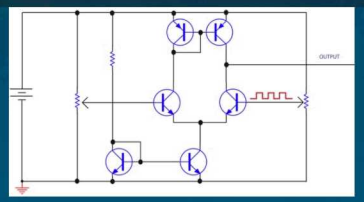
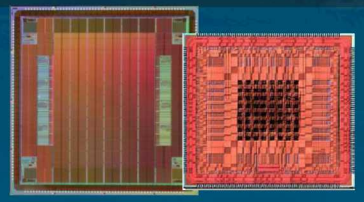
SAND2018-12739C

Xyce: Open Source Simulation for Large-Scale Circuits





PRESENTED BY

Jason C. Verley

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

The Xyce™ Parallel Electronic Simulator Analog Circuit Simulator

SPICE-Compatible syntax (Berkeley 3f5)

Two versions, **Serial** and…

Distributed Memory Parallel (MPI-based)

Unique solver algorithms

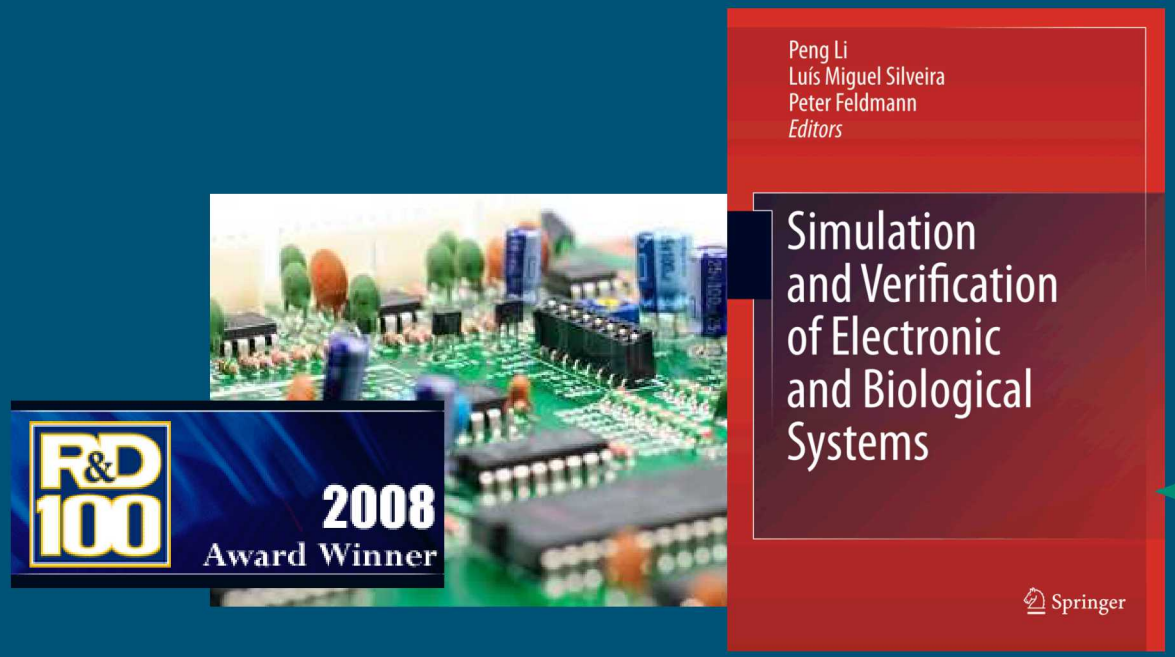
Industry standard models

Non-traditional models
- Neuron/synapse
- TCAD (PDE-based)

Open Source, GPLv3
- Since September of 2013 (Xyce 6.0)

Xyce Release 6.10
- November, 2018; 23rd major release
- >3,900 external downloads

http://xyce.sandia.gov



Keiter, et al., "Parallel Transistor-Level Circuit Simulation"

Xyce Capabilities

Typical

DC, Transient, AC, Noise
- .DC, .TRAN, .NOISE, .AC (and .STEP)

Post Processing:
- Fourier transform of transient output (.FOUR)
- Post-simulation calculation of simulation metrics (.MEASURE)

Output (.PRINT)
- Text Files (tab or comma delimited)
- Probe (PSPICE)
- Gnuplot, TecPlot, RAW (SPICE 3f5)

Analog Behavioral Modeling

Expressions, functions, parameterizations…

Others

Harmonic Balance Analysis (.HB)
- Steady state solution of nonlinear circuits in the frequency domain

Random Sampling Analysis
- Executes the primary analysis (.DC, .AC, .TRAN, etc.) inside a loop over randomly distributed parameters

Sensitivities
- Computes sensitivities for a user-specified objective function with respect to a user-specified list of circuit parameters ($\partial O/\partial p$…)
- DC or Transient
- E.g., an output voltage's dependence on a capacitance

Xyce-isms

Xyce defaults are conservative

- The industry standard is, "The simulator must never fail to provide an answer"
- …even if it's wrong.
- The Xyce philosophy: provide a numerically accurate answer, and fail if asked to do something "wrong."

Simulations in Serial vs. MPI Parallel

- Distributed parallelism can take more tuning:
  - device distribution, direct vs. iterative linear solvers,…
- Very large parallel simulations are "hard" (need to find the "right" linear solver)
- Leverages Sandia's Trilinos HPC solver framework

Xyce is the simulator (like HSPICE, SmartSpice, Spectre,…)

- There is no Schematic Design/Capture Front End (like Virtuoso, Tanner AMS, Gateway,… i.e., a GUI)
- …for now?

Others:

- Xyce is not (at the moment) 100% compatible with any other simulator
- There may be an expected feature that we don't (yet) support; e.g., .OP functionality is limited

Near-Term Work

Commercial Simulator Compatibility
- (netlist compatibility is often easier than feature compatibility; we are working on both)
- PSPICE netlist conversion tool (Sandia-only at the moment)
- HSPICE compatibility: under development (netlist and feature)
- Spectre compatibility: targeted for future development

PDK Support
- Strongly tied to simulator compatibility (HSPICE is the current path)
- Initially targeting Global Foundries 14 nm

Mixed-signal support (Verilog via VPI, VHDL via VHPI)

S-parameter analysis implementation

Build system moving to CMake

Performance Improvements
- Industry Standard Compact Models: speed improvements (Verilog-A compiler based on ADMS)
- Solver performance: solvers for parallel, choice of default parameters

Xyce Team Acknowledgements

Eric R. Keiter

Thomas V. Russo

Richard L. Schiek

Heidi K. Thornquist

Ting Mei

Jason C. Verley

Peter E. Sholander

Karthik V. Aadithya

…and many others

Contact:

`http://xyce.sandia.gov`
`xyce@sandia.gov`

Google Group Forum:

`https://groups.google.com/forum/#!forum/xyce-users`