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® and Linux® platforms, 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, BSIM4.6, BSIM SOI 4.0, EKV 2.6, MOS 9, 11, 20, 30, 31, 40, PSP, 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.


- 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 with device and lot variations.

- Virtual measurements with functions for timing, error, and statistical analysis including common measurements such as delay, risetime, frequency, period, pulse width, settling time, and slew rate.

- Parameter sweeping using linear, log, discrete value, or external file data sweeps.

- 64-bit engine for increased capacity and higher performance.

With T-Spice, you can:

- Optimize designs with variables and multiple constraints by applying a Levenberg-Marquardt non-linear optimizer.

- Perform Safe Operating Area (SOA) checks to create robust designs.

- 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 proving the results.

**W-Edit**

- 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 run-time 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.

- Provides quick and interactive measurements with built-in waveform calculator including real time dynamic annotations.

- Creates new traces based on mathematical expressions of others and is fully scriptable via the command line.

Find out what T-Spice can do for your design challenges. Contact us at DesignChallenge@tannereda.com for a 30-day evaluation.