

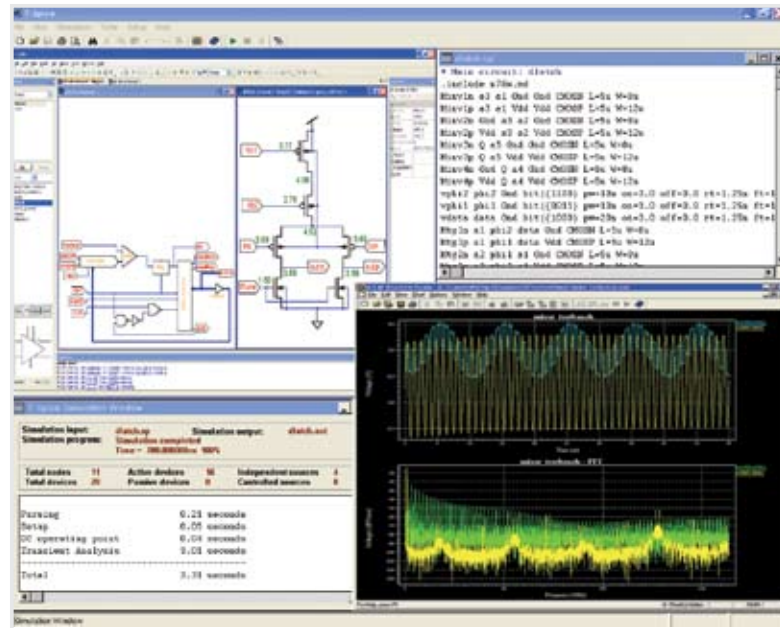
T-SPICE FOR CIRCUIT SIMULATION

Product Benefits

- Performs fast, accurate simulations for analog and mixed-signal IC designs.
- Fully supports foundry models for reliable and accurate simulations.
- Multi-threading support for multiple processor cores delivers dramatically shorter run times.
- Tight integration with schematic capture and waveform cross-probing.

Product Features

- Offers HSPICE® and PSpice® compatible syntax.
- Provides a user-friendly graphical interface with simple point-and-click environment to let you run, pause, resume, and stop simulations easily.
- Enables easy creation of syntax-correct SPICE with the built-in command wizard.
- Applies the most advanced numerical techniques for superior convergence.
- Automatically selects advanced convergence algorithms for reliable DC convergence.
- Supports the latest foundry MOSFET and BJT processes.
- Supports Verilog-A for analog behavioral modeling, allowing designers to prove system level designs before doing full device level design.
- Verilog-A support enables model developers of TFTs, MEMS, and sensors to specify compact models in a high level behavioral language.
- Lets you precisely characterize circuit behavior using virtual data measurements, Monte Carlo analysis, and parameter sweeping.
- Applies a powerful Levenberg-Marquardt non-linear optimizer to achieve performance goals within specified design constraints.



Speeding Concept to Silicon

To transform your ideas into designs, you must be able to simulate large circuits quickly and with a high degree of accuracy. That means you need a simulation tool that offers fast run times, integrates with your other design tools, and is compatible with industry standards.

Tanner T-Spice™ Circuit Simulator puts you in control of simulation jobs with an easy-to-use graphical interface and a faster, more intuitive design environment. With key features such as multi-threading support, device state plotting, real-time waveform viewing and analysis, and a command wizard for simpler SPICE syntax creation, T-Spice saves you time and money during the simulation phase of your design flow.

T-Spice enables more accurate simulations by supporting the latest transistor models—including BSIM4 and the Penn State Philips (PSP) model. Given that T-Spice is compatible

with a wide range of design solutions and runs on Windows-based systems, it fits easily and cost-effectively into your current tool flow.

Improve simulation accuracy with advanced modeling features

T-Spice provides extensive support of behavioral models using Verilog-A, expression controlled sources, and table-mode simulation. Behavioral models give you the flexibility to create customized models of virtually any device. T-Spice also supports the latest industry models, including the transistor model recently selected as the next standard for simulating future CMOS transistors manufactured at 65 nanometers and below—the Penn State Philips (PSP) model. PSP will simplify the exchange of chip design information and support more accurate digital, analog, and mixed-signal circuit behavior analysis.

T-Spice also supports foundry extensions, including HSPICE® foundry extensions to models.

- Supports PSP, BSIM3.3, BSIM4.5, BSIM SOI 4.0, EKV 2.6, MOS 9, 11, 20, 30, 31, 40, RPI a-Si & Poly-Si TFT, VBIC, Modella, and MEXTRAM models.
- Includes two stress effect models, from the Berkeley BSIM4 model and from TSMC processes, in the BSIM3 model to provide more accuracy in smaller geometry processes.
- Supports gate and body resistance networks in RF modeling.
- Performs non-quasi-static (NQS) modeling.
- Supports comprehensive geometry-based parasitic models for multi-finger devices.
- Models partially depleted, fully depleted, and unified FD-PD SOI devices.
- Models self-heating and RF resistor networks.
- Performs table-based modeling for using measured device data to model a device.
- Includes enhanced diode and temperature equations to improve compatibility with many foundry model libraries.

Work in a faster, easier design environment

T-Spice helps integrate your design flow from schematic capture through simulation and waveform viewing. An easy-to-use point-and-click environment gives you complete control over the simulation process for greater efficiency and productivity.

- Enables easy creation of syntax-correct SPICE through a command wizard.
- Highlights SPICE Syntax through a text editor.
- Provides Fast, Accurate, and Precise options to enable optimal balance of accuracy and performance.
- Enables you to link from syntax errors to the SPICE deck by double clicking.

- Supports Verilog-A for analog behavioral modeling, allowing designers to prove system level designs before doing full device level design.
- Provides “.alter” command for easy what-if simulations with netlist changes.

Perform sophisticated analysis

T-Spice uses superior numerical techniques to achieve convergence for circuits that are often impossible to simulate with other SPICE programs. The types of circuit analysis it performs include:

- DC and AC analysis.
- Transient analysis with Gear or trapezoidal integration.
- Enhanced noise analysis.
- Monte Carlo analysis over unlimited variables and trials.
- Virtual measurements with functions for timing, error, and statistical analysis.
- Parameter sweeping using linear, log, discrete value, or external file data sweeps.

With T-Spice, you can:

- Optimize designs with variables and multiple constraints by applying a Levenberg-Marquardt non-linear optimizer.
- Use plot statements that support wildcards.
- Use plot statements and parameter definitions that support mathematical expressions involving C-style math functions.
- Use bit and bus logic waveform inputs.

Benefit from flexible licensing

When you purchase a new design tool, licensing options can greatly affect your total cost of ownership. T-Spice is available in node-locked and networked configurations offering you the most flexible licensing possible. With a single solution, T-Spice will work whenever and wherever meeting the design needs of your main workgroup and remote workers. If you offshore design projects, T-Spice does not have geographic restriction on its licenses, thus, lowering your total cost of ownership.

Integrated environment for schematic capture and waveform cross-probing

- Tanner EDA's S-Edit schematic capture tool complements T-Spice by providing an integrated environment for editing circuits, setting up and running simulations and probing the results.

W-Edit

- Provides an intuitive multiple-window, multiple-chart interface for easy viewing of waveforms and data in highly configurable formats.
- Links dynamically to T-Spice with a run-time update feature that displays simulation results in real time as the simulator is running.
- Provides quick and interactive FFT analysis of transient data, with multiple windowing functions, and easy control of the accuracy and format of final plots.
- Creates new traces based on mathematical expressions of other traces.

Tanner
EDA

Corporate Headquarters
825 South Myrtle Avenue
Monrovia, CA 91016-3424 USA
Tel: +1-626-471-9700
Toll Free: 877-325-2223
Fax: +1-626-471-9800
Email: sales@tanner.com
Web: www.tannereda.com

Tanner Research Japan K.K.
Burex Kojimachi 6F
3-5-2 Kojimachi, Chiyoda-ku
Tokyo 〒 102-0083
Japan
Tel: +81 (03) -3239-2840
Fax: +81 (03) -3239-2860
Email: sales.jp@tanner.com
Web: www.tanner.jp

Tanner Research Taiwan, Inc.
3FB1, No. 1, Li-Hsin 1st Road
Science Based Industrial Park
Hsinchu, 300 Taiwan ROC
Tel: 886-3-666-2112
Fax: 886-3-666-3697
Email: sales.tw@tanner.com
Web: www.tanner.com.tw

© 2008 Tanner Research. All rights reserved.
All other company and/or product names are the property of their respective owners.