

CADENCE PSpICE CIRCUIT SIMULATION

To stay competitive in today's market, engineers must take a PCB design from engineering through manufacturing with shorter design cycles, tighter project goals and faster time to market. To be successful, designers need a set of powerful, intuitive, and integrated tools that work seamlessly across the entire PCB design flow.

Cadence PSpice simulation technology is available in the following products:

- Cadence PSpice A/D
- Cadence PSpice Advanced Analysis Option
- Cadence OrCAD EE Designer
- Cadence OrCAD EE Designer Plus
- Cadence OrCAD PCB Designer with PSpice

Cadence® OrCAD® personal productivity tools technologies (including Cadence PSpice® and OrCAD Capture) have a long history of addressing these requirements. Available as standalone products or in comprehensive suites, OrCAD technologies allow designers to take products from conception to final output. The powerful, tightly integrated PCB design technologies include design capture, librarian tools, PCB editing/routing, and analog/signal integrity simulators. Easy-to-use and intuitive, they offer exceptional value and future-proof scalability to the Cadence Allegro® series of PCB design products.

The availability of resources such as models from many vendors, built-in mathematical functions, and behavioral modeling techniques make for 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.

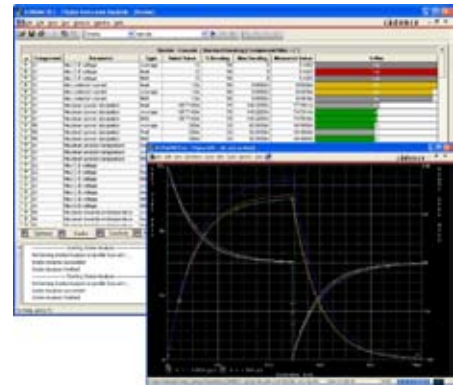


Figure 1: PSpice provides a complete simulation environment including simulation, waveform analysis with crossprobing, and bias results display on the schematic.

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

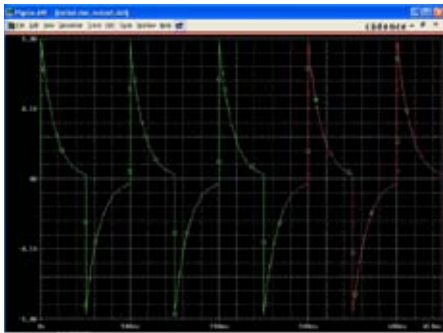


Figure 2: Use Checkpoint Restart to accelerate verification of late stabilizing circuits.

- 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 functions, 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

FEATURES

ADVANCED ANALYSIS

The PSpice Advanced Analysis Option is used in conjunction with PSpice A/D to improve design performance, yield,

and reliability. With Advanced Analysis capabilities, designers can automatically maximize the performance of circuits.

Capabilities such as temperature and stress analysis, electro-mechanical simulation, worst-case analysis, Monte Carlo, and automatic performance optimization algorithms improve the quality of designs and maximize circuit performance automatically. PSpice A/D also allows users to design and generate simulation models for transformers and DC inductors.

ORCAD PCB FLOW INTEGRATION

The PSpice simulator integrates seamlessly with OrCAD Capture, making it possible to use the same schematic for both simulation and PCB layout, which reduces rework and errors. The integration also allows for easy set up 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.



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

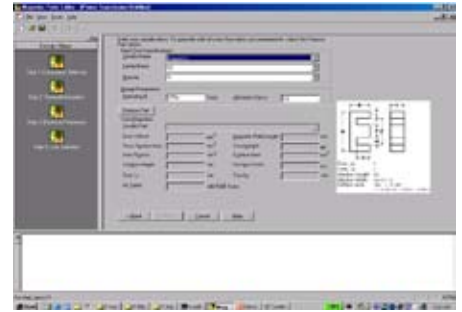


Figure 4: The Magnetic Parts Editor automates the process of designing magnetic transformers and DC inductors, and generating simulation models.

The OrCAD product line is supported by a worldwide network of Cadence Channel Partners. For sales, technical support, and training inquiries, please visit the global Cadence Channel Partner listing to find a partner in your region.

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 www.cadence.com/partners/channel_partner/index.aspx.

cadence™

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