OrCAD™

CADENCE PCB SOLUTIONS

OrCAD PSpice Designer

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

OrCAD® PSpice® Designer and OrCAD Capture combine to provide industry-leading, schematic entry, native analog and mixed-signal analysis engines to deliver a complete circuit simulation and verification solution. Whether you're prototyping simple circuits or designing complex systems, the OrCAD PSpice Designer product provides the best circuit-simulation technology to analyze and refine your circuits, components, and parameters before committing to layout and fabrication.

Overview

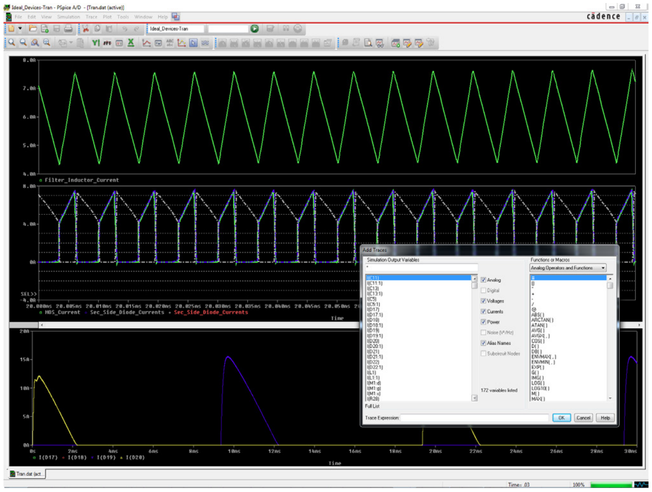
OrCAD PSpice Designer is a high-performance, industry-proven, mixed-signal simulator and waveform viewer for analog and mixed-signal circuits. As one of the most popular, general-purpose and mixed-mode circuit simulators with extensively available models from component and IC vendors, OrCAD PSpice simulation technology is applicable for product design in numerous industries such as aerospace, medical, power electronics, and automotive. It is also utilized extensively within the research community as a reference implementation. It is capable of simulating your designs from simple circuits, complex electronics, and power supplies to radio-frequency systems and targeted IC designs. With built-in mathematical functions, behavioral modeling, circuit optimization, and electromechanical co-simulation, the OrCAD PSpice environment goes far beyond general circuit simulation.

Included in the OrCAD PSpice Designer product with the OrCAD PSpice solution, OrCAD Capture provides fast, easy, and intuitive design entry, along with highly integrated flows supporting the engineering process. With an upgrade to the OrCAD PSpice Designer Plus product, advanced analysis simulation engines provide you with functional simulation to improve design performance, cost-effectiveness, and reliability.

In addition, integration with MathWorks MATLAB/Simulink technologies provide an analysis flow enabling multi-domain simulation, such as electromechanical co-simulation.

Highlights

- Extensive model library, model association and creation, multi-core support, and full integration with OrCAD Capture improve productivity and data integrity
- MATLAB/Simulink interface allows system-level interfaces to be tested with electrical designs emulating real-world applications
- Ability to determine which components are over-stressed using Smoke analysis or observing the affects of component variations on yield using Monte Carlo analysis helps prevent "in-field" failures
- Multi-vendor models, built-in mathematical functions, and behavioral modeling techniques enable highly tailored simulations
- Powerful waveform viewing and post-processing expression support speed review and analysis without having to rerun simulations
- Virtual prototyping leveraging GUI-based code generation of mixed-signal system models written in C/C++, SystemC®, and compact models from VerilogA using Automatic Device Model Synthesizer can be easily used in the PSpice environment
- Open architecture and program platform allows easy customization of algorithms and post-processing of results



The extensive capabilities of Probe enable complex measurements, multiple waveform plots, and an expansive set of mathematical functions

Simulation Features

Simulation

OrCAD PSpice simulation technology provides DC, AC, and transient analysis, so you can test the response of your circuits to varying inputs. It also provides digital worst-case timing analysis to help you find timing problems that occur with circuit signal transitions. Mixed-signal designs can also be verified where the analog portions have digital content embedded. Integrated analog and event-driven digital simulations mean improved speed without loss of accuracy, and complex measurements can be created and viewed as the simulation progresses.

Results and data display

The extensive capabilities of OrCAD PSpice Probe enable you to make complex measurements, cross-probe with the circuit design, view waveforms in multiple plots, and provide you with an expanded set of mathematical functions to apply to simulation output variables. OrCAD PSpice Probe also enables the measurement of performance characteristics of a circuit using built-in functions and the creation of custom measurements.

With OrCAD PSpice Probe, you can plot both real and complex functions of circuit voltage, current, and power consumption, including Bode plots for gain and phase margin and derivatives for small-signal characteristics. You can display Fourier transforms of time domain signals or inverse Fourier transforms of frequency domain signals. You can also vary component values over multiple runs and quickly view results as a family of waveforms with parametric, Monte Carlo, and worst-case analysis.

Models and modeling

Along with numerous vendor models and model libraries available online, the OrCAD PSpice model library offers more than 33,000 analog and mixed-signal models. This library includes parameterized models such as BJTs, JFETs, MOSFETs, IGBTs, SCRs, discretes, operational amplifiers, optocouplers, regulators, PWM controllers, and multipliers. A device equations developer's kit (DEDK) allows implementation of new and custom internal model equations.

You can describe behavior modeling through functional blocks using mathematical expressions and functions, which allow you 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.

The integrated OrCAD PSpice Model Editor provides you with 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 free-hand with the mouse to draw piece-wise linear (PWL) signals.

Flexibility and control

The OrCAD PSpice CheckPoint Restart feature provides greater control over your simulations. You can stop and restart, generate checkpoints at specified points in time of a simulation, and then restart the simulation from a specific checkpoint.

In addition, you can add assertions to detect failure or warning conditions as the simulation progresses, so you don't need to wait for complete simulation to detect error conditions. Simulation profiles allow binding of models and stimulus to enable simulation of different test conditions using same schematic, and you can also queue-up simulations for overnight results.

Stimulus editor

The OrCAD PSpice Stimulus Editor is an interactive, graphical environment to define and preview circuit stimulus characteristics. The use model allows access to built-in stimulus functions that can be described parametrically, or provides the ability to draw PWL signals freehand with the mouse to create any shape stimulus. You can create digital stimuli for signals, clocks, and buses, and then click-and-drag to introduce and move transitions.

Advanced magnetics

With PSpice Magnetic Designer, you get access to a database covering different types of core geometries, a wide variety of magnetic materials, standard wire gauge and non-linear saturable core SPICE models along with the customization option to add your own cores, wires, and material information. It is equipped with comprehensive magnetic design and modelling capabilities for designing PSpice magnetic components using commercially available cores, different types of wires like litz and foils for high-frequency switching applications, and different insulation materials to help facilitate export data for the manufacturers.

The modelling design capabilities help you generate accurate PSpice models including winding resistances and various parasitics. PSpice Magnetic Parts Editor gives you the ability to automate the design process for generating simulation models for variety of transformers and DC inductors, as well as build transformers using electrical specifications for power supplies by rapid prototyping.

Design Solutions and Flows

Capture front-end integration

OrCAD PSpice technology is seamlessly integrated with OrCAD Capture—one of the most widely used schematic design solutions—allowing you to easily cross-probe between the schematic design, simulation plot results and measurements. This integration also allows you to use the same schematic for both simulation exploration and PCB layout, reducing rework and errors. Even if you're not creating a circuit for use in the PCB flow, the integration allows for easy setup, model placement, circuit creation, and simulation.

Integration with MATLAB/Simulink

OrCAD PSpice integration with MATLAB/Simulink (SLPS) brings two industry-leading simulation tools in a co-simulation environment. SLPS integration enables designers of electromechanical systems—such as control blocks, motors, sensors, and power converters—to perform integrated system and circuit simulations that include realistic, electrical OrCAD PSpice models of physical components.

Advanced Analysis

OrCAD PSpice Advanced Analysis simulation is used to improve your design's performance, yield, and reliability. Capabilities such as temperature and stress analysis, worst-case analysis, Monte Carlo analysis, and automatic performance-optimization algorithms improve design quality and maximize circuit performance.

Open Architecture Platform

Enabling an extensible and customizable design environment, OrCAD's open architecture platform incorporates a highly integrated Tcl/HTML5 programming infrastructure that allows the creation or enhancement of features, functionality, design capabilities, and flows. The Tcl programming interface provides programming access to the user interface, command structure, simulation data, and algorithm process. Custom features that do not exist natively can be created, further enhancing and extending the OrCAD PSpice environment.

For the latest product or release information, visit us at www.orcad.com or contact your local Cadence Channel Partner.

Sales, Technical Support, and Training

The OrCAD product line is owned by Cadence Design Systems, Inc., and is supported by a worldwide network of Cadence Channel Partners (CCPs). For sales, technical support, or training, contact your local channel partner. For a complete list of authorized channel partners, visit www.orcad.com/CCP-Listing.