

MSIM[®]

High-Accuracy Circuit Simulator for Analog/RF/Mixed-signal IC, LCD, Package and PCB Designs

MSIM is an industry-proven Spice simulator with optimized algorithms to deliver unparalleled accuracy, performance and value. TSMC has certified MSIM for accuracy and performance on advanced nanometer circuit designs.

Applications

- ◆ **Analog circuit design verification**
Frequency response and transient analysis for verifying analog circuits like PLL, A/D and D/A converters, amplifiers and IO devices etc.
- ◆ **Mixed-signal circuit design verification**
Support Verilog-A behavioral modeling, vector input stimulus and vector output verification (VEC and VCD).
- ◆ **RF design analysis**
Perform high-frequency nonlinear circuit simulation with large-signal analysis.
- ◆ **Cell library and memory characterization**
 - Integrate seamlessly with commercial cell library characterization tools, and demonstrate excellent performance.
 - Optimized for use with Legend's CharFlo-Memory! memory characterization tool.
- ◆ **LCD simulation**
 - Enable dynamic LC simulation with advanced TFT modeling for best accuracy.
 - Bridge the gap between optical outputs and electrical simulation.
- ◆ **PCB and Package signal integrity simulation**
 - Perform eye diagram analysis for high-speed designs.
 - Verify signal integrity issues such as jitter, cross talk and ground bounce etc.
 - Provide an integrated channel simulation of interconnects, passive components and active IO devices by using S-parameter macro-models and IBIS models etc.

Features

- ◆ **Ease of adoption**
 - Easily integrated into the existing design environments, including waveform analysis and zipped file inclusion.
 - MSIM simulation environment included for graphic user interface of input data, stimulus generation, simulation controls, results analysis and waveform viewing etc.

- ◆ **Advanced device modeling**
 - Deliver silicon-accurate models with proven implementations of BSIM3, BSIM4 and HiSim etc.
 - Develop Hybrid Modeling technology to ensure the built-in model matches the silicon data.
 - Provide Common Model Interface (CMI) which integrates users' own device models through dynamically linked shared library.
- ◆ **Co-simulation capabilities**
 - Combine Verilog-A behavioral descriptions with transistor-level netlists to speed up the simulation.
 - Execute both optical and electrical simulations with dynamic LC data and advanced TFT models.
- ◆ **Post-layout simulation**
Built-in RC reduction modules for enhancing the performance while maintaining high accuracy.
- ◆ **Versatility**
 - Enable multi-threaded simulations on multi-core computers, with outstanding efficiency especially on large circuits.
 - Optimized for the sweeping applications including parameterized analysis, model, instance and node for total turnaround time.
 - Intelligently handle subcircuit-style of device models to prevent massive memory usage and overwhelmingly lengthy simulation time.

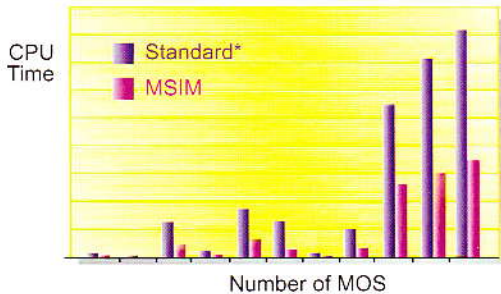
Benefits

- ◆ **High accuracy**
Certified by TSMC, and within 1% compared to the accurate mode of the most popular Spice simulator.
- ◆ **Fast speed**
More than twice the speed compared to the most popular Spice simulator.
- ◆ **Excellent convergence**
Intelligent solvers and multiple criteria provide efficient and reliable convergence.
- ◆ **Extensive model support**
Large collection of models support a full set of foundries and advanced technologies, and various applications such as LCD, IC package and PCB etc.
- ◆ **Best price performance**
Excellent value for a state of the art engine at a fraction of the cost.

Benchmarks

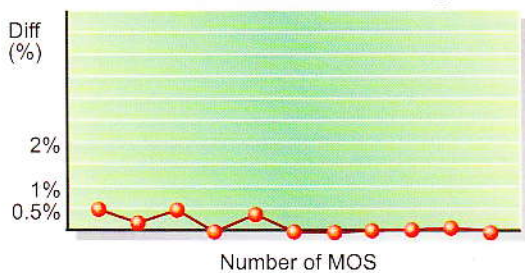
1. Performance Benchmark

MSIM is more than twice as fast as Standard*



2. Accuracy Benchmark

MSIM is as accurate as Standard* within a range of 0.5% difference.



3. Multi-Core Concurrency Benchmark

MSIM provides high efficiency multi-threaded simulations on multi-core computers as shown in below benchmark results on 2 quad-core CPU system

Threads #	Run Time (second)	Speed-up
1	319	1.0 X
2	171	1.87 X
4	97	3.29 X
6	68	4.69 X
8	57	5.60 X

4. RC-Reduction Benchmark

MSIM enhances the simulation speed on a SRAM circuit with 39,464 MOS, 200,526 Rs and 316,934 Cs, by performing AWE based RC reduction.

	Standard*	MSIM	Comparison
Accuracy	3.41 ns	3.39 ns	0.59 %
CPU Time	3,867sec	317 sec	12.2 X

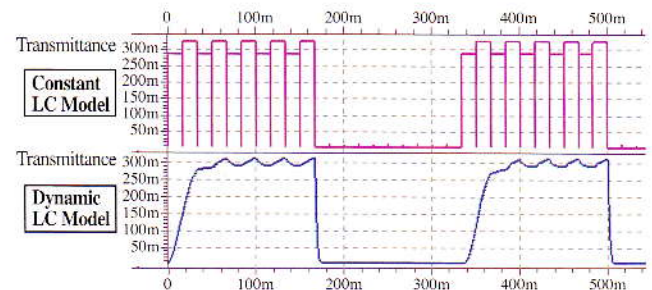
5. 'Subcircuit Model' Simulation Benchmark

By utilizing intelligent algorithms, MSIM minimizes memory usage and speeds up simulation for the circuit using about 10,000 MOS-FET models in subcircuit-style.

	Standard*	MSIM	Improvement
Memory Usage	6.7 GB	293 MB	22.9 X
CPU Time	2,294 sec	551 sec	4.2 X

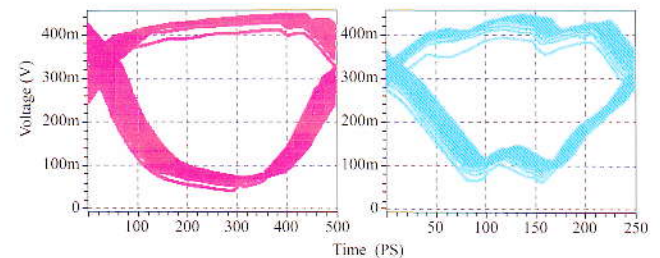
6. LCD Co-Simulation Benchmark

MSIM simulates the dynamic effects of transmittance to check for flickering and image-sticking issues on LCD panels.



7. PCB Eye Diagram Analysis Benchmark

MSIM performs eye diagram analysis for high-speed transmitter characterization and signal-integrity verification of PCB designs.



Specification

Design Inputs

- SPICE netlist
- HSPICE® compatible netlist

Device Models

- MOSFET models, including BSIM3 and BSIM4 and HiSim models
- BJT and Diode models
- Unified and compact capacitance models
- RPI and advanced TFT models
- S-parameter macro-models and IBIS models

Design Outputs

- WDF Waveform format
- FSDB Waveform format
- HSPICE® compatible Waveform and Measurement format
- ASCII text

Platforms

RedHat Enterprise Linux V3 and higher, and Window XP and higher.

* Standard means the most popular Spice simulator.

Legend
Design Technology

www.LegendDesign.com
email: sales@LegendDesign.com

Legend Design Technology, Inc.
Headquarter
2880 Lakeside Drive, Suite #101, Santa Clara, CA 95054
Tel: +1 (408) 748-8888 Fax: +1 (408) 748-8988

Taiwan Office
5F., No.176, Sec. 2, Gongdaowu Rd., Hsinchu300, Taiwan, R.O.C.
Tel: (+886) 36 111 888 Fax: (+886) 36 111 889