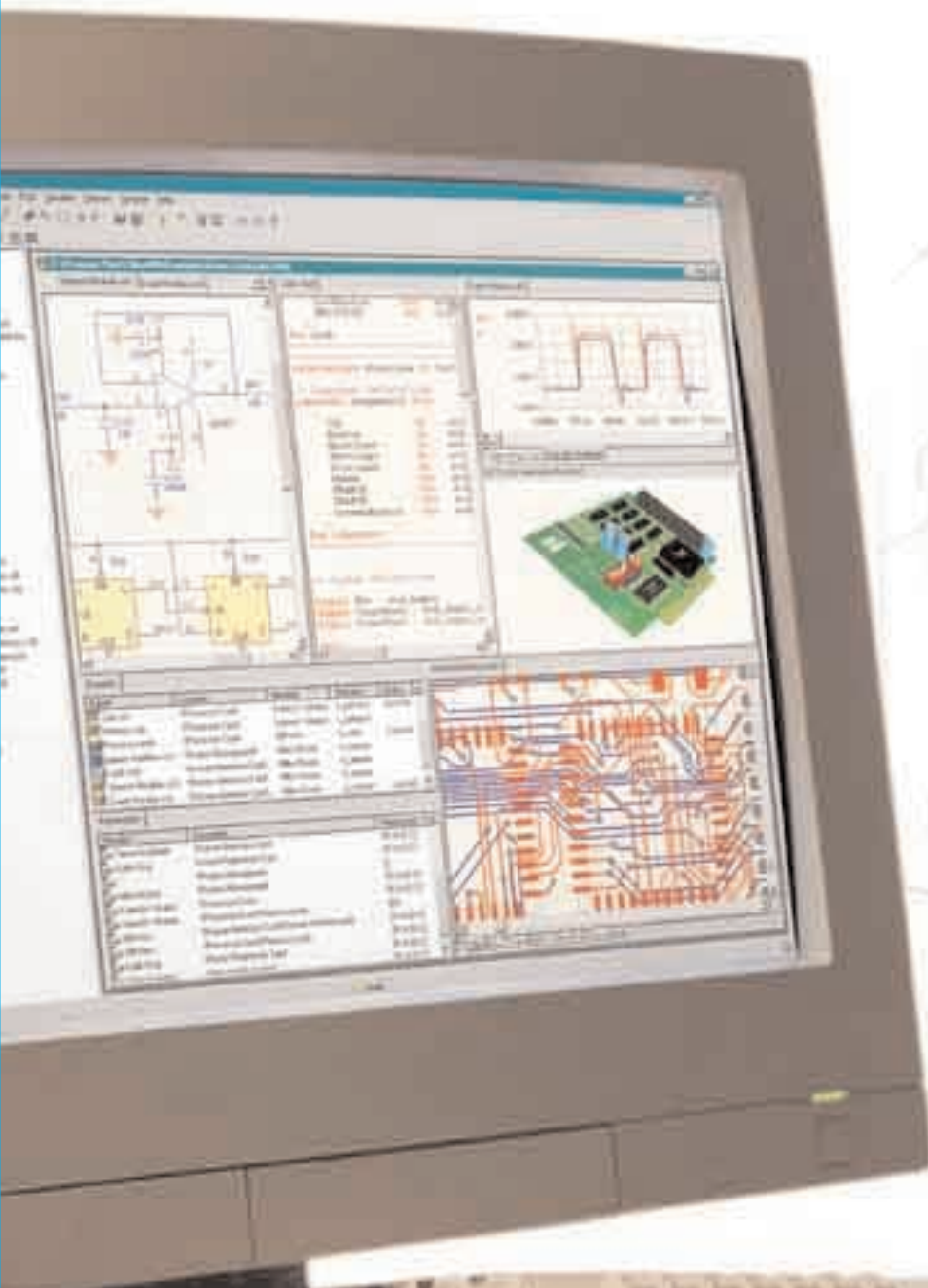


Protel 99



Advanced mixed-signal circuit simulation

Knowing your circuit will work the way you expect is crucial to timely product development. That's why you need proven simulation tools in your design kit. Based on the latest XSPICE 3f5 engine, Protel 99's Simulation Editor is a powerful mixed analog and digital circuit simulator that works hand-in-hand with Protel's schematic entry program to provide a complete front-end design solution.

Advanced simulation technology

Using advanced analog and digital modeling techniques, Protel 99's integrated simulator lets you verify and fine tune your design before laying out the board. Generate accurate, "real-world" simulations of analog, digital and mixed-mode circuits directly from the schematic at the touch of a button.

With no limit on the number of circuit nodes, and simulation-ready schematic libraries that include almost 6,000 analog and digital devices, Protel 99's simulation tools can handle designs of any complexity. Effects such as propagation delays, setup and hold times, output loading – in fact almost all physical circuit parameters – are fully accounted for, ensuring you get results you can trust.

Being a true mixed-signal environment, Protel 99's Circuit Simulator integrates the simulation of continuous analog waveforms and discrete digital signals. You can run and view complex analog and digital simulation waveforms side-by-side, giving a full picture of your circuit's performance.

True SPICE compatibility

Protel 99's Circuit Simulator uses an enhanced version of Berkeley SPICE3f5/XSPICE allowing you to accurately simulate any combination of analog and digital devices, without having to manually insert D/A or A/D converters. The simulator includes accurate, event-driven behavioral models for digital devices, including TTL and CMOS, taking the work out of mixed-mode design simulation.

And because Protel 99's Circuit Simulator provides true SPICE compatibility – the industry standard for analog simulation – you can use all the latest device manufacturers' models directly with Protel 99.

SimCode for digital simulation

In Protel 99 digital devices are accurately modeled using Digital SimCode™, a high-level language extension to XSPICE. In addition

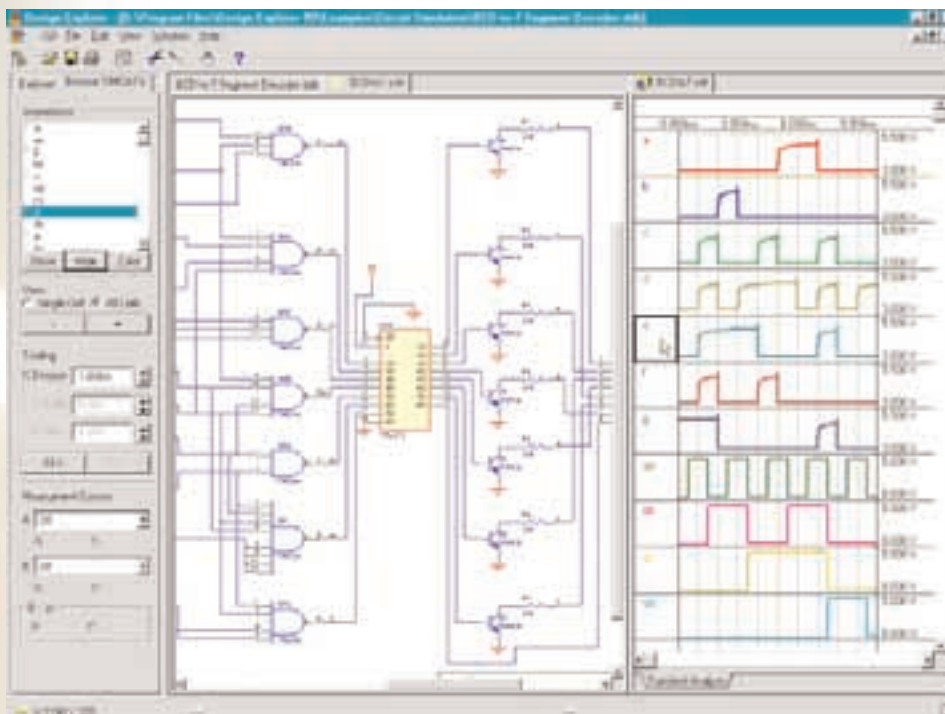
to standard event-driven component modeling, Digital SimCode allows you to specify device propagation delays, input and output loading, and power supply-dependent behavior.

Control over the digital modeling parameters means more accurate simulation results, allowing you to investigate subtle timing and loading effects. Protel 99's Circuit Simulator gets you closer to your design than ever before.

Simulation made easy

Seamless integration with the Schematic Editor makes simulation a breeze in Protel 99. Simply construct your circuit schematic using components from the supplied simulation-ready libraries and, at the press of a button, your design comes to life. Analysis results are displayed in the integrated waveform viewer, which provides extensive "CRO-like" view and measurement controls, including two user-definable measurement cursors.

And there's no need to edit cryptic text files to configure a simulation run in Protel 99. All simulation parameters are controlled through a single intuitive dialog. Simulation configuration settings are stored with the design, so you can easily make modifications to your circuit then re-run the simulation at any time to see the results.



Protel 99's true mixed-mode simulation gives you the full picture of your design's performance, with analog and digital signals displayed side-by-side.

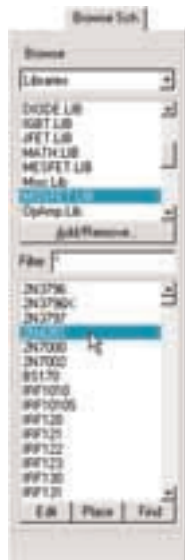
Visit www.protel.com for more product information.

at the touch of a button

Comprehensive model libraries

Protel 99 comes supplied with over 5800 simulation-ready schematic components in more than 20 libraries. These libraries are constantly maintained by the Protel Library Development Center, which specializes in developing libraries for all Protel products. New and updated libraries are available free from the Protel web site, www.protel.com.

Each simulation-ready component is linked to an appropriate simulation model, and also contains full PCB footprint information. This means you can go directly from simulation to PCB layout without modifying your schematic, saving precious design time.



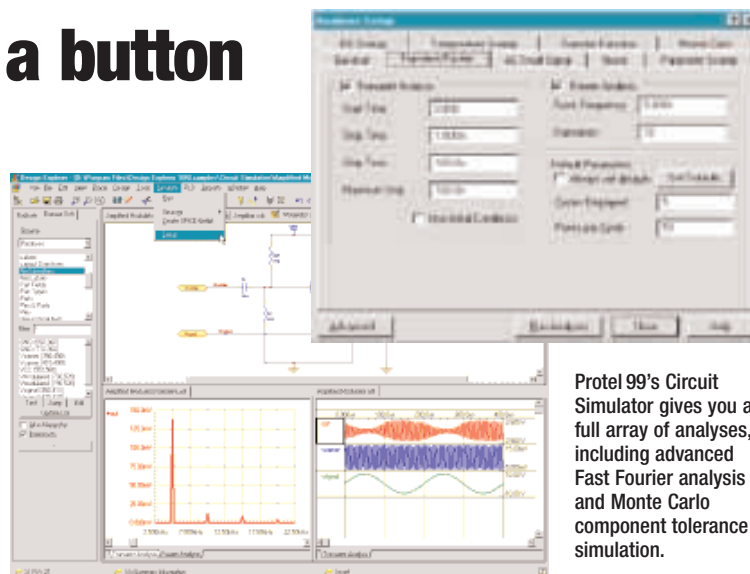
Protel 99 comes supplied with over 5800 simulation-ready schematic components in more than 20 libraries. New and updated libraries are available free from the Protel web site, www.protel.com.

Complete support for device manufacturers' models

True SPICE 3f5 compatibility means that with Protel 99 you can use pure SPICE device models from manufacturers such as Motorola, Texas Instruments and others. The simulator can read these models directly, ensuring accurate simulation of your circuit using 'real-world' component parameters.

With Protel 99's Circuit Simulator you have the widest choice of components available to use in your design.

Protel 99's Circuit Simulator offers a full range of linear and non-linear dependent sources for defining "black box" circuit behavior. Use standard mathematical operators and functions such as LOG, LN, EXP, SIN, etc, to create transfer functions of any complexity.



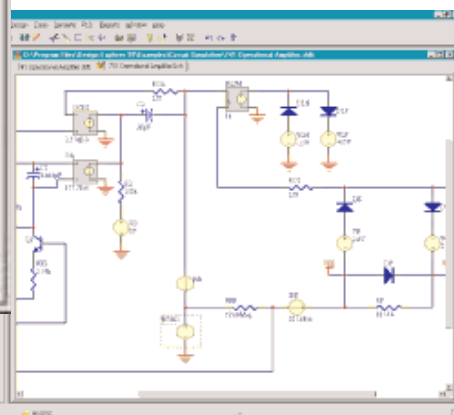
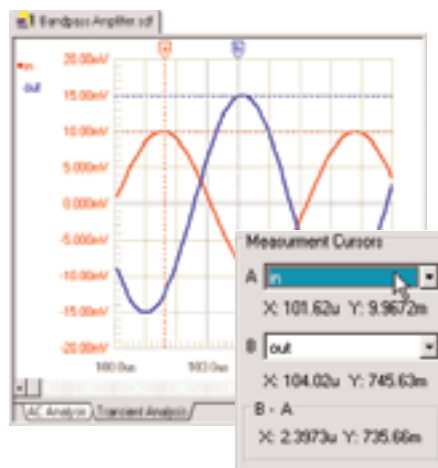
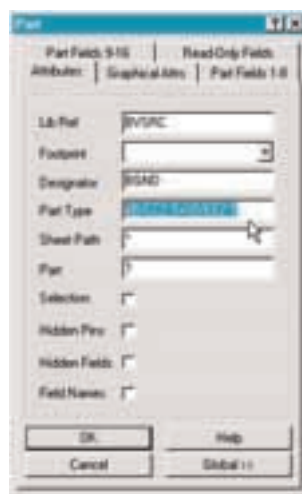
Protel 99's Circuit Simulator gives you a full array of analyses, including advanced Fast Fourier analysis and Monte Carlo component tolerance simulation.

Comprehensive and powerful analysis tools

Protel 99's Circuit Simulator supports an array of powerful circuit analysis tools, including AC Small Signal, Transient, Noise and DC Transfer. Beyond the basics, you can test your circuit using sophisticated Monte Carlo component tolerance analysis, component value and temperature sweeps, and Fast Fourier analysis.

All analyses offer a host of configuration options and are backed by proven simulation algorithms, providing a true picture of your circuit's performance under all conditions.

Measurements can be taken directly from the simulation waveforms using assignable cursors. Each cursor can be assigned to any displayed waveform, making cross-waveform phase and level measurements easy.



Versatile and dependent sources

Protel 99's Circuit Simulator offers a full range linear and non-linear dependent sources for defining "black box" circuit behavior, including simple linear effects or complex mathematical functions based on voltages and currents present in the circuit. These dependent sources allow you to easily integrate voltage or current-controlled devices into your circuit. Use standard mathematical operators and functions such as LOG, LN, EXP, SIN, etc, to create transfer functions of any complexity.

MIXED SIGNAL CIRCUIT SIMULATION



Feature Highlights

- Leading edge mixed-signal simulation technology
- Seamless integration with the Schematic Editor
- Comprehensive, simulation-ready device libraries
- XSPICE 3f5 compatibility for analog simulation
- Impressive array of device models provided
- Comprehensive analyses, including AC small signal, Transient, Noise, and DC Transfer
- Sophisticated component sweep and Monte Carlo analysis modes for testing the effects of component variations and tolerances
- Support for complex sources, including controlled sources, equation-defined sources, and source waveforms based on lookup tables
- Integrated oscilloscope-like waveform viewer with selective waveform scaling, color control and measurement cursors

Specifications

Analyses:
Operating point, Transient, Fourier, AC small signal, DC sweep (2 variables), Noise, Transfer function, Temperature sweep (2 variables), Parameter sweep (2 variables), Monte Carlo

Analog modelling:
SPICE 3f5 compatible

Digital modelling:
Digital SimCode (XSPICE compatible)

AC sweep types:
Linear, Decade, Octave

Monte Carlo distributions:
Uniform, Gaussian, Worst Case

Contact your local sales office today for your FREE Protel 99 30-Day Trial CD.