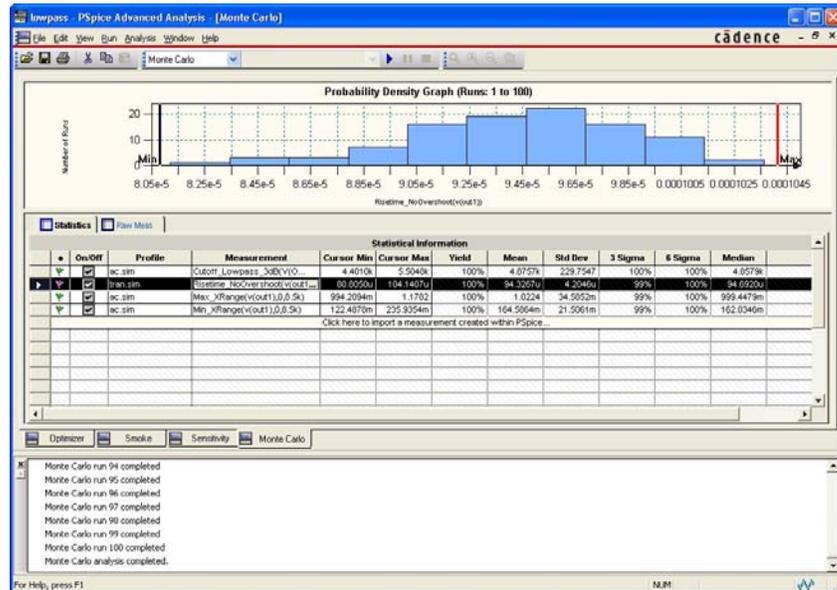
Figure 4: Monte Carlo predicts the behavior of a circuit statistically when part values are varied within their tolerance range

SMOKE

Smoke warns of component stress due to power dissipation, increases in junction temperature, secondary breakdowns, or violations of voltage/current limits. Over time, these stressed components cause circuit failure. Smoke compares circuit simulation results to the component's safe operating limits. If limits are exceeded, Smoke identifies the problem parameters. Devices can also be derated to meet design requirements Smoke can also display average, RMS, or peak values from simulation results and be used to compare these values against corresponding safe operating limits.
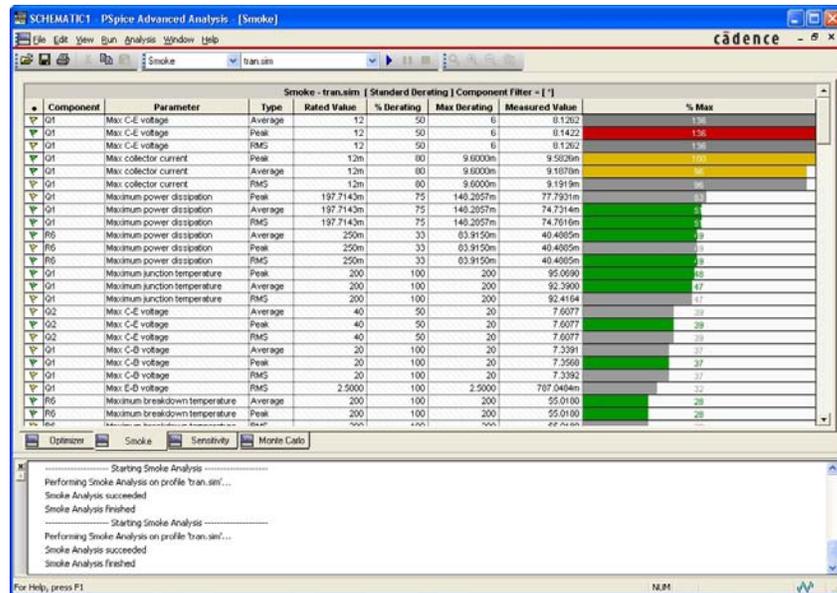
Figure 5: Smoke compares the simulated values with manufacturers' limits to highlight devices operating outside their safe operating range

Use Smoke to:

- Identify components exceeding manufacturers' limits
- Breakdown voltage across device terminals
- Maximum current limits
- Power dissipation for each component
- Secondary breakdown limits
- Junction temperatures

PARAMETRIC PLOTTER

The Parametric Plotter enables sweeping of multiple parameters once a simulated circuit has been created. It also provides an efficient way to analyze sweep results, sweep any number of design and model parameters (in any combinations), and view results in Parametric Plotter / Probe in tabular or plot form.

Use Parametric Plotter to:

- Sweep component values, model parameters and design parameters
- Select sweep-type: Discrete, Log or Linear
- Do nested sweeps to explore solution space for multiple measurements
- Examine solution space as tabular view and sort the data within already sorted columns.
- Create and Display multiple plots to examine measurements as function of various parameters

INTEGRATION WITH MATLAB / SIMULINK

PSpice integration with The MathWorks' MATLAB Simulink (SLPS) combines two industry-leading simulation tools in a co-simulation environment. 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.
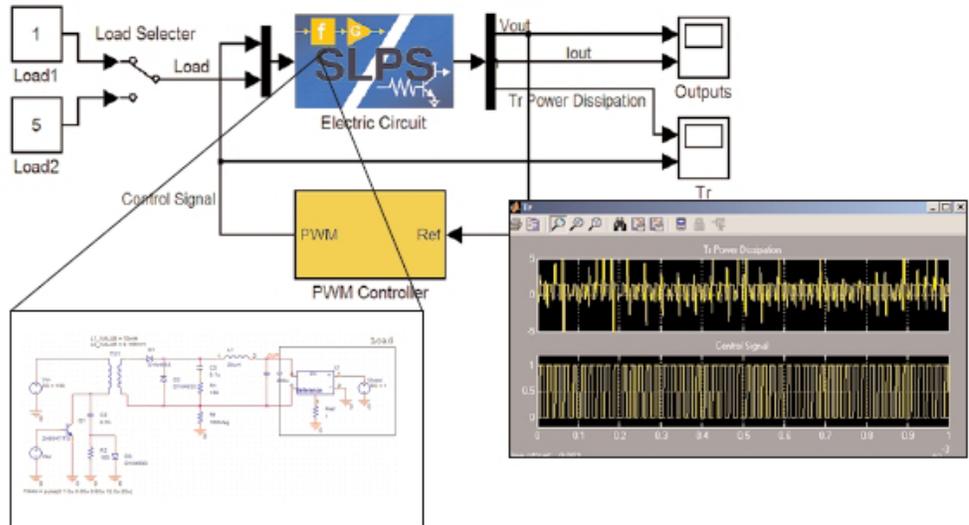
Figure 6: SLPS enables designers of electro-mechanical systems—such as control blocks, sensors, and power converters—to perform integrated system and circuit simulation

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 (VARs). For sales, technical support, or training, contact your local Cadence Channel Partner (VAR). For a complete list of authorized Cadence Channel Partners (VARs), visit www.cadence.com/Alliances/channel_partner.

cādence ®

Cadence Design Systems, Inc.

CORPORATE HEADQUARTERS
2655 Seely Avenue
San Jose, CA 95134
P: +1.800.746.6223 *(within US)*
    +1.408.943.1234 *(outside US)*
F: +1.408.943.5001
www.cadence.com

21202 01/10 KM/DM/PDF