SISMA: SIMULATOR FOR STATISTICAL MISMATCH ANALYSIS

Giorgio Biagetti, Simone Orcioni, Alessandro Curzi, Paolo Crippa, Claudio Turchetti

DEIT - Università Politecnica delle Marche, I-60131 Ancona, Italy

`http://www.sisma.deit.univpm.it/` — `sisma@deit.univpm.it`

ABSTRACT

SiSMA (Simulator for Statistical Mismatch Analysis) is a CAD tool for the statistical analysis of analog CMOS integrated circuits affected by technological tolerance effects including device mismatch.

1. INTRODUCTION

High-performance analog, digital, and mixed-signal ICs are currently utilizing sub-100 nm fabrication processes. Unfortunately, the scaling of process tolerances has progressed more slowly than the scaling of feature sizes, so that nanoscale devices exhibit ever more increasing performance variations.

Because of this, as reported in ITRS2005 [1], "In less than five years (between 2008 and 2010) statistical methods will be completely embedded in the design flow. In the meantime, they will be selectively applied as they mature, or will be part of premium design technologies ..."

The tool presented here is aimed at helping designers maximize yield of the designed circuits by providing them with easy-to-use statistical models, and with a statistical simulation technique which is orders of magnitude faster than other readily available methods, e.g. Monte Carlo (MC). The proposed technique is based on direct statistical simulation (i.e. a non-MC approach), and is the core of the SiSMA tool [2, 3].

2. THEORY OF OPERATION

Statistical analysis is performed by modeling the electrical effects of tolerances by means of stochastic current or voltage sources. They alter the behavior of both linear and non-linear components according to predefined stochastic models, which depend on both device geometry, through a shape-factor $g(\cdot)$, and spatial localization on the die, through a bi-dimensional stochastic process $\gamma(x, y)$ described by a user-adjustable covariance function.

As an example, for MOSFETs we put a stochastic source $\zeta = I_D\,\eta$, where $\eta = g(W, L)\,\gamma(x, y)$, in parallel with the nominal I_D. Such functions are determined by technological process fitting parameters and are totally customizable by the end user. This model lets SiSMA take into account the relative distances between devices when performing statistical analyses.

The tool is able to perform DC, AC, and transient analyses, and is based on the circuit equations

$$\dot{\mathbf{q}}(\mathbf{x}) + \mathbf{k}(\mathbf{x}) + \mathbf{f}(\mathbf{x}, \boldsymbol{\eta}) + \mathbf{n}(t) = \mathbf{0} \qquad (1)$$

derived from the modified nodal analysis (MNA), where $\mathbf{q}$ represents reactive components, $\mathbf{k}$ instantaneous components, and $\mathbf{n}$ the excitations, just like in ordinary MNA. In addition to these, (1) adds a stochastic term $\mathbf{f}(\mathbf{x}, \boldsymbol{\eta})$ that depends on both circuit bias and a random variable vector $\boldsymbol{\eta}$.

Let us consider the variation $\boldsymbol{\xi}(t) = \mathbf{x}(t) - \mathbf{x}_0(t)$ around a generic "nominal" solution $\mathbf{x}_0(t) = \mathbf{x}(\boldsymbol{\eta}_0, t)$ as the unknown in (1). The main goal of statistical analysis is to determine the covariance matrix $\mathbf{C}_{\boldsymbol{\xi\xi}}$, that gives enough information for practical applications. To this end, by linearization of the relation between $\boldsymbol{\xi}(t)$ and $\boldsymbol{\eta}$, we get:

$$\boldsymbol{\xi}(\cdot) = \mathbf{H}(\cdot)\,\boldsymbol{\eta} \qquad (2)$$

so that it results:

$$\mathbf{C}_{\boldsymbol{\xi\xi}} = \mathbf{H}(\cdot)\,\mathbf{C}_{\boldsymbol{\eta\eta}}\,\mathbf{H}^{\mathrm{H}}(\cdot) \qquad (3)$$

where $\mathbf{C}_{\boldsymbol{\eta\eta}}$ is the covariance matrix of the random variables η.

According to the desired accuracy and the degree of non-linearity which the tolerances are able to trigger, linearization can be done either once (around the nominal value) or on a few automatically selected points (thus requiring a few SPICE runs). In the latter case, by denoting with $(\cdot)^{(m)}$ the variables pertaining to each linearization, the global solution can be found as:

$$\mathbf{C}_{\boldsymbol{\xi\xi}} = \sum_{m=1}^{M} P^{(m)} \left[\mathbf{C}_{\boldsymbol{\xi\xi}}^{(m)} + \left(\boldsymbol{\mu}_{\boldsymbol{\xi}}^{(m)} - \boldsymbol{\mu}_{\boldsymbol{\xi}} \right)^{\#2} \right] \qquad (4)$$

where the symbol $(\cdot)^{\#2}$ denotes matrix multiplication by its own adjoint, i.e. $(\cdot)^{\#2} = (\cdot)\,(\cdot)^T$, $\boldsymbol{\mu}_{\boldsymbol{\xi}}$ is the mean vector, and $P^{(m)}$ is an appropriate weighting factor.

3. WORKING ENVIRONMENT AND EXAMPLES

The SiSMA tool has been developed with the goal of achieving maximum portability and efficiency. It is written in C++ and runs under Sun® Solaris™, various flavors of Linux®, and Microsoft® Windows™. It is splitted into two separate tools: a simulation tool and a graphical front-end, a screenshot of which is reported in Fig. 1.

SiSMA can easily be integrated in a design flow environment to interact with the most popular deterministic circuit simulators (e.g. SPICE, Cadence® Spectre™ and PSPICE™), and also implements a more traditional Monte Carlo simulation method, which has been extended to take full advantage of the same position-dependent stochastic models. Several representative circuits have been selected to demonstrate the tool capabilities. As an example, we report the results of a temperature DC analysis of a band-gap reference (BGR, Figs. 2–4) and a transient analysis of a CMOS charge pump

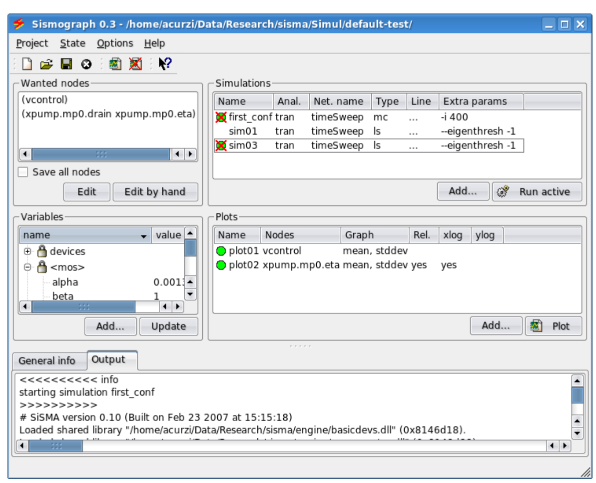

Fig. 1. Working Environment

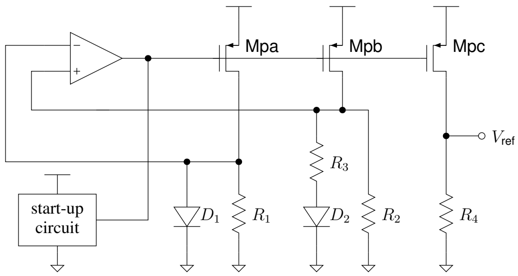

Fig. 2. Current-Mode Band-Gap Reference (BGR).

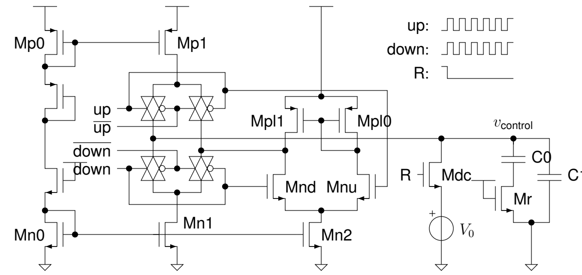

Fig. 5. Charge Pump (CP).

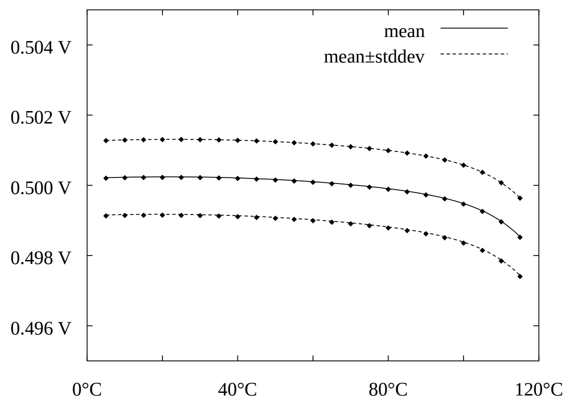

Fig. 3. Output V_{ref} vs. temperature. Dots represent MC simulation.

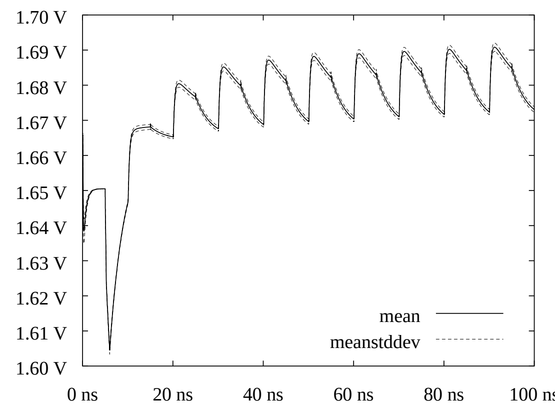

Fig. 6. Transient response of V_{control}: mean $\pm$ standard deviation.

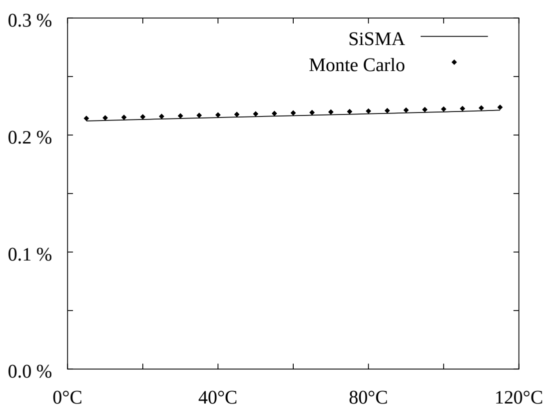

Fig. 4. BGR: Relative standard deviation. Dots represent MC simulation.

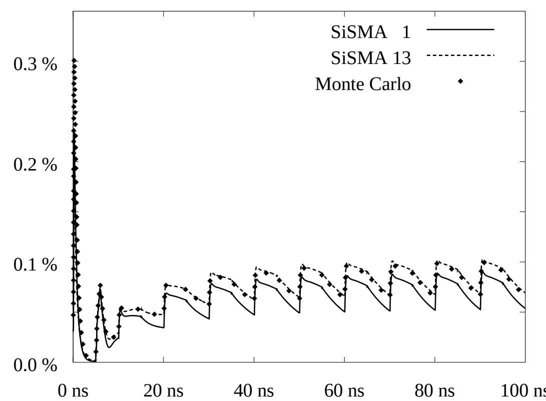

Fig. 7. CP: Relative standard deviation. Dots represent MC simulation.

(CP, Figs. 5–7), performed with both the single (CP 1) and the multiple linearization method with $M = 13$ points (CP 13). Table 1 summarizes execution times and accuracy with respect to a reference MC simulation.

Circuit	SiSMA time	MC iters.	MC time	speed gain	error
BGR	24 s	3000	32400 s	1350	~ 1 %
CP 1	1305 s	3000	306000 s	234	~ 8 %
CP 13	—	3000	—	24	< 1 %

Table 1. SiSMA performance and accuracy.

4. CONCLUSIONS

SiSMA requires a simulation time several orders of magnitude lower than that required by Monte Carlo analysis, while ensuring a good accuracy. Therefore, statistical simulation can now be performed throughout the design flow without incurring in long delays, thus helping designers working with a tight time-to-market constraint to exploit better design alternatives.

5. REFERENCES

[1] *ITRS, 2005 Edition: Design*. International Technology Roadmap for Semiconductors, Dec. 2005.

[2] G. Biagetti, S. Orcioni, L. Signoracci, C. Turchetti, P. Crippa, and M. Alessandrini, "SiSMA: A statistical simulator for mismatch analysis of MOS ICs," in *Dig. 20th IEEE/ACM Int. Conf. Computer Aided Design (ICCAD 2002)*, San Jose, CA, Nov. 2002, pp. 490–496.

[3] G. Biagetti, S. Orcioni, C. Turchetti, P. Crippa, and M. Alessandrini, "SiSMA—a tool for efficient analysis of analog CMOS integrated circuits affected by device mismatch," *IEEE Trans. Computer-Aided Design*, vol. 23, no. 2, pp. 192–207, Feb. 2004.