

Cadence PSpice A/D Circuit Simulation

Advanced circuit simulation and analysis for analog and mixed-signal circuits

Cadence® PSpice® A/D 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. Used in conjunction with PSpice A/D, PSpice Advanced Analysis helps designers improve yield and reliability.

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 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 fully integrated with one of the industry's most widely used schematic capture tools: Cadence OrCAD® Capture.

With resources such as models from many vendors, built-in mathematical functions, and behavioral modeling techniques, PSpice A/D enables an efficient simulation process. Scalability options include advanced analysis simulation capabilities and integration with MathWorks MATLAB Simulink for co-simulation, simulation optimization, parasitic extraction, and re-simulation techniques.

PSpice Advanced Analysis takes design analysis beyond functional simulation. The Advanced Analysis Option brings together several technologies

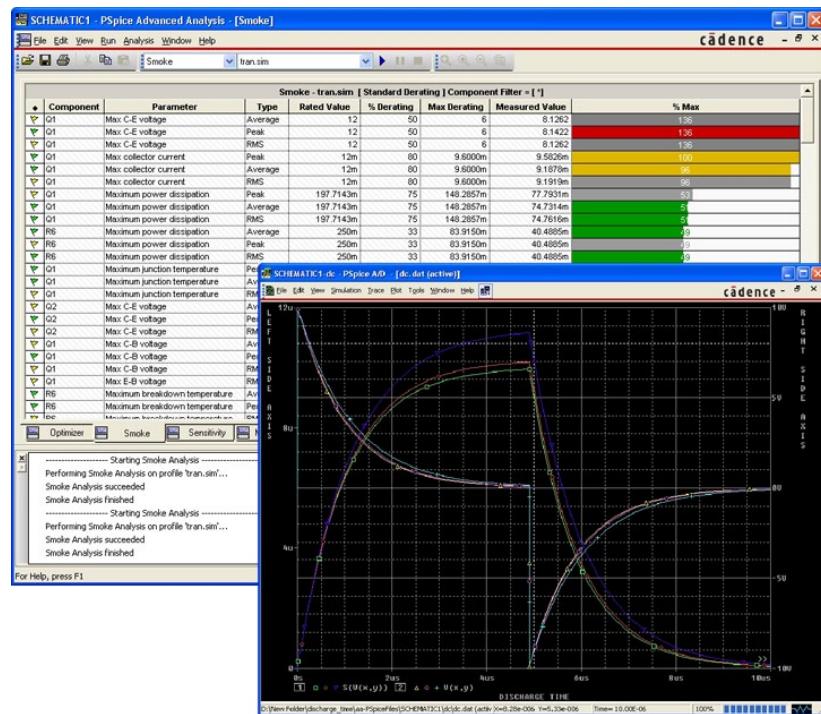


Figure 1: PSpice technology provides a complete simulation environment including simulation, waveform analysis with cross-probing, and bias results display on the schematic

that improve design performance, cost-effectiveness, and reliability. These technologies include Sensitivity,

Optimizer with multiple engines, Smoke (stress) analysis, and Monte Carlo (component yield) analysis.

Benefits

- New simulation performance technology 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
- Multi-vendor models, built-in mathematical functions, and behavioral modeling techniques enable an efficient simulation process
- Magnetic Parts Editor automates the design of inductors and transformers to reduce time and errors
- Single-button simulation, cross-probing, and full integration with OrCAD Capture improve productivity and data integrity

Features

PSpice simulation technology is capable of simulating a wide range of designs and design elements including power supplies, filters, amplifiers, digital decoding logic using gates, PWMs, transistors, MOSFETs, non-linear magnetic cores, varistors, zeners, and diodes. PSpice Probe 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.

With PSpice technology, engineers can analyze and explore circuit performance and functional relationships with “what if” scenarios and simulate complex mixed-signal designs. These designs can contain both analog and digital parts, supporting models like IGBTs, pulse width modulators, DACs, and ADCs.

Engineers can also design and simulate electro-mechanical, body-electronics, and hydraulic designs using behavioral models and integration with MATLAB’s Simulink product via SLPS (Simulink-PSpice) links.

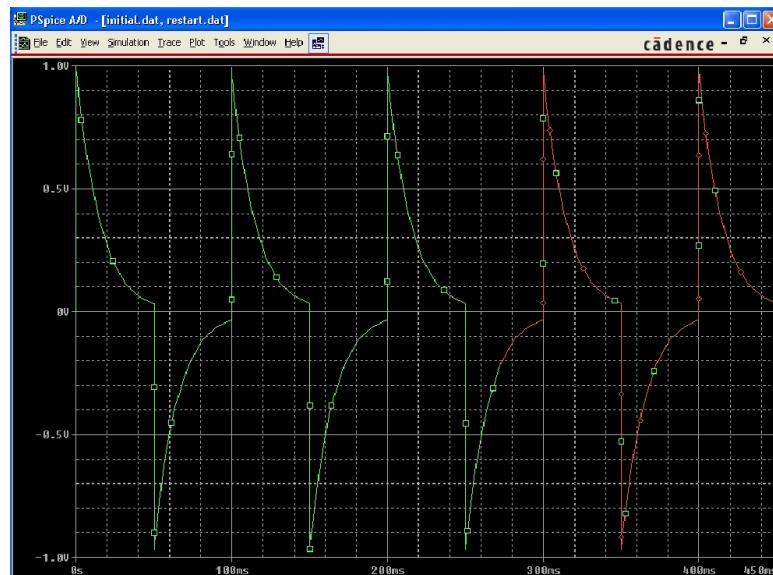


Figure 2: Checkpoint Restart accelerates verification of late stabilizing circuits

Simulation

PSpice technology ensures functional “correctness” of PCB designs by verifying the analog portions for node voltages, branch currents, and device power. Complex measurements can be created and viewed as the simulation progresses. Support is provided for multiple simulation profiles that enable the recall and execution of different simulations on the same schematic. Simulation bias results can be viewed directly on the schematic including node voltages, device power calculations, and pin/subcircuit current.

Mixed-signal designs can also be verified where the analog portions have digital content embedded along with setup/hold times. Integrated analog and event-driven digital simulations mean improved speed without loss of accuracy. Digital functions support 5 logic levels and 64 strengths, load-dependent delays, and hazard/race checking. PSpice simulation also features propagation modeling for digital gates and constraint checking (such as setup and hold timing violations).

Results and Data Display

PSpice Probe enables designers to make complex measurements, cross-probe, and view waveforms in multiple plots and allows the choice of an expanded set of mathematical functions to apply to simulation output variables. Designers can

create plot window templates and use them to easily make complex measurements by simply placing markers directly on the desired pins, nets, and parts in the schematic. Probe also enables the measurement of performance characteristics of a circuit using built-in functions and the creation of custom measurements. For data displays, additional capabilities allow plotting of 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.

Models

Along with numerous vendor models and model libraries available online, the PSpice model library offers more than 30,000 analog and mixed-signal models. This library includes 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. Also included are a large variety of accurate internal models—many of which include temperature effects—that add flexibility to simulations. A device equations developer’s kit (DEDK) allows implementation of new internal model equations.

Behavior modeling though functional blocks are described using mathematical expressions and functions, which allows designers to leverage a full set of mathematical operators, nonlinear functions, and filters. Circuit behavior can be defined in the time or frequency domain, by formula (including Laplace transforms) or by look-up tables, and error and warning messages can be specified in different conditions.

An integrated Model Editor provides an easy way to create models using device characteristic curves. An intuitive stimulus creation capability makes it easy to create a variety of simulation stimuli. Any shape stimulus can be created with built-in functions and can be described parametrically or freehand with the mouse to draw piece-wise linear (PWL) signals. Digital stimuli can be used for signals, clocks, and buses.

PSpice model IP can be protected using the encryption feature, which allows simulation models to be encrypted using the 56-bit DES algorithm. Finally, the Magnetic Parts Editor allows rapid prototyping of transformers, relays, and coils in an automated manner using electrical specifications.

Ease and Speed

The PSpice CheckPoint Restart feature enables designers to stop and restart, and to generate checkpoints at specified points in time of a simulation and then restart the simulation from a specific checkpoint. Thus, not only can simulations be restarted to account for any changes, but they can be run from only a range of interest instead of running the entire simulation again from the start.

With AutoConvergence, designers can suggest 'relaxed' values for specific parameters that can be used by the PSpice simulation engine to achieve convergence. If the simulation does not converge with predefined simulator options, the simulation engine will intelligently relax one or more parameters to achieve convergence.

Designers can add assertions to detect failure or warning conditions as the simulation progresses and, hence, need not wait for complete simulations to detect error conditions. Simulation profiles allow

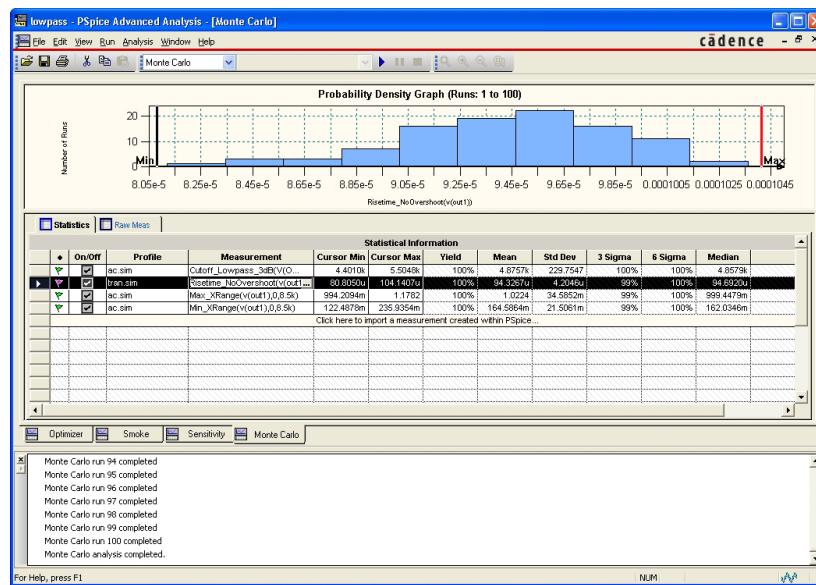


Figure 3: Monte Carlo analysis predicts the behavior of a circuit statistically when part values are varied within their tolerance range

binding of models and stimulus to enable simulation of different test conditions using same schematic, and users can also queue-up simulations for overnight results.

Front-End Integration

PSpice technology is fully integrated with Cadence OrCAD Capture CIS and Allegro® Design Entry-HDL, allowing for excellent cross-probing between the schematic and plot measurements.

Advanced Analysis

The PSpice Advanced Analysis Option is used in conjunction with PSpice A/D to improve design performance, yield, and reliability. Capabilities such as temperature and stress analysis, electro-mechanical simulation, worst-case analysis, Monte Carlo analysis, and automatic performance optimization algorithms improve design quality and maximize circuit performance automatically. PSpice A/D also allows for the design and generation of simulation models for transformers and DC inductors.

Sensitivity Analysis

Sensitivity analysis identifies which component parameters are critical to the circuit performance goals. It examines how much each component affects circuit behavior by itself and in comparison with

the other components by varying tolerances to create worst-case (minimum and maximum) values. Sensitivity analysis can also be used to identify which components affect yield the most; then tolerances of sensitive components can be tightened and tolerances of non-sensitive components loosened. This information can be used to evaluate yield versus cost tradeoffs.

Optimizer Analysis

Optimizer analyzes analog circuits and systems, fine-tuning designs faster than trial-and-error bench testing. It helps find the best component values to meet the performance goals and constraints. A specification can be as simple as an output voltage maximum; it can be a more complex output calculation like the cutoff frequency for a low-pass filter; or it can be an entire curve using the Optimizer curve-fitting capability.

Smoke Analysis

Smoke analysis 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 can cause circuit failure. Smoke compares circuit simulation results with the component's safe operating limits. If limits are exceeded, Smoke identifies the problem parameters. Smoke analysis can

be used for displaying average, RMS, or peak values from simulation results, and for comparing these values against corresponding safe operating limits.

Monte Carlo Analysis

Monte Carlo analysis predicts the behavior of a circuit statistically when part values are varied within their tolerance range. It also calculates yield, which can be used for mass-manufacturing predictions. With history stored in a separate file, model parameter values can be used for each Monte Carlo run and the values later reused.

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 combination), and view results in PPlot/ Probe in tabular or plot form.

OrCAD PCB Flow Integration

The PSpice simulator integrates seamlessly with OrCAD Capture, allowing you to use the same schematic for both simulation and PCB layout, which reduces rework and errors. The integration also allows for easy setup and simulation runs as well as cross-probing of simulation results. The hierarchical netlister with parametric subcircuits expedites the netlisting of complex hierarchical designs. The Magnetic Parts Editor helps designers build transformers for power supplies.

Integration with MATLAB Simulink

PSpice integration with 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

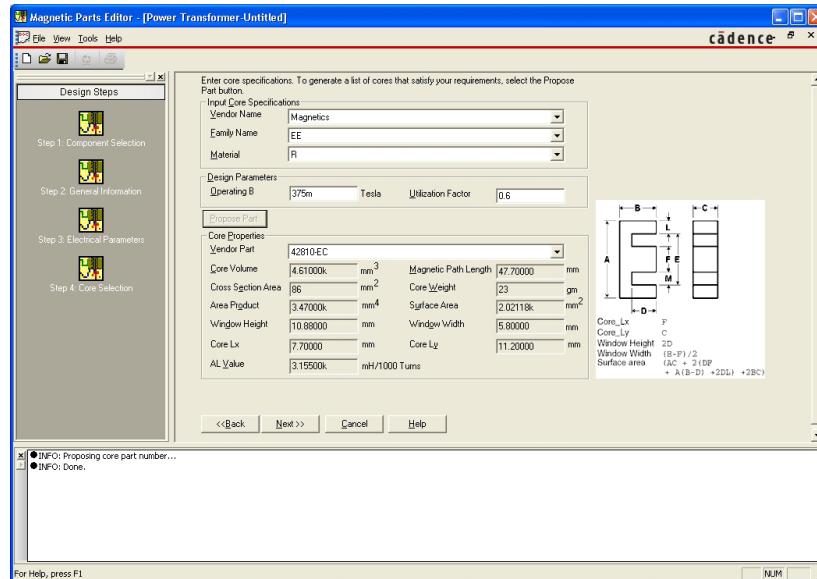


Figure 4: Magnetic Parts Editor automates the process of designing magnetic transformers and DC inductors for generating simulation models

perform integrated system and circuit simulations that include realistic electrical PSpice models of actual components.

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 channel partner. For a complete list of authorized channel partners, visit www.cadence.com/Alliances/channel_partner.



Cadence is transforming the global electronics industry through a vision called EDA360. With an application-driven approach to design, our software, hardware, IP, and services help customers realize silicon, SoCs, and complete systems efficiently and profitably. www.cadence.com