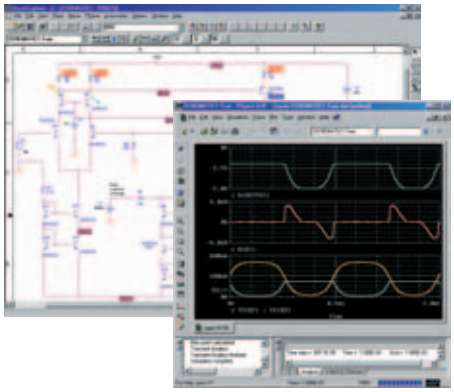
PSpice A/D

ADVANCED SIMULATION SOLUTIONS FOR ANALOG AND MIXED-SIGNAL ENVIRONMENTS

PSpice® simulation solutions are ideal for accurate analog and mixed-signal simulations and are supported by a wide range of board-level models. Since its introduction in 1985, PSpice has been continuously enhanced to support the latest hardware and operating systems. Each new release addresses current technological requirements and customer requests for enhancements.

KEY FEATURES FOR PSPICE A/D:

- Graphical design entry with OrCAD Capture/OrCAD Capture CIS
- New use-model of varying model parameters through property control from OrCAD Capture
- Simulation setup with easy to use dialogs
- Hierarchical netlisting
- Cross-probing with OrCAD Capture/OrCAD Capture CIS
- Plot window templates
- Probe Windows waveform: viewer and analyzer
- Multiple-named simulation profiles
- DC sweep, AC sweep, and transient analysis
- Noise, Fourier, and temperature analysis
- Parametric analysis (STEP)
- Monte Carlo and sensitivity/worst case analysis
- Parametric Plotter
- Preemptive simulation
- Interactive simulation control
- Analog behavioral modeling
- Propagation delay modeling for digital gates
- Constraint checking (e.g. setup-and-hold timing)
- Digital worst-case timing
- Charge storage on digital nets
- Stimulus Editor (STIMULUS and STIMLIB)
- Model Editor for device characterization
- Measurements and performance analysis
- Save/load bias point (.SAVEBIAS/.LOADBIAS)
- Power measurement with cross-probing
- Two simulation engines
- PWMs



PSpice provides a complete simulation environment including simulation, waveform analysis with cross-probing, and bias results display on the schematic

PSpice A/D is a mature, proven, advanced, native mixed-signal simulator. It is used to simulate complex mixed-signal designs, containing both analog and digital parts and to support models such as IGBTs, pulse width modulators, DACs, and ADCs. The viewing of simulation results, both analog and digital, has been simplified by employing a single display for the mixed-signal analysis results while retaining the same time axis.

PRODUCT FEATURES AND BENEFITS

Design entry and editing

- Enter designs with OrCAD Capture® design entry, or OrCAD Capture CIS — the world's most popular schematic entry systems
- Select from a library of over 18,000 simulation models, or choose from OrCAD Capture/OrCAD Capture CIS library of parts for general schematic entry
- Easily import existing PSpice designs — created with MicroSim Schematics — into the OrCAD Capture/PSpice environment to upgrade your design process
- Navigate through complex designs quickly with the hierarchical browser
- Create hierarchical block diagrams with automatic pin placement on hierarchical blocks
- Wiring of analog and digital components will reflect true signal analysis. The simulator automatically handles the tradeoffs between analog and digital domains
- Cadence® Allegro® Design Entry HDL is also fully integrated with PSpice, including one-button simulation and cross-probing

Stimulus creation

- Invoke the interactive, graphical PSpice Stimulus Editor from OrCAD Capture/OrCAD Capture CIS to define and preview stimulus characteristics
- Access built-in functions that can be described parametrically, or draw piecewise linear (PWL) signals freehand with the mouse to create any shape stimulus
- Create digital stimuli for signals, clocks, and buses — click-and-drag to introduce and move transitions

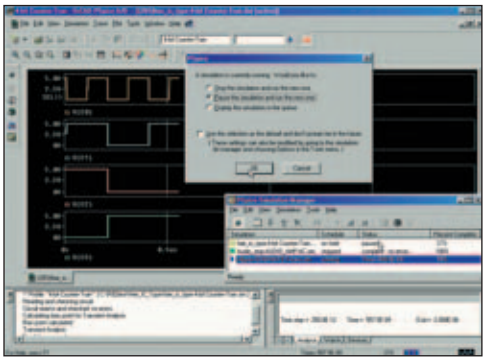
CIRCUIT SIMULATION

OrCAD Capture/PSpice integration

- Set up and run simulations, and cross-probe simulation results from OrCAD Capture/OrCAD Capture CIS
- Use the hierarchical netlister with parametric subcircuits to expedite netlisting of complex hierarchical designs

OrCAD®

- Run expanded simulations in the background while you continue editing new or existing designs
- Create multiple simulation profiles and save them in OrCAD Capture/OrCAD Capture CIS with Project Manager, so you can recall and run different simulations on the same schematic
- View simulation bias results directly on the schematic, including node voltages, pin and sub-circuit currents, and device power calculations



Pause a long simulation to run a shorter one. The Simulation Manager shows the current status of the simulations being performed

Simulation control

- Monitor simulation runs, view simulation messages and results as setup files are edited — all from a unified simulation environment
- Utilize analog analysis capabilities such as user-defined accuracy, automatic time-step control, and proprietary convergence algorithms to control the simulation process
- Interactively trade off accuracy and simulation time by loosening tolerances and time steps during non-critical periods of transient analyses, or by extending a transient analysis beyond pre-specified end time
- Preempt the current simulation to immediately run another one, then return to complete the preempted simulation later; control the queue of simulations waiting to be performed

Mixed analog/digital simulation

- PSpice A/D automatically recognizes A-to-D and D-to-A signals and properly simulates them by inserting interface subcircuits and power supplies
- Integrated analog and event-driven digital simulations improve speed without loss of accuracy
- Single graphical waveform analyzer displays mixed analog and digital simulation results on the same time axis
- Digital functions support five logic levels and 64 strengths, load-dependent delays, and hazard/race checking

Analog analysis

- Explore circuit behavior using basic DC, AC, noise, and transient analyses
- View node voltages, pin currents, and power consumption or noise contributions of individual devices
- Include specific local temperature effects on individual devices for more accurate analyses
- Show circuit behavior variations as components change, via parametric, Monte Carlo, and worst case analyses

GRAPHICAL RESULTS

Probe windows

- Choose from an expanded set of mathematical functions to apply to simulation output variables
- View simulation results waveform windows
- Select waveforms by name or by marking a net, pin, or part in the schematic
- Use cross-probing markers once and they remain with the analysis — as you change and re-simulate the design, the marked wave-forms appear after each simulation
- View continuous, real-time "marching wave-forms" as simulation progresses
- Copy and paste high-resolution, scalable waveforms into other applications for producing documentation
- Create plot window templates and use them to easily make complex measurements by simply placing markers on desired pins, nets, and parts in the schematic
- Measure performance characteristics of your circuit using built-in measurement functions or create your own measurements

DATA DISPLAY

- Plot both real and complex functions of circuit voltage, current, and power consumption including Bodé plots for gain and phase margin and derivatives for small-signal characteristics
- Display Fourier transforms of time domain signals or inverse Fourier transforms of frequency domain signals
- Vary component values over multiple runs and quickly view results as a family of wave-forms with parametric, Monte Carlo, and worst-case analyses
- Plot waveform characteristics, such as rise time versus temperature or supply voltage, using parametric analysis
- Create histograms after Monte Carlo analyses to display the distribution of a characteristic, such as overshoot
- Waveforms stored in tabular form (.txt or .csv files) can now be imported into PSpice

MODELS

Accurate internal models

- Large variety of built-in models adds flexibility to your simulations — most include temperature effects
- Shipped models include R, L, C, and bipolar transistors, plus:
  - Built-in IGBTs
  - Seven MOSFET models, including industry standard BSIM3v3.2 and the new EKV 2.6 model
  - Five GaAsFET models, including Parker-Skellern and TriQuint TOM-2, TOM-3 models
  - Nonlinear magnetic models complete with saturation and hysteresis
  - Transmission line models that incorporate delay, reflection, loss, dispersion, and crosstalk
  - Digital primitives, including bi-directional transfer gates with analog I/O models
  - Two battery models are included which allow accurate simulation of the discharge cycle and operating conditions
- Device Equations Developer's Kit (DEDK) allows implementation of new internal model equations which can be used with PSpice

Model library

- Select from more than 18,000 analog and mixed-signal models of devices made in North America, Japan, and Europe
- More than 4,500 parameterized models for BJTs, JFETs, MOSFETs, IGBTs, SCRs, magnetic cores and toroids, power diodes and bridges, operational amplifiers, optocouplers, regulators, PWM controllers, multipliers, timers, and sample-and-holds — these models allow passing simulation parameters as properties from the Schematic Editor
- Access basic components plus a variety of macro-models for more complex devices, including operational amplifiers, comparators, regulators, optocouplers, ADCs, and DACs
- Device Equations Developer's Kit (DEDK) is available by special arrangement with PSpice Technical Support — DEDK is intended for use by experienced device physicists and requires knowledge of C programming

SYMBOLS FROM MODELS

- Automatically generate OrCAD Capture/OrCAD Capture CIS parts for the models created by the Model Editor
- Automatically generate OrCAD Capture/OrCAD Capture CIS and Allegro Design Entry HDL part libraries from simulation model libraries obtained from part vendors or colleagues

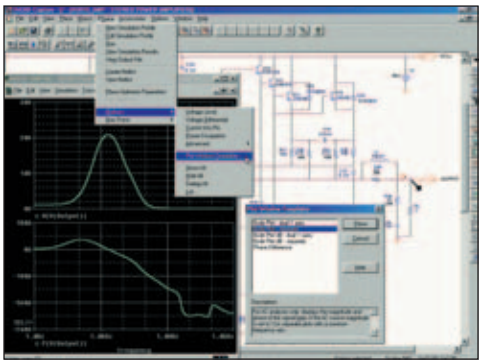
- Base the symbol generation on the PSpice symbol set or on your own
- Generate symbols for analog, digital, or mixed-signal devices (both primitives and macro models)
- New Symbol Wizard simplifies the creation of libraries from PSpice models

PSPICE MODEL EDITOR

- Click on a part in OrCAD Capture/OrCAD Capture CIS, and use the intuitive user interface of the PSpice Model Editor to view or edit its simulation model
- Extract a model of a supported device type by simply entering required data from the device's data sheet
- Proceed quickly through the extraction process using fully interactive features to guide you. Device characteristic curves give you quick graphical feedback
- New Symbol Wizard makes it easy to convert models to libraries with appropriate symbol shapes

BEHAVIORAL MODELING

- Describe functional blocks using mathematical expressions and functions
- Leverage a full set of mathematical operators, nonlinear functions, and filters
- Implement any transfer function via controlled voltage and current sources
- Define circuit behavior in the time or frequency domain, by formula (including Laplace transforms), or by look-up tables
- Select parameters which have been passed to subcircuits in a hierarchy and insert them into transfer functions
- New behavioral capabilities include many mathematical functions like in(x), exp(x), and sqrt(x)
- Specify error and warning messages in different conditions
- Create Boolean expressions that reference internal states and pin-to-pin delays using digital behavioral modeling

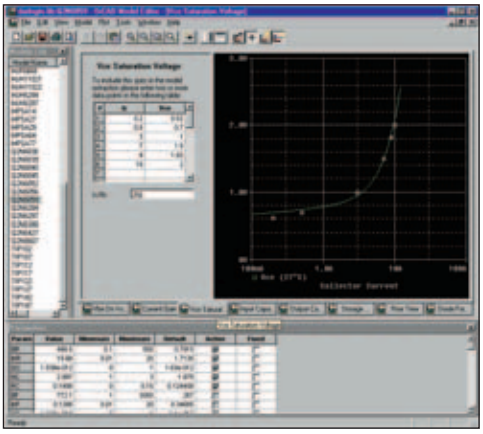


Use plot window markers to quickly perform common measurements. Choose from the markers provided, or create your own in PSpice

MAGNETIC PARTS EDITOR

Magnetic Parts Editor helps you overcome the issues involved in designing transformers manually. Using the Magnetic Parts Editor, you are able to:

- Design magnetic transformers and DC inductors
- Generate simulation models for transformers and inductors — these models can then be used in PSpice simulatable circuits
- Generate data required for manufacturing the transformers or inductors. The Manufacturer Report that is generated by MagDesigner after the completion of the design process contains the complete data required by a vendor to develop the transformer for commercial use



Use the Model Editor to create new PSpice models based on manufacturers' datasheets

ADVANCED ANALYSIS OPTIONS

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

SENSITIVITY

Sensitivity 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, then export the components to Optimizer to fine-tune the circuit behavior. Also use Sensitivity to identify which components affect yield the most, 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're also able to control the range around a point to determine sensitivity (default is 40% tolerance).

OPTIMIZER

The Optimizer in AMS Simulator analyzes analog circuits and systems, fine-tuning your designs faster than trial-and-error bench testing can. Optimizer helps you find the best component values to meet your performance goals and constraints. You can specify multiple goals and constraints to handle competing specifications. A specification can be as simple as an output voltage maximum, or it can be a more complex output calculation like the cutoff frequency for a low-pass filter, or it can be an entire curve using the Optimizer's curve fitting capability.

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
- 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

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.

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

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

PSpice A/D now has history support. You can store model parameter values used for each Monte Carlo run in a separate file and later reuse these values.

PARAMETRIC PLOTTER

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 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
- Post-analysis measurement evaluation

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)

TECHNICAL SUPPORT

Contact your EMA Design Automation support engineer for questions on any of the OrCAD products.

support.ema-eda.com
techsupport@ema-eda.com
585-334-6001

SALES, PRICING INFORMATION, AND TRAINING

Contact your EMA sales representative for exact pricing information and details on internet training or classroom training in your area.

EMA Design Automation
225 Tech Park Drive
Rochester NY 14623
877-362-3321
www.ema-eda.com

OrCAD®

6321 7/05