

SmartSpice Pro

FastSPICE Circuit Simulator

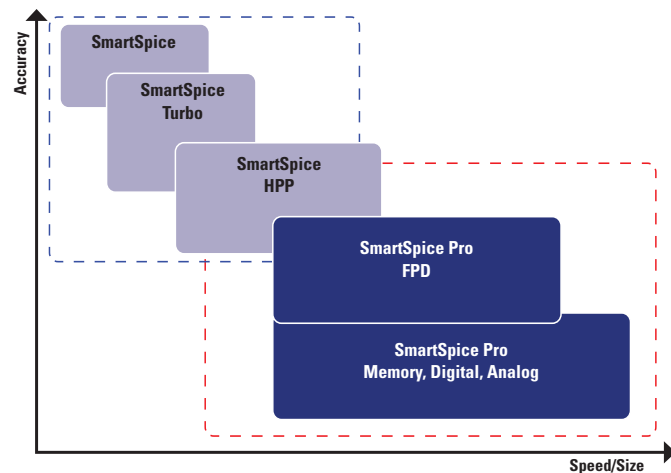
SILVACO

Overview

SmartSpice Pro delivers superior performance for SRAM and TFT designs. Unique fast parallel technology provides SPICE-accurate and fast simulation of multimillion element memory and display circuits. SmartSpice Pro is fully integrated with industry standard SmartSpice Analog Simulator to achieve maximum productivity. Advanced multirate simulation technology provides core functionality for transient circuit verification for designer.

Features

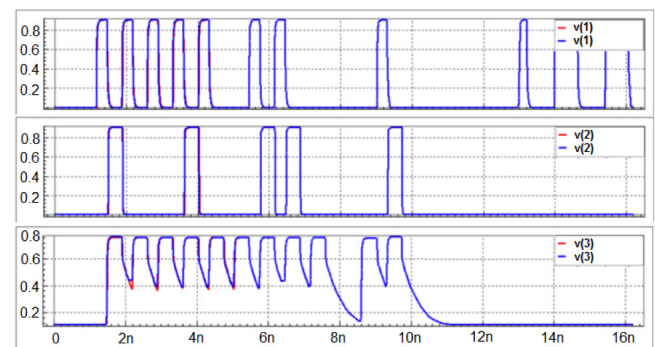
- HSPICE™ and Spectre™ compatible
- Provides accurate simulation using advanced multirate simulation technology
- Scalable fast parallel solver
- Integrated Verilog-A
- Support of advanced modeling technologies FinFET 7nm, 10nm, 16nm
- Integration with TSMC TMI modeling interface
- Designer oriented set of postprocessor functions for circuit sanity check
- Rich set of industry standard technology models for various areas (TFT display, memory and analog)



Accuracy vs. Speed for different SmartSpice simulators.

Accuracy

- Exploits traditional SPICE simulation algorithms for better accuracy
- Uses event-driven approach for communication between active and dormant parts of circuits
- Uses transient multirate simulation technology
- Uses adaptive timestep control algorithms and time grid mapping

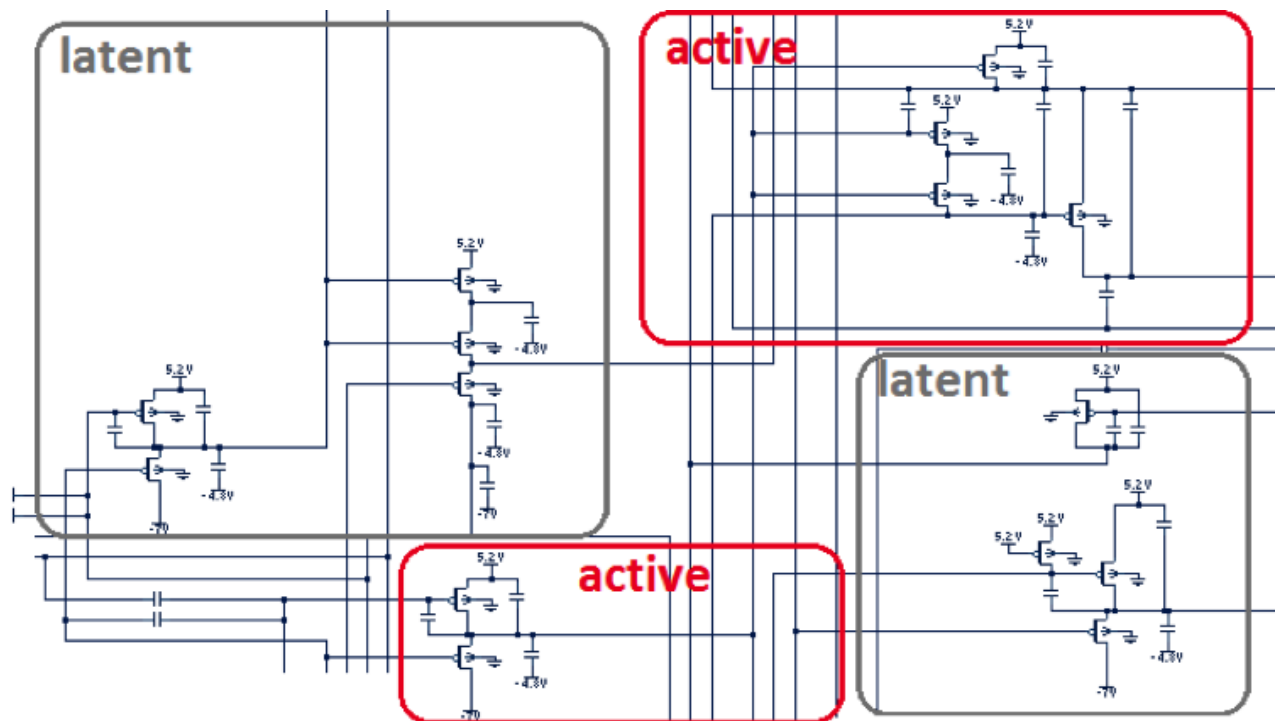


SmartSpice (red) vs SmartSpicePro (blue)

Waveform Overlay.

Simulation

- Support DC and transient analysis
- Support of block latency control
- Support block activity control
- Support of Event-driven resolution control
- Support of block coupling control
- Provides control over block solution control using Newton-Raphson solver
- Deliver composite set of options for accuracy vs. performance trade-off



Strongly connected components partitioning.

Partitioner

- MOS Channel Connected Components
- Automatic detection of power blocks
- Efficient handling of floating capacitors coupling
- Effective feedback loops simulation

Models

- Analytical models FinFET, BSIM3v3, BSIM4, TFT, BSIMSOI4, HiSIM,
- Lookup table BSIM4

Input

- SPICE/HSPICE/Spectre netlist, Verilog-A

Output

- FSDB waveforms
- Measurements
- Log file
- Analysis results