PSpice Advanced Analysis Options
AUTOMATICALLY FINE-TUNE CIRCUITS

Maximize the performance of your circuit automatically with the PSpice® Advanced Analysis suite of tools. Advanced Analysis is used in conjunction with PSpice A/D to improve design performance, yield, and reliability.

**THE ADVANCED ANALYSIS TOOL SUITE INCLUDES:**

- **Sensitivity:** Identifies critical circuit components
- **Optimizer:** Optimizes key circuit components
- **Smoke:** Detects component stress
- **Monte Carlo:** Analyzes statistical circuit behavior and yield
- **Parametric Plotter:** Provides an efficient way to sweep any number of design and model parameters (in any combinations) and view results in PPlot/Probe in tabular or plot form

**SENsitIVITY**

Sensitivity analysis identifies which component parameters are critical to the goals of your circuit performance. Sensitivity 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. Use Sensitivity to identify the sensitive components, and then export the components to Optimizer to fine-tune the circuit behavior. Also use Sensitivity to identify which components affect yield the most, and then tighten tolerances of sensitive components and loosen tolerances of non-sensitive components. With this information you can evaluate yield versus cost trade-offs. You can also control the range around a point to determine sensitivity (default is 40% tolerance).

**Use Sensitivity for:**

- Identifying critical performance parameters of components
- Examining how much components affect circuit behavior independent from, or dependent on, other components
- Varying component tolerances to determine worst-case component (minimum and maximum) values

**OPTIMIZER**

The Optimizer analyzes analog circuits and systems, fine-tuning your designs faster than trial-and-error bench testing can. It helps you find the best component values to meet performance goals and constraints; goals and constraints can be specified as either measurements or curves. You can specify multiple goals and constraints to handle competing specifications.

**Use Optimizer for:**

- Improving design performance
- Updating designs to meet new specifications
- Optimizing behavioral models for top-down design and model generation
- Tuning a circuit to match known results in the form of measurements or curves

**Optimizer Includes Four Engines**

- **Least Squares Quadratic (LSQ) engine:** Uses a gradient-based algorithm that optimizes a circuit by calculating sensitivities and adjusting parameter values to meet goals
- **Modified LSQ engine:** Uses both constrained and unconstrained minimization algorithms to optimize goals subject to nonlinear constraints

Sensitivity in PSpice Advanced Analysis calculates the sensitivity of circuit parameters across multiple measurements and estimates worst-case performance
• 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 in PSpice Advanced Analysis is showing the spread of measured specifications caused by device tolerances and uses these results to predict manufacturing yields

SMOKE (also available separately)
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 can 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. Use Smoke for displaying average, RMS, or peak values from simulation results and comparing these values against corresponding safe operating limits. You can also use Smoke for creating, modifying, and configuring derate files for use with Smoke Analysis.

Monte Carlo in PSpice Advanced Analysis is maximizing bandwidth within noise and voltage constraints by varying the components selected in the schematic

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

Smoke in PSpice Advanced Analysis 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

The Parametric Plotter added to Advanced Analysis provides the functionality of sweeping multiple parameters. Once you have created and simulated a circuit, you can use the Parametric Plotter to perform this analysis. The Parametric Plotter gives users the flexibility of sweeping multiple parameters. It also provides an efficient way to analyze sweep results. Using Parametric Plotter, you can sweep any number of design and model parameters (in any combinations) and view results in PPlot/Probe in tabular or plot form.

Use the Parametric Plotter for:
• Sweeping multiple parameters
• Allowing device/model parameters to be swept
• Displaying sweep results in spreadsheet format
• Plotting measurement results in Probe UI
• Posting analysis measurement evaluation

The Optimizer in PSpice Advanced Analysis is maximizing bandwidth within noise and voltage constraints by varying the components selected in the schematic

SYSTEM REQUIREMENTS
• Pentium 4 (32-bit) equivalent or faster
• Windows XP Professional, Windows XP Home Edition, or Windows 2000 (SP4)
• Minimum 256MB RAM (512MB recommended)
• 300MB swap space (or more)
• CD-ROM drive
• 32,768 color Windows display with minimum 1024 x 768 (1280 x 1024 recommended)

SALES, TECHNICAL SUPPORT, AND TRAINING
The OrCAD product line is owned by Cadence Design Systems, Inc. and supported by a worldwide network of value-added resellers (VARs) or distributors. For sales, technical support, and training, contact your local VAR. For a complete list of authorized VARs, visit www.orcad.com.

PRICING INFORMATION
For product pricing and availability, contact the OrCAD VAR nearest you. To locate a VAR in your area, visit www.orcad.com.