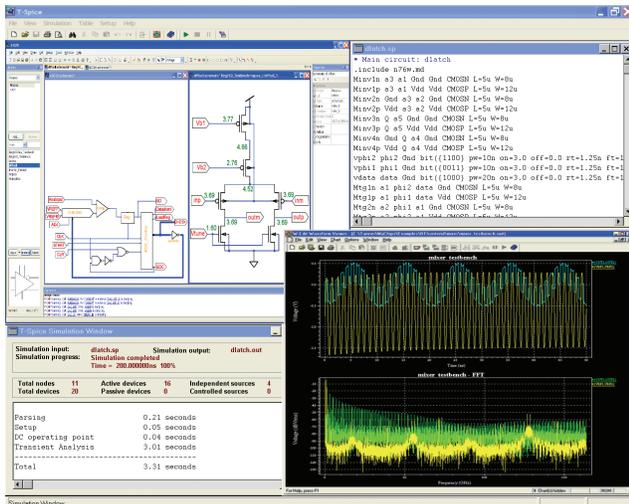


# Tanner T-Spice Simulation



Tanner T-Spice Simulator UI displays schematic, spice netlist, simulation logs and waveform data.

## Fast, Accurate Simulations for Analog and Mixed-signal IC Designs

Tanner T-Spice simulation provides fast, accurate simulation for analog and analog/mixed-signal (AMS) IC designs. T-Spice not only simulates circuits quickly and with a high degree of accuracy, but also is compatible with industry leading standards and integrates easily with the Tanner S-Edit schematic capture tool and Tanner Waveform Viewer. T-Spice includes improved accuracy with advanced modeling, multi-threading support, device state plotting, real-time waveform viewing and analysis, and a command wizard for simple SPICE syntax creation.

## Improve Simulation Accuracy with Advanced Modeling

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 BSIM4 and the Penn State Philips (PSP) model. And T-Spice supports foundry extensions, including HSPICE foundry extensions to models. Other modeling-related benefits include:

- Supports PSP, BSIM3.3, BSIM4.8, BSIM SOI 4.5, EKV 2.6, MOS 9, 11, 20, 30, 31, 40, RPI a-Si & Poly-Si TFT, HiSim, VBIC, Modella, and MEXTRAM models; also includes enhanced diode and temperature equations to improve compatibility with many foundry model libraries
- 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; also performs non-quasi-static (NQS) modeling
- Supports geometry-based parasitic models for multi-finger devices
- Models FD-PD SOI devices; and self-heating and RF resistor networks
- Performs table-based modeling using measured device data to model a device

## FEATURES AND BENEFITS:

- Fast, accurate analog and analog/mixed-signal (AMS) circuit simulation
- Multi-threading support for shorter run times
- HSPICE- and PSpice-compatible syntax to allow easy integration of legacy designs and foundry models
- Support for the latest industry foundry models, including PSP, BSIM3.3, BSIM4.8, BSIM SOI 4.4, EKV 2.6, MOS 9, PSP, RPI a-Si & Poly-Si TFT, VBIC, and MEXTRAM models for reliable and accurate simulations
- Verilog-A and Verilog-AMS support for mixed-signal co-simulation
- Accurately characterize circuit behavior using virtual data measurements, Monte Carlo analysis, parameter sweeping, DC analysis, AC/noise analysis and transient analysis
- Automatically selects advanced convergence algorithms for reliable DC convergence
- Ease of use: intuitive and quick learning curve
- Unparalleled customer support
- Flexible licensing

## 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 productivity. Among the other benefits to your design flow:

- Enables easy creation of syntax-correct SPICE through a command wizard; also highlights SPICE syntax through a text editor
- Provides fast, accurate, and precise options to enable optimal balance of accuracy and performance
- Enables you to jump from error messages to the SPICE deck where the error is located by double-clicking
- Supports Verilog-A for 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, including AC noise analysis
- Transient analysis with gear or trapezoidal integration
- 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

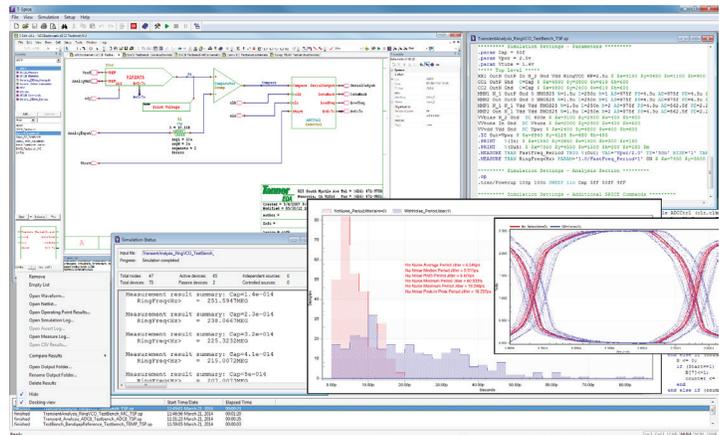
- Use plot statements that support wildcards to view internal device properties
- Use bit and bus logic waveform inputs

## Tanner S-Edit Schematic Capture

- Complements T-Spice by providing an integrated environment for editing circuits, setting up and running simulations, and probing the results

## Tanner Waveform Viewer

- Provides an intuitive multiple-window, multiple-chart interface for easy viewing and analyzing of waveforms and data in highly configurable formats
- Links dynamically to T-Spice with a runtime update feature that displays simulation results in real time as the simulator is running
- Performs fast display of large data files with support for simultaneously viewing multiple simulations allowing quick what-if analysis
- Built-in waveform calculator gives interactive measurements, including real time dynamic annotations
- Creates new traces based on mathematical expressions of other traces using the waveform calculator



Tanner T-Spice is integrated with Tanner S-Edit schematic capture and Tanner Waveform Viewer, providing a productive integrated environment.

For the latest product information, contact us at: [www.mentor.com](http://www.mentor.com), (800) 547-3000

©2015 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

**Corporate Headquarters**  
Mentor Graphics Corporation  
8005 SW Boeckman Road  
Wilsonville, OR 97070-7777  
Phone: 503.685.7000  
Fax: 503.685.1204

**Sales and Product Information**  
Phone: 800.547.3000  
[sales\\_info@mentor.com](mailto:sales_info@mentor.com)

**Silicon Valley**  
Mentor Graphics Corporation  
46871 Bayside Parkway  
Fremont, CA 94538 USA  
Phone: 510.354.7400  
Fax: 510.354.7467

**North American Support Center**  
Phone: 800.547.4303

**Europe**  
Mentor Graphics  
Deutschland GmbH  
Arnulfstrasse 201  
80634 Munich  
Germany  
Phone: +49.89.57096.0  
Fax: +49.89.57096.400

**Pacific Rim**  
Mentor Graphics (Taiwan)  
11F, No. 120, Section 2,  
Gongdao 5th Road  
HsinChu City 300,  
Taiwan, ROC  
Phone: 886.3.513.1000  
Fax: 886.3.573.4734

**Japan**  
Mentor Graphics Japan Co., Ltd.  
Gotenyama Garden  
7-35, Kita-Shinagawa 4-chome  
Shinagawa-Ku, Tokyo 140-0001  
Japan  
Phone: +81.3.5488.3033  
Fax: +81.3.5488.3004

**Mentor Graphics**

MGC 07-15 1035530-w