

OrCAD PSpice Designer Plus

Optimize design performance, cost-effectiveness, and reliability

Upgrading to OrCAD® EE (PSpice®) Designer Plus provides the PSpice Advanced Analysis simulation engines that are used in conjunction with core PSpice simulation to maximize design performance, yield, cost-effectiveness, and reliability. These simulation capabilities—Sensitivity, Monte Carlo, Smoke (Stress), Optimizer, and Parametric Plotter—help you deal with the problems of manufacturing variations of electronic components by providing you with an in-depth understanding of circuit performance that goes beyond basic validation.

Overview

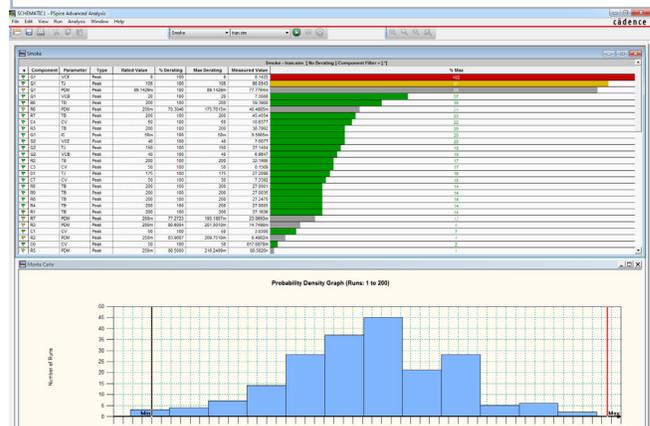
Manufacturing variations inherent in the electronic components that make up your circuits can greatly affect their function and performance. These manufacturing variations are generally expressed as tolerances on key specifications of the components and it is not usually possible or practical to manually test a circuit to the complete range of tolerances of all the components.

Without simulation, a common option is to simply specify tighter tolerances on all the components to try to account for the variations. But tighter device tolerances mean higher-priced components, increasing design costs. The other option is to hope for the best and assume that variations will not impact a circuit's function or performance to the point of a failure. While this assumption may save costs, it tends to result in lower yields.

A robust design can be produced by taking component tolerances into account such that the circuit functions properly throughout the range of all the possible combinations of component values. Simulation with PSpice Advanced Analysis provides a proven, quick, and easy way to perform circuit calculations to determine and account for the component variations. With this powerful environment and capability, you can be confident that circuits function as intended and the tolerances specified are in-line with your requirements: not so tight as to be overly costly and not so loose as to produce inordinate numbers of defective products.

Highlights

- Determining which components are over-stressed by using Smoke (Stress) analysis or observing the affects of component variations on yield by using Monte Carlo analysis helps to prevent “in-field” failures
- Identifying which components most affect yield allows you to tighten the tolerances of sensitive components and loosen the tolerances of non-sensitive components to optimize costs
- Determining the best component values to meet design performance goals and constraints helps to fine-tune designs faster than trial and error
- Simulating the behavior of a circuit statistically when part values are varied within their tolerance range helps to explore operating conditions



The PSpice Advanced Analysis simulation engines help maximize design performance, yield, cost-effectiveness, and reliability

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

Simulation Features

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.

Sensitivity analysis

Sensitivity analysis identifies which component parameters are critical to your circuit performance goals by asking such questions as, "Does the value of R1 affect my bandwidth more than the value of R2?" It examines how a component's inherent manufacturing variations affect circuit behavior both individually and in comparison with other components by varying manufacturing tolerances to create worst-case (minimum and maximum) results. Sensitivity analysis can also be used to identify which components affect yield the most, allowing you to choose the sensitive components with tighter tolerances. The analysis allows evaluation of yield versus cost tradeoffs.

Optimizer analysis

Optimizer analyzes analog circuits and systems, fine-tuning design parameters faster than trial-and-error to find the best component values to achieve your performance goals and constraints. Circuit specifications can be as simple as an output voltage maximum, a more complex output calculation such as the cutoff frequency for a low-pass filter, or an entire curve using the Optimizer curve-fitting capability.

Smoke (Stress) analysis

Smoke (Stress) analysis warns of component stress due to power dissipation, increases in junction temperature, secondary breakdowns, or violations of voltage and current limits. Over time, these stressed components can cause your designs to fail, often long after the design stage. Smoke (Stress) analysis compares circuit simulation results with the component's safe operating limits, and if the limits are exceeded, Smoke (Stress) analysis identifies the problem parameters.

Monte Carlo analysis

Monte Carlo analysis predicts the behavior of a circuit statistically when multiple components are varied within their tolerance ranges. By changing all the parts of your circuit randomly within their tolerances over a number of simulations, you can approximate the yield of building numerous boards. These measurements help you to determine what percentage of boards has the potential to be out of spec and affecting yield. If the percentage is too high, running Sensitivity analysis can help identify the component(s) that might need to be tightened up. For example, you can swap a 10% tolerance component with a 1% component to improve your yield, or conversely, if you have components that aren't too critical, you can use 10% or 20% parts to reduce cost.

Parametric Plotter

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

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 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 (VARs). 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.



www.orcad.com

