DATASHEET

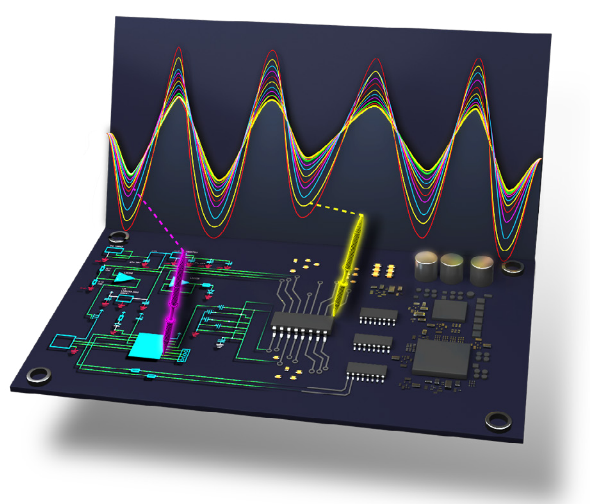
PSpice

Optimize Design Performance, Cost-Effectiveness, and Reliability

Cadence® PSpice® is the industry leading virtual simulation solution with native integration into OrCAD® X Capture. Its advanced modeling capabilities and extensive components libraries enable electrical engineers to accurately simulate analog and mixed-signal circuits under various operating conditions to validate circuit functionality and reliability. Engineers can predict and mitigate potential problems, such as stress factors or reliability issues, before physical prototyping, saving time and resources.

Moreover, the PSpice integration with MathWorks® MATLAB® and Simulink® enables co-simulation, allowing electrical engineers to perform conceptual design and what-if analysis for electro-mechanical systems such as control blocks, sensors, and power converters.

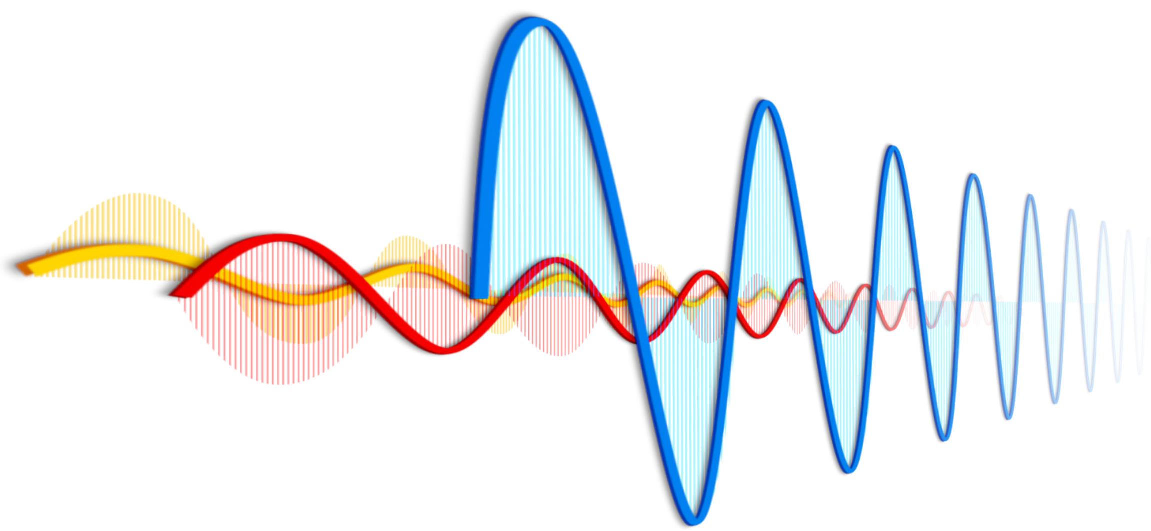
This approach increases productivity by significantly cutting down time-to-market and improves product reliability by helping identify design and integration problems earlier.



Benefits

The core benefits of PSpice are:

- **Circuit Analysis** - Run multiple analyses such as Transient, Bias Point, Worst-case, Monte Carlo, AC/DC Sweep, Temperature Sweep, and more to explore and evaluate your circuit's behavior.
- **Circuit Optimization** - Fine-tune design parameters to improve circuit performance, reliability, and yield with the following analyses: Monte Carlo, Optimizer, Parametric Plotter, Sensitivity, and Smoke (Electrical Over Stress - EOS).
- **Electro-Mechanical System Simulation** - Model, analyze, and optimize the performance and behavior of your electromechanical systems under various scenarios with co-simulation.
- **Extensive Component Library** - From discretes to MOSFETs, access, select, and simulate your circuits from a library of over 35,000 parameterized models.
- **PSpice Modeling Applications** - Easily create custom simulation-ready models with specific parameters for devices such as diodes, switches, transformers, and more.
- **Waveform Analysis** - Evaluate circuit waveform results with customizable plot windows, measurement functions, and vital trace data to verify circuit performance against design requirements.



Advanced Analysis Features

Monte Carlo Analysis - Predict the behavior of a circuit statistically when multiple components are varied within their tolerance ranges. By adjusting the tolerances of all the parts in your circuit over several simulations, you can approximate the yield of building many boards. These measurements help you determine what percentage of boards have the potential to be out of spec and affect yield. If the percentage is too high, running a 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 reduce cost by going with a 10% or 20% tolerance.

Optimizer Analysis - Analyze analog circuits and systems fine-tune 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.

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 the results in Plot or Probe in tabular or plot form.

Sensitivity Analysis - Identify 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. It can also 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.

Smoke Analysis – Electrical Over Stress (EOS) - Identify 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. Compare circuit simulation results with the component's safe operating limits, and if the limits are exceeded, Smoke (Stress) analysis identifies the problem parameters.

cādence®

Cadence is a pivotal leader in electronic design and computational expertise, using its Intelligent System Design strategy to turn design concepts into reality. Cadence customers are the world's most creative and innovative companies, delivering extraordinary electronic products from chips to boards to systems for the most dynamic market applications. **www.cadence.com**

© 2024 Cadence Design Systems, Inc. All rights reserved worldwide. Cadence, the Cadence logo, and the other Cadence marks found at www.cadence.com/go/trademarks are trademarks or registered trademarks of Cadence Design Systems, Inc. All other trademarks are the property of their respective owners.

04/24 CPG/PDF