You can count on PSpice® for accurate circuit simulation results and regular innovations. PSpice has been tried and proven by thousands of engineers. Since the first PSpice simulator was introduced in 1985, it has been continuously enhanced and structured to use the latest hardware and operating systems. Each subsequent generation has addressed technological advances in electronics, and helped designers with the simulation capabilities they need. The latest edition continues this tradition with new enhancements to the PSpice simulation engine and user interface.

**PSpice** is a full-featured simulator for serious analog designers. With its sophisticated internal models, you can simulate everything from high-frequency systems to low-power IC designs. Draw on PSpice’s large library of models for off-the-shelf parts, or create models for new devices from their data sheets. Fully understand and explore the relationships in your design with “what-if” and advanced analyses.

**PSpice A/D** is a sophisticated, native mixed-signal simulator and a superset of PSpice. Use it to simulate mixed-signal designs of any size, containing analog and digital parts ranging from IGBTs and pulse width modulators to DACs and ADCs. View your simulation results, both analog and digital, in the same window and on the same time axis.

**PSpice A/D Basics** is an entry-level, mixed-signal simulation tool. It’s ideal if you need a friendly environment for simulating simple or basic analog and mixed-signal designs. It imposes no limits on circuit size, performs functional simulations of digital parts in mixed-signal simulations, and lets you perform all of the basic PSpice analyses.

**PSpice Optimizer** automates the iterative process of re-running simulations and fine-tuning your design. Use it in conjunction with PSpice or PSpice A/D. Specify the parameters to vary and the measures of performance you want to optimize. PSpice Optimizer calculates the optimal parameter values for you.
Features

Circuit Design
Design entry and editing
• Use the advanced capabilities of Orcad Capture® or Capture CIS®, the world's most popular schematic entry system, to enter your designs
• Select from a library of over 16,000 parts for simulation*, or choose from Orcad Capture/CIS's large library of parts for general schematic entry
• Easily import existing PSpice designs created with PSpice® Schematics, into the Orcad Capture – PSpice environment
• Navigate through complex designs quickly with the hierarchical browser
• Create hierarchical block diagrams, with automatic pin placement on hierarchical blocks
• Connect analog and digital components on your schematic to reflect their physical connections; the simulator automatically handles the transitions between analog and digital domains
• Or continue to use PSpice Schematics until you are more familiar with Orcad Capture

Analog or mixed-signal simulator with Probe graphical post-processor

PSpice Model Library

PSpice Model Editor

Capture CIS

SpinCircuit

Engineering Database

MRP/ERP/PDM System

Device Equations

Measurements

Parameters

Symbols from models, bias info

Netlist, stimuli, simulation settings, cross-probing

---

BSIM3v3.1 models, EKV models

Data sheet specifications

Vendor models, www.PSpice.com

PSpice Model Library

Analog or mixed-signal simulator with Probe graphical post-processor
**Stimulus creation**
- Invoke the interactive, graphical PSpice Stimulus Editor from 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/CIS
- Use the hierarchical netlister with parametric subcircuits for faster netlisting of complex hierarchical designs
- Run long simulations in the background while you edit other designs
- Create multiple simulation profiles and save them in Capture/CIS's Project Manager, so you can recall and run different simulations on the same schematic

**Simulation control**
- Perform and monitor simulations, view simulation messages and graphical results, and view and edit text files, all from a unified simulation environment
- Take advantage of analog analysis capabilities such as user-defined accuracy, automatic time-step control and proprietary convergence algorithms
- Save time by loosening tolerances and time steps during non-critical periods of transient analyses, or by extending a transient analysis beyond its pre-specified end time without restarting
- 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
- Digital functions support 5 logic levels and 64 strengths, load-dependent delays, and hazard/race checking

Pause a long simulation to run a shorter one. The Simulation Manager shows the current status of the simulations being performed.
Mixed analog/digital simulation*  
- PSpice A/D automatically recognizes A-to-D and D-to-A connections, and properly processes them by inserting interface subcircuits and power supplies.  
- Integrated analog and event-driven digital simulation engines improve simulation speed without loss of accuracy.  
- Single graphical waveform analyzer displays mixed analog and digital simulation results on the same time axis.  

Basic and advanced 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 advanced parametric, Monte Carlo, and worst-case analyses.  

Graphical Results  
**Probe windows**  
- Choose from a rich set of mathematical functions to apply to simulation output variables.  
- View simulation results in multiple waveform windows.  
- Select waveforms by name or by marking a net, pin, or part in the schematic.  
- Place cross-probing markers once and they stay with the analysis as you change and re-simulate the design; the marked waveforms appear after each simulation.  
- View continuous, real-time “marching waveforms” 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, just by placing markers on desired pins, nets and parts in the schematic.  

Data display  
- 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.  
- 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 waveforms 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.  

*Use plot window markers to quickly perform common measurements. Choose from the markers provided, or create your own in 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 BSIM 3v3.1 and the new EKV 2.6 model
  - Five GaAsFET models, including Parker-Skellern and TriQuint TOM-2 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
- 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 16,000 analog and mixed-signal models of devices made in North America, Japan, and Europe
- Access basic components plus a variety of macro-models for more complex devices, including operational amplifiers, comparators, regulators, optocouplers, ADCs and DACs

**Symbols from models**

- Automatically generate Orcad Capture/CIS parts for the models created by the Model Editor
- Automatically generate Orcad Capture/CIS part libraries from simulation model libraries obtained from part vendors or colleagues
- Base the symbol generation on the PSpice symbol set, or your own
- Generate symbols for analog, digital, or mixed-signal devices (both primitives and macro-models)

**PSpice Model Editor**

- Click on a part in 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

**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
- Create Boolean expressions that reference internal states and pin-to-pin delays, using digital behavioral modeling

---

**New**

* Items denoted with an asterisk may not be available in some product versions; consult the product table on inside back cover.

** 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.
Fine-Tune Your Circuits Automatically With PSpice Optimizer

You can maximize the performance of your circuit automatically with PSpice Optimizer, which works in conjunction with PSpice or PSpice A/D. Optimizer offers more than a “batch mode” or scripting capability. It intelligently modifies parameters between simulation runs, using advanced goal-seeking to zero-in on the optimal set of values.

Set up the PSpice Optimizer to run your simulations automatically. On completion, you’ll know the values that deliver the best circuit performance.

- Tune the values of circuit parameters, automatically improve the performance of analog circuits, or match circuit behavior to one or more data curves
- Invoke the PSpice Optimizer from Orcad Capture/CIS, and back-annotate the optimized parameters automatically
- Choose up to eight circuit parameters to optimize
- Optimize parameters to meet up to eight performance specifications, fit simulation results to up to eight curves, or a combination of up to eight specifications and curve fits
- Choose specifications from standard measurements such as bandwidth, overshoot, rise time, and center frequency, or create custom measurements using the performance analysis wizard
- Specifications can either be goals or constraints; the optimization algorithms can satisfy nonlinear constraints
- Modified Newton method improves the target function with each iteration
- In addition to the optimal parameter values, PSpice Optimizer also reports:
  - The margins of each constraint
  - Sensitivity (derivative) of each specification with respect to each parameter
  - Performance cost (i.e., the Lagrange multipliers) of meeting each constraint

Here, PSpice Optimizer is minimizing amplifier power consumption within bandwidth and gain constraints by varying bias current and transistor dimensions.
# Major Feature Summary for PSpice, PSpice A/D and PSpice A/D Basics

<table>
<thead>
<tr>
<th>Key features</th>
<th>PSpice</th>
<th>PSpice A/D</th>
<th>PSpice A/D Basics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphical design entry (with Orcad Capture/CIS)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Simulation setup with easy to use dialogs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hierarchical netlisting</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cross-probing (with Orcad Capture/CIS)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Probe Windows waveform: viewer and analyzer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Symbols from models</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Multiple named simulation profiles</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Notable PSpice analysis and simulation features</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DC sweep, AC sweep and transient analysis</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Noise, Fourier, and temperature analysis</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Parametric analysis (STEP)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Monte Carlo and sensitivity/worst case analysis</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Preemptive simulation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interactive simulation control</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Notable PSpice devices and model libraries</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>GaAsFETs: Curtice, Statz, TriQuint, Parker-Skellern</td>
<td></td>
<td></td>
<td>Statz</td>
</tr>
<tr>
<td>MOSFETs: SPICE3 (1-3 with charge conservation)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MOSFETs: BSM1, BSM3 (versions 2 and 3.1), EKV 2.6</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IGBTs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Darlingtonons</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DACs and ADCs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>JFETs, BJTs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Resistor, capacitor, and inductor .MODEL support</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ideal and non-ideal lossy transmission lines</td>
<td></td>
<td></td>
<td>Ideal</td>
</tr>
<tr>
<td>Coupled transmission lines</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Coupled inductors</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nonlinear magnetics</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Voltage and current-controlled switches</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analog model library</td>
<td>14,000</td>
<td>14,000</td>
<td>7,500+(^A)</td>
</tr>
<tr>
<td>Digital primitives</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Digital model library</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Product options</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSpice Optimizer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Network Licensing</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Device equations support(^C)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\(A\) PSpice A/D Basics includes all libraries except IGBTs, SCRs, thyristors, PWMs, magnetic cores, transmission lines and ADCs and DACs.

\(B\) PSpice A/D Basics does not include bi-directional transfer gates.

\(C\) PSpice and PSpice A/D packages can utilize any device models developed using Device Equations Developer’s Kit (DEDK). DEDK itself is only available by special arrangement with PSpice Technical Support. DEDK is intended for use by experienced device physicists and requires knowledge of C programming.
System Requirements
PSpice “plugs-in” to Orcad Capture/CIS; please consult those products’ data sheets for hardware and operating system requirements.

PSpice requires these system resources:
• Intel Pentium™ or equivalent processor
• Windows 2000®, Windows NT® 4.0 (with Service Pack 3 or later installed), Windows 98®, or Windows 95®
• 32 MB RAM (64 MB recommended. Additional RAM improves performance)
• 50 MB free hard disk space (in addition to Capture or Capture CIS requirements)
• 800 X 600 minimum display resolution (1024 X 768 recommended)

Year 2000 Compliance
All PSpice products for Microsoft Windows 2000, Windows NT, Windows 98 and Windows 95 are Year 2000 Compliant.

Product Support
Every PSpice product comes with:
• One year of technical support via phone, email and fax
• Access to the PSpice Internet-based technical information and support connection at www.orcad.com under Enterprise Services
• A one-year free subscription to product updates
• Training workshops and other services are also available. Contact Cadence PCB Systems Division at 1-800-671-9505 or a Cadence International Reseller for more information.

Upgrade Policy
When you purchase a PSpice product, you will receive a generous trade-in allowance if you later decide to upgrade to a version with higher functionality. You can load existing designs into a higher-functionality product, and use it without conversion or modifications.

Pricing and Ordering Information
In North America:
• Contact Cadence PCB Systems Division at 1-800-671-9505
• Email to pcbinfo@cadence.com
• Fax to (503) 671-9501

Outside of North America:
• Contact your Cadence International Reseller
• Email to intl@orcad.com

For a list of resellers visit our Web site at www.orcad.com/contact/contact.asp.