

**TANNER EDA STREAMLINES IC DESIGN SIMULATION AND ANALYSIS
WITH T-SPICE PRO 9**

To demo at DAC 2003, June 2-4, Anaheim, CA, booth #2031

PASADENA, California --June 2, 2003-- Tanner EDA, a leading developer of electronic design automation (EDA) software for integrated circuit (IC) designers, has launched the latest version of its T-Spice Pro software.

Designed for faster and more accurate circuit simulation, schematic entry and waveform analysis of IC designs on the Windows® platform, T-Spice Pro 9 can be used for analog, mixed-signal and MEMS designs. The software features enhancements to its user interface both in command input functionality and new data output options, as well as performance improvements to the simulator engine.

For input, the software includes one-click linking from syntax errors back to the offending source line, a new text editor with SPICE syntax highlighting and an advanced Command Insert tool for guiding command script creation and eliminating syntax errors.

For output, T-Spice Pro 9 features three new options. The first option displays detailed information for each device in the circuit, including electrical parameters, node connectivity and physical dimensions. The second option controls output of comprehensive information on each transistor model used in the circuit. The model listing may be used for determining default settings for each model parameter and for verifying the settings that the user has entered. The third output option enables advanced users to access detailed internal electrical properties of circuit elements, such as internal voltages, currents, charges, capacitances and control parameters. These values may then be plotted with the W-Edit waveform viewer to assist designers in understanding how the circuit works.

Version 9 builds on key T-Spice Pro features including Monte Carlo analysis, the powerful Levenberg-Marquardt optimizer and parameter sweeping. The software continues to refine advanced numerical techniques to allow for quick table-based simulations and precise circuit behavior characterization.

It also includes all of the same robust features found in previous versions, such as an option for trapezoidal integration and a new time-step algorithm that result in a more accurate transient analysis, support for user defined functions in electrical and physical device parameters, and save and load commands to capture the state of a circuit simulation and continue from that point onward in a later simulation.

“Simulation and analysis are crucial steps in the IC design process. They identify whether a design will actually perform as designed, and they pinpoint specific areas for improvement early in the device development cycle. Today’s requirement for first-time silicon success requires a simulator like T-Spice,” said Dr. John Tanner, President and CEO of Tanner Research, Inc. “T-Spice Pro 9 can handle tough high-feedback circuits, giving fast, accurate results and it sets a new standard in ease of use”

T-Spice Pro 9 is now available for new customers or as an upgrade option for current T-Spice Pro users. Support is provided for Windows® XP, 2000, NT and 98/ME platforms. Please contact Tanner EDA sales for pricing at sales@tanner.com or toll-free at 877/325-2223.

About Tanner EDA

Tanner EDA is a division of Tanner Research, Inc., a privately held company headquartered in Pasadena, California. Tanner EDA develops layout, verification and electronic simulation design software for analog and mixed-signal IC, MEMS and integrated optical designs on the Windows® platform. Tanner’s L-Edit Pro™ and T-Spice Pro™ provide a complete design flow including schematic entry, waveform probing, full-custom layout editing, placement and routing, LVS and DRC verification, netlist extraction and circuit simulation. Founded in 1988, Tanner Research, Inc. consists of three major divisions in addition to Tanner EDA. For more information about Tanner EDA, please visit www.tannereda.com.