Monte Carlo and Corner Simulations for Analog VLSI Design

Fernando Paixão Cortes, Alessandro Girardi, Felipe Ródio, Eric Fabris, Sergio Bampi
{fpcortes, girardi, fcrodio, fabris, bampi}@inf.ufrgs.br

Instituto de Informática – Universidade Federal do Rio Grande do Sul (UFRGS)
Caixa Postal 15.064 – 91.501-970 – Porto Alegre – RS – Brazil

ABSTRACT

The design of an analog circuit depends on several factors such as good device modeling and technology characterization. In this context, the electrical simulations used to predict the performance of circuits must use models that characterize the device technology. However these models must also consider that the process variables are subjected to random variations that affect the performance of any device. Most of this paper addresses the analysis of an amplifier, considering corners (extreme variations expected in the process, voltage and temperature values) and statistical (Monte Carlo) variations. A Miller operational amplifier designed in AMS0.35μm technology is analyzed using the Spectre simulator tool in the CADENCE environment. Electrical simulations of the circuit's performance are presented.

1. INTRODUCTION

Analog integrated circuit design procedure is based on meeting several design constraints that are closely related to transistor sizing and technology dependent factors. This leads to an important task: using electrical simulation to predict the performance of the circuit. The electrical simulation is accomplished by the use of models that characterize the device technology using extracted electrical parameters. These parameters are supplied by the foundry for a specific model, and have exact values. However, the process variables are subjected to random variations that affect the performance of the related electrical parameters. When the analysis considers the most extreme variations expected in the process, voltage and temperature values (the corners), it is called corners simulation. The statistical simulation (Monte Carlo analysis) includes device models that are assigned statistically with the variations of the parameters values.

This work focuses on the statistical and corners analysis of an analog block using the Spectre simulator tool in the Cadence environment. A Miller operational amplifier designed in AMS0.35μm technology is analyzed. Electrical simulations of the circuit's performance are presented, considering corners and statistical variations.

2. STATISTICAL ANALOG CIRCUIT ANALYSIS

In a real VLSI manufacturing process, variables are subjected to random fluctuations that affect the circuit performance. Due to the analog circuit design dependence of device technology, these variations must be considered to foresee correctly the expected results. In this context, corners and Monte Carlo analysis are very powerful tools for predicting these variations in analog circuit design. These two types of analysis are briefly described in the following sections.

2.1. Corner Analysis

In electrical simulation, corner analysis is a simulation that considers the most extreme variations expected in process, supply voltage and temperature values (the corners). With this information, it is possible to determine whether the circuit performance specifications will be met, even when the random process variations combine in their most unfavorable patterns.

A set of models is used to verify the characteristics of a circuit at the extremes of devices variations. These models are composed by electrical parameters extracted considering the circuit operating at extreme conditions, such as worst case in power and worst case in speed operations. Corner simulation performs circuit analysis for each corner (model, supply voltage and temperature values).

2.2. Monte Carlo Analysis

Numerical methods that are known as Monte Carlo methods can be loosely described as statistical simulation methods that utilize sequences of random numbers to perform the simulation. In analog VLSI design, Monte Carlo analysis is performed to model the process statistical variations that affect the circuit performance. This analysis allows the investigation of process spread and also device mismatch. For this purpose, each electrical parameter is assumed to have a continuous probability distribution function (*pdf*). The type of each *pdf* is based on the conditions surrounding that variable and can be normal (gaussian), uniform, exponential, triangular, etc. [2]

A Monte Carlo simulation generates several random samples of the parameter vectors and performs the desired circuit analysis for each sample. Then, the distribution of performance variables can be analyzed by means of descriptive statistics (mean, standard deviation, etc.), as well as by means of graphical statistical tools (histograms, scatter plots, etc.).

3. CASE STUDY: STATISTICAL SIMULATION OF A MILLER AMPLIFIER

In order to exemplify the statistical analysis described before, we will use the Miller amplifier designed in [3]. The schematics are shown in figure 1. The circuit was implemented in the CADENCE environment [1] for the AMS0.35μm technology. Electrical simulations of the circuit's performance are presented, considering corners and Monte Carlo variations. The simulations were realized in the Spectre simulator tool using the MOSFET BSIM3v3.2.2 model.

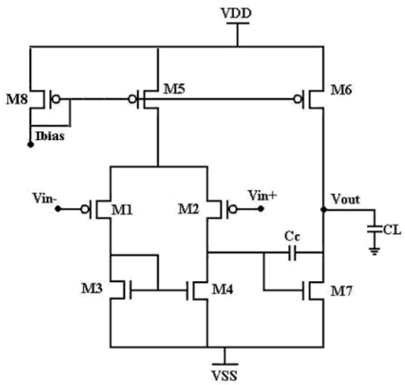

Figure 1 - Miller amplifier schematics

3.1. Corner Analysis Results

Table 1 shows the corners simulations obtained for the Miller amplifier, considering extreme variations expected in supply voltage (± 10%) and temperature values (industrial range – 0°C to 80°C), using the typical electrical model. The analysis shows small performance variations of DC gain (A_{v0}), phase margin (PM), gain-bandwidth product (*GBW*), slew rate (SR) and supply current (I_{DD}). The power dissipation achieved by the circuit increases with the operation temperature and supply voltage, as expected.

Table 1 - Corners simulations - Miller Amplifier

	Temperature (°C)			Supply Voltage (V)	
	0	25	80	+10%	-10%
Av (dB)	92.44	92.01	91.28	92.98	90.19
F_{-3dB} (Hz)	377.7	378.2	376.6	346.9	469.1
PM (°)	55.61	55.29	54.24	54.35	54.35
GBW (MHz)	15.88	15.10	13.83	15.49	15.20
SR (V/μs)	15.55	15.48	15.33	15.50	14.60
I_{DD} (μA)	324.7	325.4	326.8	327.7	323.1

Corners simulations using the worst case power and worst case speed model were also realized. The analysis showed small performance variations of the circuit performance.

3.1. Monte Carlo Analysis Results

The Monte Carlo simulation of the Miller amplifier is presented here. The analysis was set to perform 100 iterations of the circuit simulation, considering both process and mismatch statistical variations. The analysis procedure is illustrated here aiming the performances in current mismatch of the input differential pair and offset voltage.

Figure 2 shows the histogram of the distribution during the statistical analysis for the performance of offset voltage, obtaining a variation of ±7mV. The results obtained show the dependence of the circuit performance (offset voltage) on the mismatch variations of both input differential pair and current mirror (pair transistors M1-M2 and M3-M4 of figure 1).

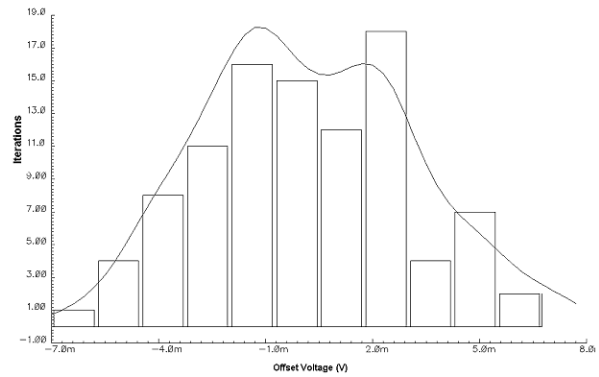

Figure 2 - Miller amplifier offset voltage distribution during the statistical analysis run – process and mismatch statistical variations

In order to improve the offset voltage response, the designer must increase the size (maintaining the same W/L of each transistor) of both input differential pair and current mirror. However, this results in more area. So, the designer must find the best solution, in order to obtain the lower total area (differential pair and current mirror sizes).

4. CONCLUSIONS

Statistical simulations are fundamental for the complete pre-production analysis of an analog integrated circuit. This paper presented corner and Monte Carlo simulations of a Miller amplifier, showing that random variation of process and mismatch parameters can modify the overall circuit performance. However, as in all simulated systems, there is always an error margin. Only test chips measurements can determine the real circuit functionality.

5. REFERENCES

[1] Cadence Design Systems. Spectre On-Line Help. 2003.

[2] Monte Carlo Simulations Web-Page. www.montecarlosimulations.org. Accessed in June 2003.

[3] Paixão Cortes, Fernando. "Modelagem e Projeto de Módulos Amplificadores e Comparadores em Tecnologia CMOS 0,35um". Master Dissertation. Porto Alegre: CPGCC da UFRGS, 2003.

[4] Rencher, Mark. "Analog Statistical Simulation". IEEE Custom Integrated Circuits Conference, 1991