278 *SIMULATION OF SEMICONDUCTOR DEVICES AND PROCESSES Vol. 6*
Edited by H. Ryssel, P. Pichler - September 1995

Influence of Analytical MOSFET Model Quality on Analog Circuit Simulation

M. Miura-Mattausch, A. Rahm, and O. Prigge

Corporate Research and Development, Siemens AG, 81730 Munich, Germany

Abstract

Though the importance of CAD increases, the analog circuit design is still mostly done by experience. It is known that this is because of insufficient model quality, or quality of extracted parameter values. We have developed a new MOSFET model based on the drift-diffusion approximation. By comparing a conventional piece-wise model with our precise model, critical shortage of the conventional models, which restricts the application of CAD for analog circuits, is demonstrated.

1. Introduction

Different from digital circuits, analog circuit performances are very much dependent on small signal conductances [1]. Describing these values correctly is a key in predicting the circuit performances correctly. However, these small signal values are sensitive to small deviations of applied voltages and also to small deviations of technological values. This makes prediction difficult.

Simplified piece-wise analytical MOSFET models have been developed and mostly used for circuit simulation. The main simplification of the models is the drift approximation neglecting the diffusion contribution. This simplification violates smooth transitions to different applied voltage regions. To get smooth transitions some fitting parameters are investigated [2]. To describe the saturation behavior the channel length modulation concept is introduced modeling with additional fitting parameters. Thus the piece-wise models describe transistor characteristics empirically, exactly where analog circuit performances are determined.

Circuit Performance with Our Model

Our model is based on the drift-diffusion model under the charge-sheet approximation [3]. All equations needed for circuit simulation are described analytically as functions of surface potentials at the source side and the drain side, which are solved iteratively during the circuit iteration. The key idea of our model development is to put as much as physics into the equations describing these surface potentials. Due to the inclusion of the lateral electric field in the equations, resulting surface potential values are dependent not only on applied voltages but also on channel length. It has been shown that our precise MOSFET model, in spite of the two additional iteration

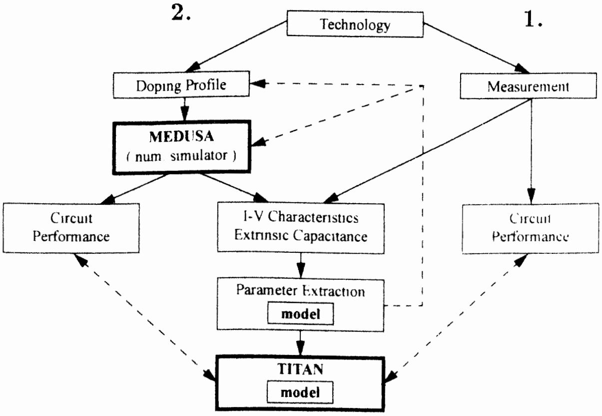

Figure 1: Flow chart of our overall simulator. TITAN is our circuit simulator.

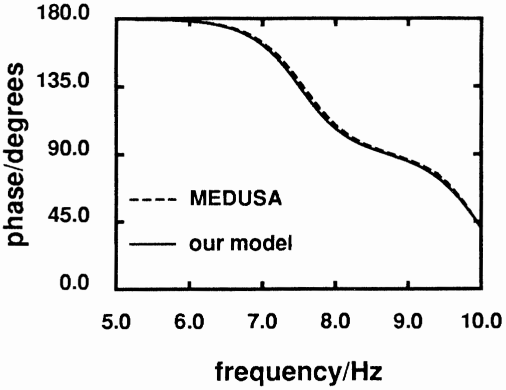

Figure 2: Comparison of the simulated phase of a linear amplifier with our model and MEDUSA.

procedures, reduces calculation time drastically in comparison with a conventional model.

As shown in Fig. 1 there are two approaches to verify the model quality. One is to produce chips with a certain technology and measure their $I - V$ characteristics in order to perform parameter extraction for circuit simulation.

The simulated circuit performances are compared with measurements. The other possibility is to use a numerical simulator in stead of measurements. We have done first the second approach. An advantage of this second approach is that we can know the transistor structure exactly even the doping profile, which can be used further for studying the influence of technological deviations on circuit performances.

Figure 2 shows a calculated phase of a linear amplifier for the channel length $L_{poly} = 1\mu m$. Figure 3 shows the influence of the oxide thickness T_{ox} variation on the circuit performance. For comparison only the T_{ox} value both in our model parameters and MEDUSA [4] input file was varied by ± 10 % from nominal thickness of $7nm$. By reducing the T_{ox} thickness the node voltage of the circuit transistor decreases, which

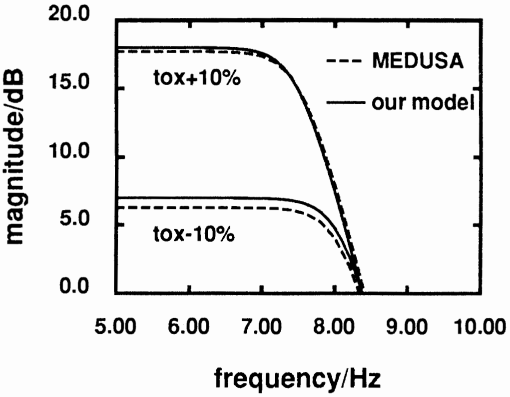

Figure 3: Influence of T_{ox} variation on the performance of a linear amplifier.

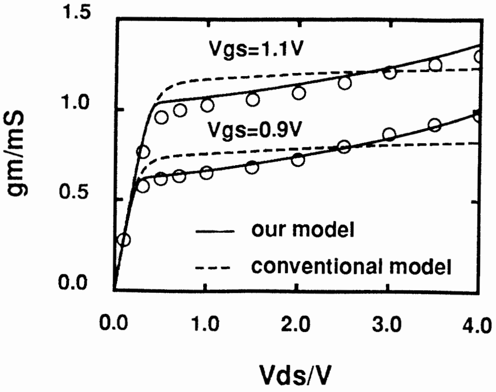

Figure 4: Comparison of the calculated g_m values with our model and a conventional model. Open circles are measurements.

causes the increase of g_{ds}. This is the reason of the amplitude decrease. Our model can predict the circuit performance not only for the nominal T_{ox} value but also the variation correctly.

2. Comparison with a piece-wise model

For the comparison with a piece-wise model we performed the first approach shown in Fig. 1. The model parameter values are extracted for a measured data set with two different models, one with our model and the other with a conventional piece-wise model based on the similar concept as BSIM3. Both models reproduce the measured $I - V$ characteristics well. Figure 4 compares the g_m values of these two models in the moderate inversion region for $L_{poly} = 0.9\mu m$. The conventional model cannot describe g_m correctly for a wide range of applied voltages. This is due to artificial fitting parameters to smooth the transition regions. These parameters can not describe complicated potential behaviors for a wide range of applied voltages. This causes inaccuracy in extracted parameter values. Figure 5 shows a comparison of a simulated amplitude of an operational amplifier. The input current is chosen so that the V_{gs} values on transistors are around 1.0V. The two models show the difference

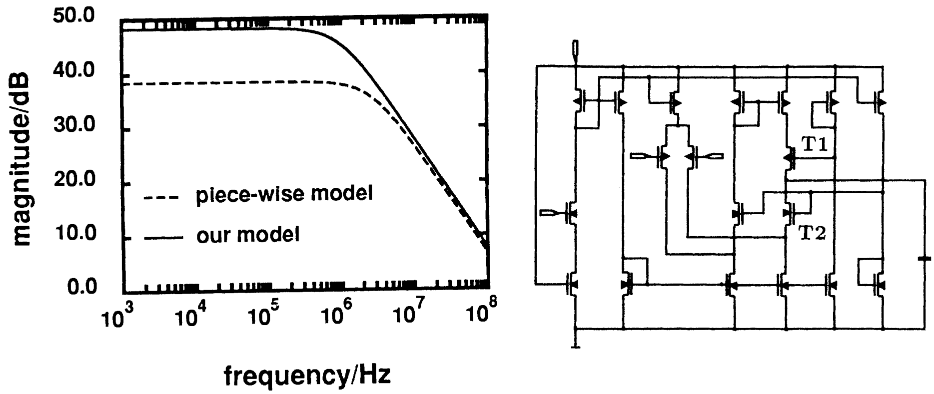

Figure 5: Simulated amplitude of an operational amplifier shown in the right.

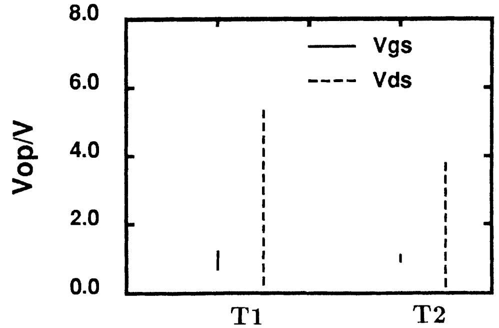

Figure 6: Voltage variation on two key transistors in the operational amplifier shown in Fig. 5 for the frequency range of $10^3 - 10^8$ Hz.

of 10dB in amplitude. Figure 6 shows the voltage variations of two key transistors in the amplifier for the frequency range of $10^3 - 10^8$ Hz. As can be seen the variation of the voltage between the source and the drain V_{ds} is large. For relative small frequency the V_{ds} values stay at their maximum, where the maximum inaccuracy of g_m occurs in the conventional model.

References

[1] Y. P. Tsividis and K. Suyama, "MOSFET modeling for analog circuit CAD: Problems and Prospects," *IEEE J. Solid-State Circuits*, vol. 29, no. 3, pp. 210-216, 1994.

[2] J. A. Power, "An enhanced SPICE MOSFET model suitable for analog applications," *IEEE Trans. Computer-Aided Design*, vol. 11, no. 11, pp. 1418-1424, 1992.

[3] M. Miura-Mattausch, U. Feldmann, A. Rahm, M. Bollu, and D. Savignac,"Unified complete MOSFET model for analysis of digital and analog circuits," *Proc. ICCAD*, pp. 264-267, 1994.

[4] MEDUSA User's Guide, RWTH Aachen, Germany, 1989.