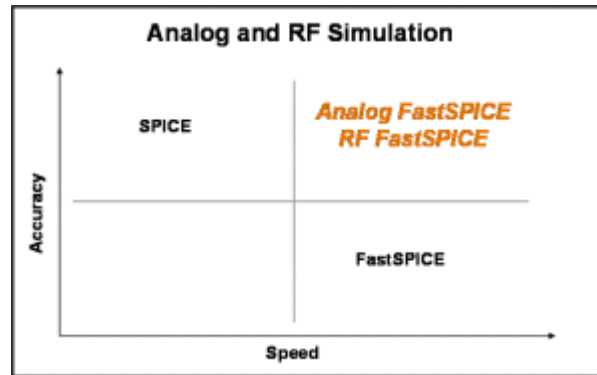


# Analog FastSPICE™



***True SPICE Accuracy, 5X-10X Faster, 5x-10x Higher Capacity for Analog/RF***

## Overview

*Analog FastSPICE™* uses Berkeley Design Automation's proprietary *Precision Circuit Analysis™* technology to deliver true SPICE accuracy, 5x-10x faster and with 5x-10x higher capacity, and extremely robust DC convergence compared to traditional SPICE tools – all without block-level tuning. It is ideal for any transient simulation that requires true SPICE accuracy and would take multiple hours or longer in a traditional SPICE simulator – including post-layout designs with up to 1 million or more elements. *Analog FastSPICE* has been proven on a wide range of production circuits including full 802.11 a/b/g transceivers, sigma-delta ADCs, automatic gain control circuits with bandgap and bias, high-speed I/Os, memories and memory interfaces, and multi-GHz PLLs and DLLs.

## Key Features

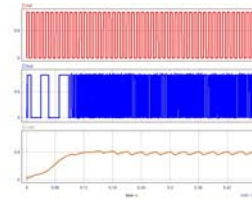
- True SPICE accuracy
- Robust DC analysis with unmatched convergence
- Fast transient analysis based on proprietary multi-rate engine
- AC analysis
- Linear noise analysis
- Supports standard netlist and model formats
- Integrated into the Cadence Virtuoso ADE environment

## Target Applications

- >1 GHz, nanometer-scale, analog and RF CMOS circuits
- RF transceivers (802.11, WiFi, UWB, GSM,...)
- High-speed I/O (SerDes, DDR, PCI-Express,...)
- Multiple-block (performance and functional)
- Post-layout circuits with 1M+ elements
- Extensive PVT corner and configuration verification

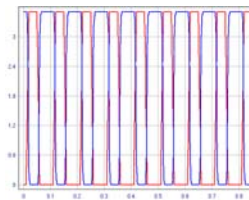
## Technology

*Analog FastSPICE* is based on *Precision Circuit Analysis* technology. This technology is a proprietary combination of innovative applied mathematics and optimized numerical analysis. Among the core components are a fast sparse matrix solver, globally convergent nonlinear solvers, and efficient stiff differential algebraic equation solver. Unlike digital fastSPICE simulators, *Analog FastSPICE* makes no simplifying assumptions or approximations. It solves the original device equations on the original circuit with accuracy that is equivalent to, and may even exceed, traditional SPICE simulators. This means that it needs no block-level tuning and always provides true accuracy at every node in the circuit; hence every simulation is a full performance simulation. The result is breakthrough technology that can address a whole new class of verification problems.

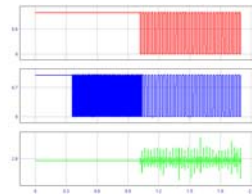


### *Industry's First and Only Multi-Rate Transient Engine*

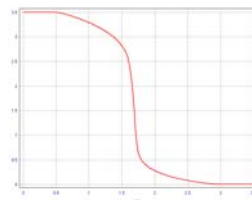
*Analog FastSPICE* utilizes a proprietary multi-rate transient engine to provide outstanding performance with true accuracy – and it never requires block-level tuning. This approach is vastly superior for multi-hour simulations that require true SPICE accuracy.



Traditional SPICE simulators rely on single-rate transient engines that become inefficient with increasing circuit complexity, number of frequencies, nonlinearity, and device model complexity. Digital fastSPICE simulators contain a single-rate transient engine that users must map and tune to semi-autonomous blocks in the circuit. To do so, users must thoroughly understand the design and the simulator controls, and run a number of time-consuming experiments before finding the best settings for trading off accuracy for speed. This approach is especially time consuming and inherently problematic for complex, tightly coupled circuitry. Digital fastSPICE inaccuracies typically run into the several percent range for even 3X to 5X higher performance. The resulting accuracy is good enough for functional verification at best.



By contrast, *Analog FastSPICE's* multi-rate transient engine treats the circuit as a whole just as SPICE simulators do, that is, by solving the original, entire matrix without any approximations. The mathematical foundations for the multi-rate transient engine deliver highly efficient solving of stiff differential algebraic equations. It makes no approximations. The multi-rate engine solves the same device equations as SPICE. It just does so in a fraction of the time. The result is true SPICE accuracy with an increasing performance advantage as the circuit size, number of frequencies, and frequency disparity increases.



### *Fast, Robust DC Analysis*

DC (or operating point) analysis plays an important role in analog/RF circuit analysis. Achieving DC convergence on complex,

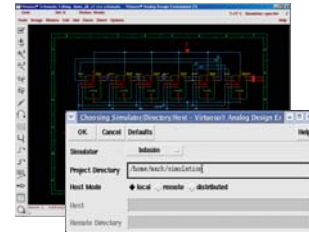
high speed analog and RF CMOS circuits using traditional SPICE simulators can be extremely unreliable and time consuming.

*Analog FastSPICE* includes an innovative, globally convergent solver that provides unprecedented dc analysis speed and reliability.

### Fits Your Existing Flow

*Analog FastSPICE* easily fits into existing HSPICE and Spectre-based verification flows, and supports popular device models.

*Analog FastSPICE* runs in both batch mode and interactive mode using the Cadence Virtuoso Analog Design Environment. Results can be viewed with BDA's waveform viewer, WaveCrave, or third party waveform viewers from Cadence, and Synopsys.



- Netlist compatibility
  - HSPICE<sup>®</sup>
  - Spectre<sup>®</sup>
- Model support
  - BSIM3, BSIM4, BSIMSOI, MOS1
  - Gummel-Poon, Mextram
  - HICUM, VBIC, Juncap
  - Verilog-A, s-parameter, w-element
- Outputs
  - PSF ASCII, PSF binary, tr0
  - Nutmeg ASCII, Nutbin, Nutbinf
- Integration
  - Comprehensive Cadence<sup>®</sup> Analog Design Environment (ADE) integration
- Waveform viewer
  - *WaveCrave*<sup>™</sup> waveform processor

### *Analog FastSPICE* Analyses

- DC (.op and sweep)\_AC, transient
- .measure for DC and transient, noise
- Network analysis, pole-zero, transfer
- Monte Carlo, alter, and sweep
- HDL co-simulation (beta)

### Platform Support

*Analog FastSPICE* is available for SUN and Linux OS.

#### **Berkeley Design Automation, Inc.**

2902 Stender Way, Santa Clara, CA 95054 USA

Phone: (408) 496-6600

Fax: (408) 496-6633

Email: [info@berkeley-da.com](mailto:info@berkeley-da.com)

Web Site: [www.berkeley-da.com](http://www.berkeley-da.com)

Berkeley Design is a registered trademark and Analog FastSPICE, RF FastSPICE, PLL Noise Analyzer, Stochastic Nonlinear Engine, and Precision Circuit Analysis are trademarks of Berkeley Design Automation.

HSPICE is a registered trademark of Synopsys, Inc. ADE and Spectre are registered trademarks of Cadence Design Systems, Inc.