



[SYSTEMS PRODUCTS](#)

[LOGICAL PRODUCTS](#)

[PHYSICAL
IMPLEMENTATION](#)

[SIMULATION AND
ANALYSIS](#)

[Cosmos-Scope](#)

[CosmosSE](#)

[Saber](#)

[SaberCo-Simulation](#)

[SaberScope](#)

[Star-Hspice](#)

[Star-MS](#)

[Star-MTB](#)

[Star-RC](#)

[Star-Sim](#)

[Star-Time](#)

[LIBRARIES](#)

[TCAD](#)

Star-Hspice

High Accuracy Circuit Simulation

Star-Hspice is the industry standard for accuracy in circuit simulation and is used for sign-off by most of the world's IC foundries. In addition to giving designers the best possible assurance that their design will perform as predicted, Star-Hspice offers excellent convergence, so precise results can be obtained on all types of digital, analog and mixed-signal circuits. Star-Hspice provides the broadest selection of device models available in the industry and continuously maintains a lead position in modeling technology and implementation. The wide range of analysis features and supported hardware platforms make Star-Hspice the most widely used circuit simulator in the world.

STAR-HSPICE BENEFITS

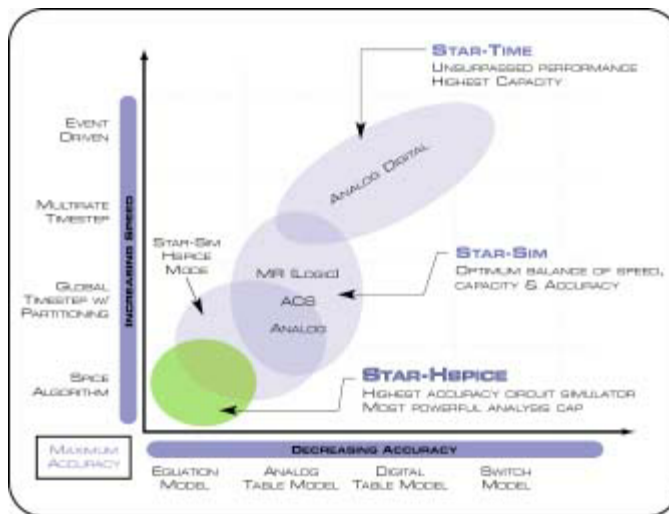
- Reduce time-to-market with Star-Hspice's unsurpassed reliability and accuracy.
- Verify and optimize designs with the widest range of analysis features.
- Increase design portability by calibrating circuits to the same standard used for sign-off by the world's major foundries.
- Optimize circuit performance with access to the broadest selection of state-of-the-art device models.
- Improve designer productivity by providing superior convergence algorithms for all circuit types and analyses.

In today's consumer-driven electronics market, the importance of accurate circuit simulation is increasing. The advent of systems-on-chip and the popularity of such technologies as wireless communications mean that analog circuits will be a growing percentage of all designs. Shrinking time-to-market windows leave no room for silicon iterations due to simulation inaccuracy.

Analog circuits must be analyzed across a broad range of performance factors such as gain, frequency response and noise performance. Meeting these specifications requires the use of many specialized analysis features. In addition, analog ICs appear in a variety of process technologies from bipolar and GaAs to CMOS. A simulator must incorporate the most up-to-date and accurate modeling techniques to provide designers with the flexibility to choose the best technology for their needs.

Integration of high-speed ICs at the system-level poses additional challenges for circuit simulation in the area of signal integrity. Modeling of on-chip drivers and receivers, accurate simulation of transmission line effects and analysis of phenomena such as crosstalk and ground bounce are critical to ensuring PC board performance.

The silicon-proven accuracy of Star-Hspice, together with its many sophisticated analysis capabilities and advanced process models have made it the most trusted circuit simulator in the world. It is widely used for many different applications, but is especially powerful for analog circuit design, cell characterization and signal integrity analysis, both for IC and PCB designs.



Avant! Transistor-Level Simulation

STAR-HSPICE CAPABILITIES

- Supports more than 40 device models, including public-domain (e.g. BSIM3 v3.2), customized proprietary models (Level 28) and many other MOSFET, BJT, JFET, diode & capacitance models.
- Multi-threaded capability for simulation speedup.
- Wide platform support-Sun Solaris, HP, IBM, SGI, Alpha, PC Windows 95/NT.
- Built-in flexible parameter sweeps in all analyses allows quick and thorough design characterization.
- Industry-standard measurement instrumentation enables batch-mode design analysis and documentation.
- Incorporates advanced features for signal integrity analysis, including frequency-dependent lossy transmission line models, IBIS models & built-in 2D field solver.
- Optimizes circuits for models and cells, with multi-parameter optimizations in AC, DC and transient simulations.
- Provides corners analysis and Monte Carlo yield analysis support.
- Efficient and powerful graphical analysis.

AVANT! CIRCUIT SIMULATION PRODUCT LINE

For circuits of any type that contain up to 50,000 transistors, Star-Hspice is unsurpassed for accurate circuit simulation. Moreover, simulation performance is significantly enhanced by the implementation of multi-threading for spreading the model evaluation task across multiple CPU

For many other simulation applications Avant! recommends Star-Sim, which handles larger circuits (up to 5 million circuit elements) with higher performance (10-1,000X faster than Star-Hspice) for timing and power simulation of mixed-signal and embedded memory blocks (accuracy within 5% Star-Hspice). For performing full-chip verification of large custom logic and memory chips, Avant! offers Star-Time, with a capacity of 50 million circuit elements and speeds 10,000X faster than Spice (timing accuracy within 5% of Spice). Finally, for verification of the largest SOC designs, Star-MS is Avant!'s next-generation mixed-signal simulator, incorporating Star-Sim analog and Polaris Verilog algorithms into a unique and highly efficient single kernel architecture.

THE INDUSTRY'S BEST MODEL SUPPORT

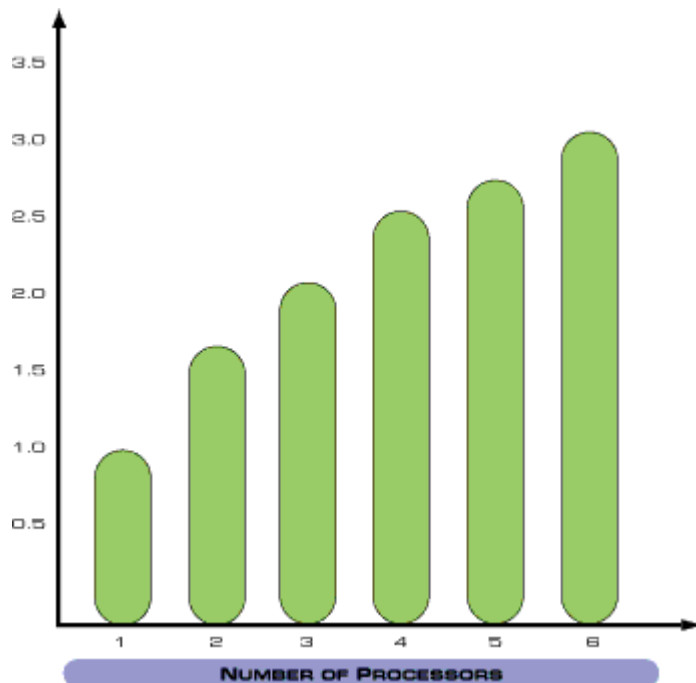
Star-Hspice continues its tradition of providing the broadest range of model types in the industry. Additions to Star-Hspice's model support include the latest public-domain BSIM3 v3.2 model, EPFL-EKV and UF-SOI models for low power applications, the VBIC-95 BJT model and Philips MOS9. Models can be incorporated into Star-Hspice using the Common Model Interface (CMI) option.

In addition to having access to highly accurate models, Star-Hspice users benefit from proprietary features that improve device model accuracy. Specifically, geometric overlays for model binning improve accuracy over the range of transistor channel lengths and widths. Further, capacitance calculations can be varied depending on geometric effects such as drain and source sharing.

EXPLORATION OF THE DESIGN SPACE IMPROVES CIRCUIT PERFORMANCE

Cell characterization is an ideal application for Star-Hspice. Star-Hspice's versatile measure statement allows you to quickly determine polynomial delay coefficients for timing simulators. A data-driven parameter sweep automates characterization by simultaneously varying any number of parameters, possibly replacing hundreds of Star-Hspice runs.

Exploring process corners and operating conditions is crucial for maximizing IC performance and yield. Star-Hspice incorporates sophisticated analyses such as Monte Carlo, worst-case corner and circuit optimization, which allow the user to center the design against various process, voltage and temperature ranges.



Star-Hspice Circuit Analysis Capabilities & Applications.



Multi-Thread Performance.

TOOLS FOR SIGNAL INTEGRITY ANALYSIS

For signal-integrity analysis of high-speed PC boards, Star-Hspice provides insight into problems such as ringing, ground bounce and crosstalk that can limit the operation of today's high-speed circuits. The *W* element, a multi-conductor, frequency-dependent transmission line model, is based on a novel state-of-the-art simulation method. A built-in 2-D field solver that automatically calculates the matrix coefficients based on process and geometry information further simplifies the use of the *W* element. Star-Hspice also incorporates the capability to directly simulate IBIS models. This allows IC vendors to protect their intellectual property while enabling system-level integrators to accurately analyze design performance.

GRAPHICAL ANALYSIS

To improve design productivity, take advantage of Avant!'s AvanWaves for analyzing and measuring Star-Hspice simulation results with an intuitive drag-and-drop user interface. AvanWaves provides built-in mathematical functions and user-definable expressions. The Star-viewing tool is also available for analyzing Star-Hspice transient results.

PLATFORMS & OPERATING SYSTEMS

- Sun SPARC: SunOS 4.1.4, Solaris 2.6