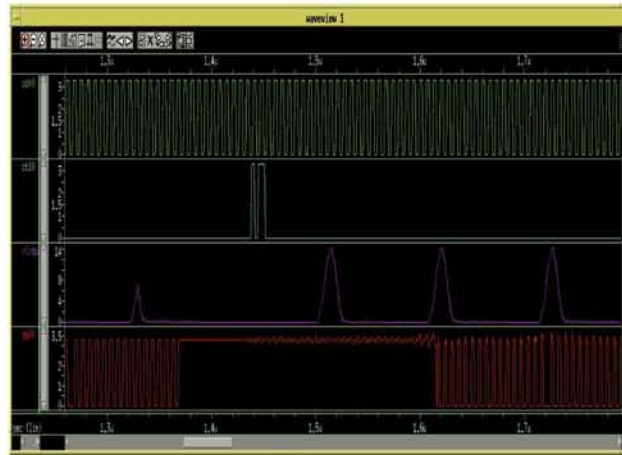


# Turbo-MSIM™

## High-Speed and High-Capacity Circuit Simulator



Turbo-MSIM™ Full-Chip Simulation Waveforms for a Mixed-Mode LCD Display Controller with Multi-Million MOSFETS

### Highlights

- Super high speed
- Extremely large capacity
- Exceptional accuracy
- Broad model support
  - Leakage current model
  - High voltage model
  - Foundry models
  - Nanometer technology
- Built-in RC reduction
- Excellent price-performance

Turbo-MSIM is a high-speed and high-capacity circuit simulator with SPICE-like accuracy. Turbo-MSIM is ideal for full-chip circuit simulations.

### Benefits

- **Accelerated simulation speed**  
Simulation speed is orders of magnitude faster than conventional Spice simulators.
- **Full-chip simulation capacity**  
Efficiently simulates full-chip designs, either hierarchical or flattened netlists.
- **Exceptional accuracy**  
Turbo-MSIM uses optimized time-based algorithms and proven models to deliver silicon-accurate results.
- **Scalable speed, capacity and accuracy**  
Achieves best-in-class throughput with scalable speed, capacity and accuracy.
- **Ultimate post-layout analysis**  
Enables speedy post-layout simulation and includes state-of-the-art RC reduction.

### Applications

- **Full-chip simulations for SoC designs**  
Provides the capacity for full-chip simulation, either hierarchical circuit or flattened layout-extracted netlist.
- **Functional verification**  
Supports transistor-level functional verification,

with vector input stimulus and vector output verification (VEC and VCD).

- **Timing and power analysis**  
Performs transient analysis for timing verification and accurate power simulation, especially for leakage power. Turbo-MSIM is extraordinarily accurate for circuits with multiple voltage sources.
- **Memory and mixed-mode designs**  
Proficiently recognizes repeated structures in memory circuits to optimize simulation throughput. Incorporates advanced algorithms and interfaces for mixed-mode designs.

### Features

- **Tabular device models for speed-up**  
Generates tabular device models from complicated analytical models. Table-lookup models facilitate much faster simulation speed.
- **Latency detection for throughput**  
Simulates only active circuits by latency detection to dramatically enhance speed.
- **Subcircuits and hierarchy rebuilding**  
Extracts sub-circuits and re-build hierarchy from flattened circuit for speed and memory efficiency.
- **Advanced RC reduction**  
Built-in RC reduction modules are used to enhance the performance by reducing the size of circuit matrix during DC and transient iterations.

## Benchmark Results

### 1. Speed, Capacity and Accuracy Benchmark

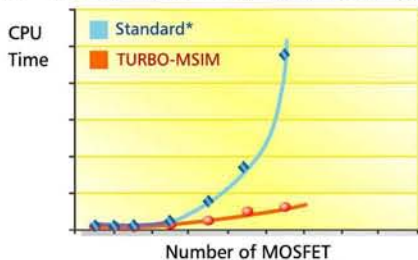
'Access time' simulations on a 0.13um high-speed SRAM circuits with MOSFET counts ranging from 31,741 to 6,395,298. If taking HSPICE (accurate mode) as the gold, the difference of Turbo-MSIM (accurate mode) is less than 0.5% and the difference of Turbo-MSIM (default mode) is less than 1.5%.

Circuit Type	MOS Count	Spice CPU Time	Turbo-MSIM CPU Time	Turbo-MSIM Speed Up	Accuracy Difference
Circuit 1	31,741	11,707 sec	37 sec	316 X	0.66 %
Circuit 2	57,079	21,483 sec	46 sec	467 X	1.50 %
Circuit 3	110,567	73,355 sec	78 sec	940 X	0.37 %
Circuit 4	1,619,735	N/A	756 sec	N/A	N/A
Circuit 5	3,198,065	N/A	1,636 sec	N/A	N/A
Circuit 6	6,395,298	N/A	5,725 sec	N/A	N/A

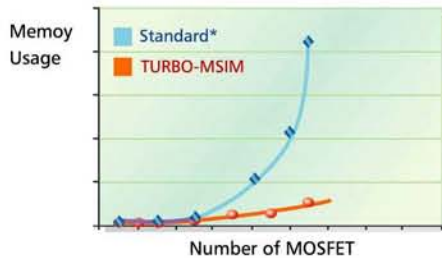
### 2. Performance Benchmark

Simulations have been run on different size circuits with the following results

- Turbo-MSIM is exponentially faster than the Standard\*

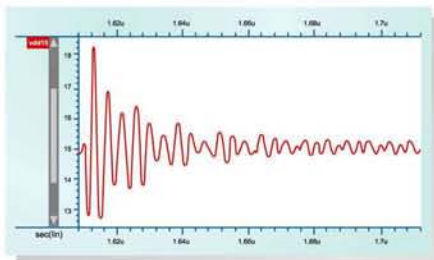


- Turbo-MSIM uses exponentially smaller memory than the Standard\*



\* Standard means the most popular Spice simulator

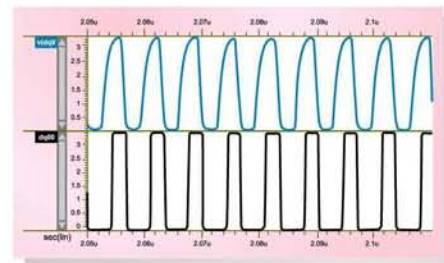
### 3. Ground Bounce Simulation Benchmark



Turbo-MSIM performs full-chip simulation and is used to analyze ground bounce effects with superior accuracy. This helps minimize the risk associated with ground bounce on real silicon.

### 4. Full-chip Functional Verification

Turbo-MSIM performs transistor-level full-chip simulation to verify the results with the expected outputs in vector form. The comparison between simulation results and expected output vectors are shown below



## Specification

### Design Inputs

- SPICE netlist
- HSPICE™ compatible netlist

### Device Models

- MOSFET models, including BSIM3 and BSIM4 models
- Bipolar junction transistor (BJT) models
- Diode models
- Unified and compact capacitance models
- Charge conservation and high voltage models

### Design Outputs

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

### Platforms

Sun Solaris, HP-UX and Linux (RedHat 7.3 and higher)

**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  
5Fl., No. 29, Alley18, Lane 81, Jianguang 2nd Rd., Hsinchu Taiwan, R.O.C.  
Tel: (+886) 36 111 888 Fax: (+886) 36 111 889