SILVACO

SmartSpice Circuit Simulator

SmartSpice™ Circuit Simulator delivers the highest performance and accuracy required to design complex analog circuits, analyze critical nets, characterize cell libraries, and verify analog mixed-signal designs. SmartSpice is compatible with popular analog design flows and foundry-supplied device models.

- 100% HSPICE™ compatible for netlists, models, analysis features, and results
- Provides the most accurate circuit simulation results for critical analog and nanometer effects
- Offers largest capacity of any true SPICE circuit simulator – up to 400 thousand active devices in 32 bit and 8 million active devices in 64 bit version
- Fastest run-time of any true SPICE circuit simulator and the only SPICE supporting multiple threads for parallel operation
- Robust convergence using multiple solvers and stepping algorithms
- Largest collection of calibrated SPICE models for traditional technologies (Bipolar, CMOS) and emerging technologies (TFT, SOI, HBT, FRAM, etc.)
- Provides open model development environment and extensive analog behavioral capability with Verilog-A option
- Supports Cadence analog flow through OASIS™

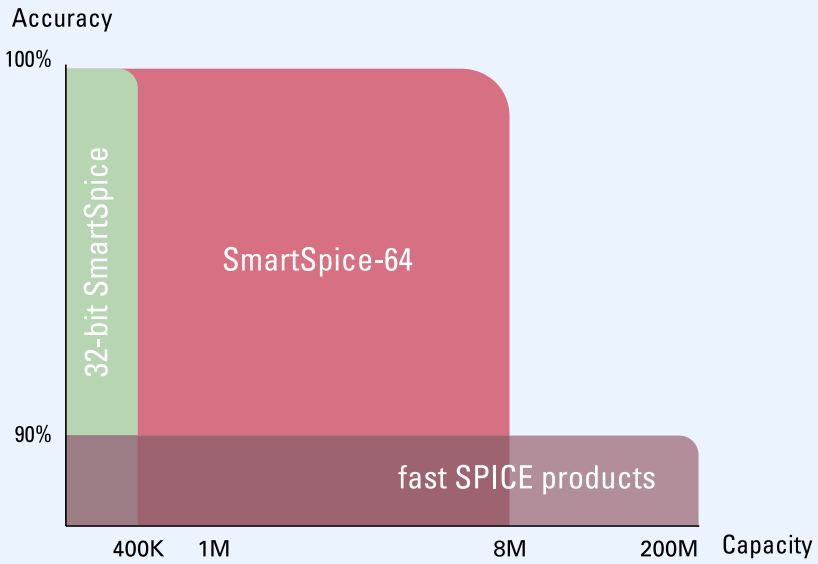
Accuracy: SmartSpice is the most accurate circuit simulator for critical analog and nanometer effects

- Uses Gaussian elimination in an efficient matrix-based on the original Berkeley 3C1 solver
- Includes multiple improved solvers that enhance the classic Newton-Raphson algorithm for the iterative solving of non-linear equations
- Verifies and validates Berkeley physics-based model parameters at run-time for continuity, linearity, and valid parameter range
- Detects inconsistencies in poorly-extracted foundry models and prevents these errors from degrading the final product performance and accuracy
- The simulator of choice at foundries that focus on analog and mixed-signal process accuracy
- Offers a full set of options for controlling speed accuracy of simulations

Capacity: SmartSpice simulates more active devices than any other analog circuit simulator

- Simulates circuits with more than 400K active devices (limited by addressable memory)
- SmartSpice-64 supports simulation of an unprecedented 8 million active devices with full SPICE accuracy on 64-bit workstations

SmartSpice-64 simulates up to 8 million active devices with true SPICE accuracy.



Speed: SmartSpice runs faster than other true SPICE simulators

- Simulates at 2 to 4 times the raw speed of other SPICE products
- Supports multiple parallel 32/64 bit CPUs for near-logarithmic multi-threaded operation

Convergence: SmartSpice selects the right solver for optimal convergence integrity, speed, and stability

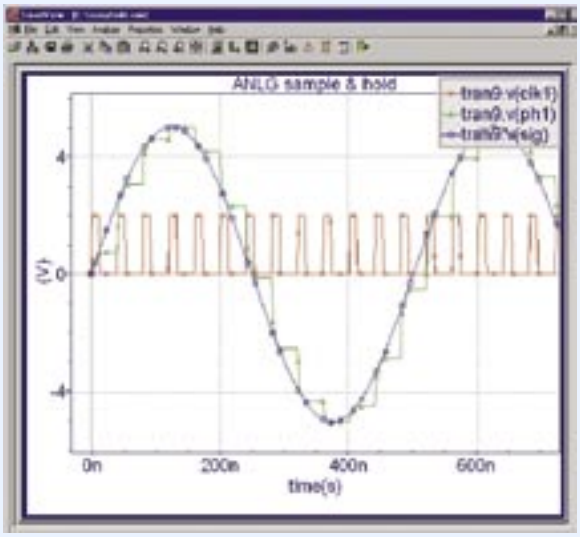
- Surveys initial conditions and iteratively sequences through a series of methods and algorithms to attain optimal convergence
- Selects from multiple solvers based on circuit topology to use the best solver for the circuit application
- Offers multiple options for controlling convergence

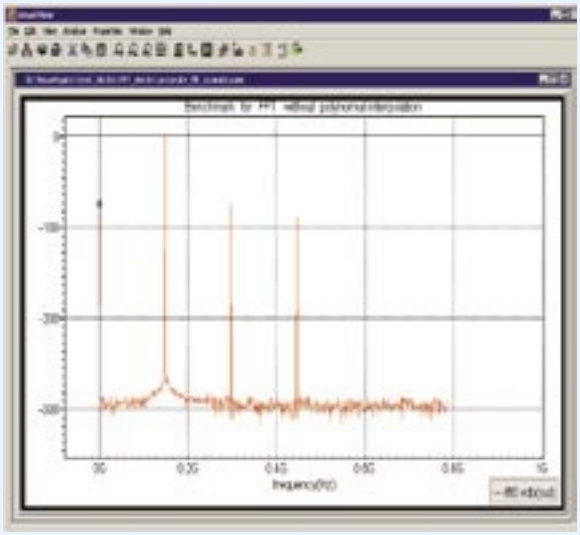
Ease of Adoption: SmartSpice fits your design flow and foundry models

- Supports foundry-supplied SmartSpice and HSPICE™ models
- Supports legacy netlists from HSPICE™, PSPICE™, and Berkeley SPICE
- Seamless integration with Cadence analog environment through OASIS™
- Total integration with Silvaco Custom IC CAD tool flow

Model Development Capabilities

- Core competence in SPICE modeling, data acquisition and model parameter extraction since 1984 with UTMOST™ for the highest accuracy in analog models
- Verilog-A models offer fastest method for implementing OVI standard electric-thermal models, sensor models, and other mixed physical effects
- "C" Interpreter enables modeling engineers to quickly interpret and debug models external to SmartSpice
- SmartSpice is first to deliver Berkeley models with proven BSIM4 solution correcting negative capacitance in original Berkeley implementation
- Silvaco offers accurate and prompt SPICE Modeling Services to extract DC, AC (S-parameters), capacitance, temperature, noise, and SPICE parameters over full temperature and corner models using statistical analysis

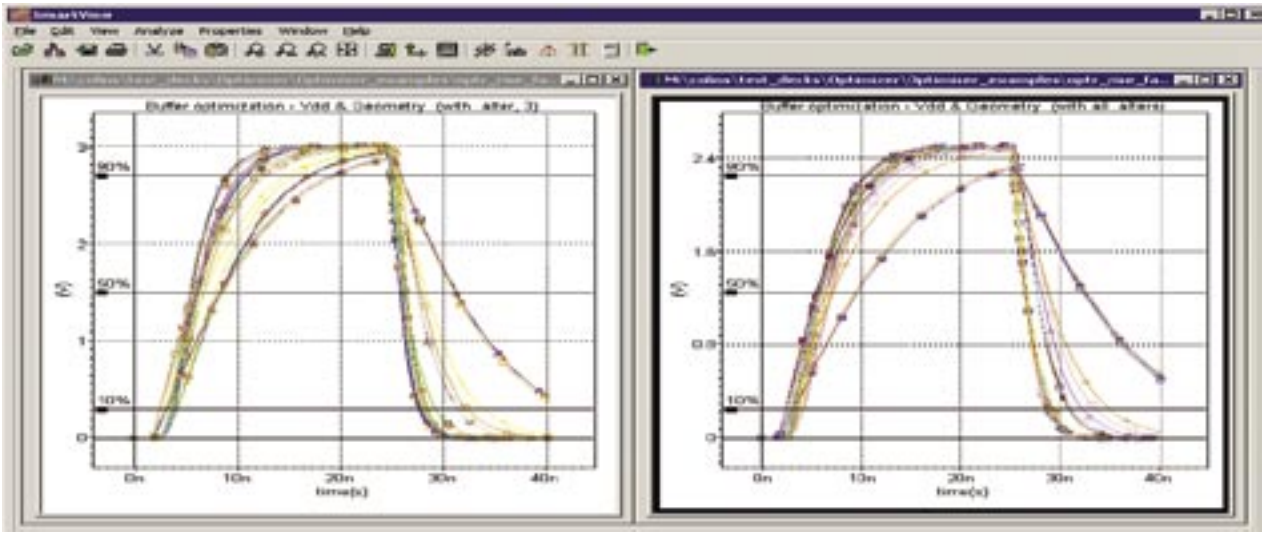




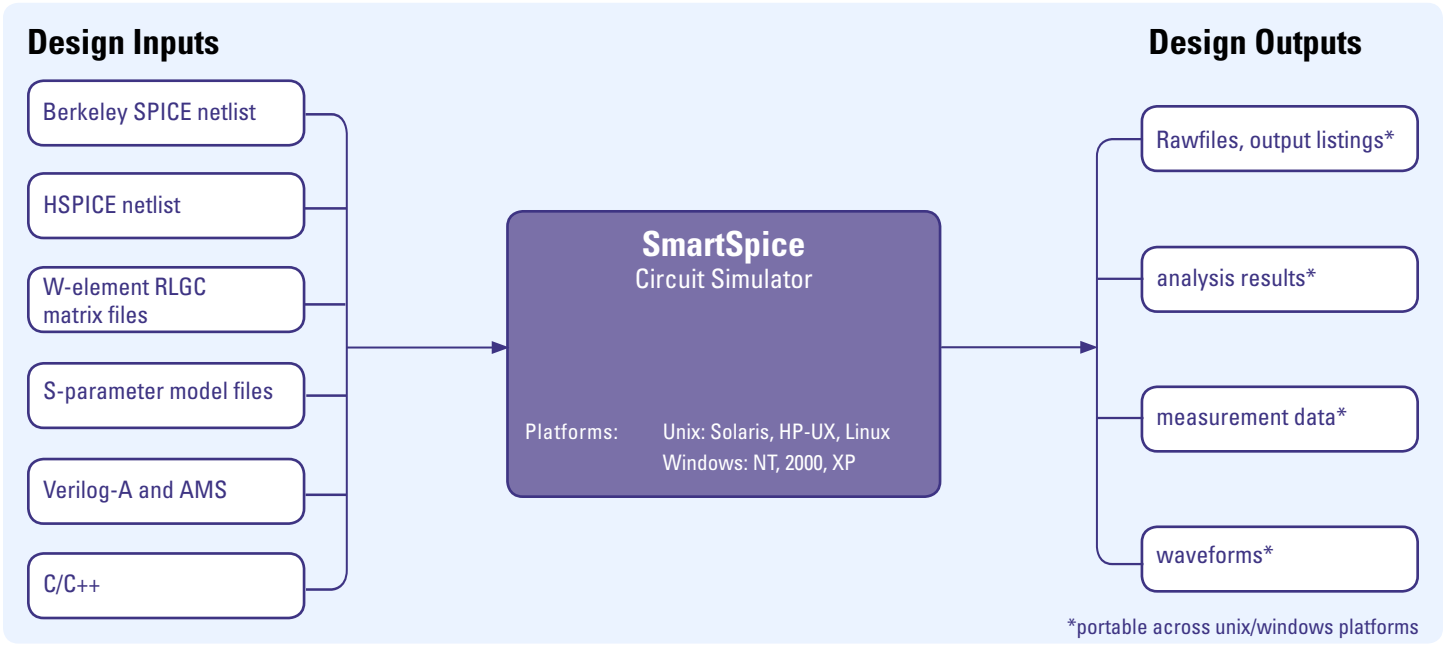
SmartView™: produces annotated plots and graphs of measurements of time, voltage, current, and power for rise time, slope, vector calculator, and eye diagrams from SmartSpice and HSPICE™ simulation results.

Models Available

BJT:	Philips Mextram/MODELLA, HiCUM, VBIC, Quasi-RC, IGBT, QBBJT, Modified Gummel-Poon
MOSFET:	Berkeley (Levels 1, 2, 3); BSIM (1, 3v3,4); Philips MOS (9,11), Hspice Level 28, EKV, HiSIM, HVMOS
TFT:	Amorphous TFT, Polysilicon TFT, Berkeley, Leroux, RPI
SOI:	Berkeley BSIM3SOI PD/DD/FD, STAG, UFS, LETISOI
HBT:	UCSD
MESFET:	Stats, Curtice I & II, TriQuint 1/2/3
JFET:	Berkeley Levels 1, 2
Diode:	Berkeley, Fowler-Nordheim, Philips JUNCAP/Level 500
FRAM:	Ramtron FCAP



Integrated Optimizer iterates device or model parameters to achieve target specifications in the form of DC, AC, transient curves, propagation delay, rise and fall times, power dissipation, etc.



SILVACO

USA Headquarters:

Silvaco International
4701 Patrick Henry Drive, Bldg. 2
Santa Clara, CA 95054 USA
Phone: 408-567-1000

Fax: 408-496-6080
sales@silvaco.com
www.silvaco.com

Worldwide:

Japan: jpsales@silvaco.com
Korea: krsales@silvaco.com
Taiwan: twsales@silvaco.com
Singapore: sgsales@silvaco.com
UK: uksales@silvaco.com
France: frsales@silvaco.com
Germany: desales@silvaco.com