Connection of Network and Device Simulation

P. Dickinger, Ph. Lindorfer, G. Nanz, S. Selberherr
Institute for Microelectronics
Technical University Vienna
Gußhausstraße 27–29, A–1040 Vienna, AUSTRIA

Tel. +43/222/58801–3713

Abstract — We have combined the network simulation program SPICE with the device simulator MINIMOS. The results of this new method in comparison with the SPICE level 3 MOS model are shown.

To achieve accurate results in certain applications of circuit simulation, particularly analog MOS circuits, it is necessary to determine the exact static and dynamic behavior of all transistors. This is usually done by the default MOS models in SPICE or more sophisticated analytical transistor models like BSIM or CSIM. On the other hand, the fully two-dimensional device simulator MINIMOS provides the possibility to get exact results based on fundamental physical principles for the specific structure of each transistor. The default dynamic model in SPICE was developed by Meyer [1] and is not applicable for charge sensitive circuits, the model recommended in [2] does not work in SPICE 2.G6 [3].

The implementation of a fully two-dimensional AC-analysis in MINIMOS enables the accurate calculation of capacitances and conductances [4]. With this capability the combination of network and device simulation is rather straightforward from an electrical engineering perspective. Additionally all short channel effects and the influence of degradation effects in new technologies are taken into account thus yielding realistic predictions for the behavior of the circuits. A similar approach was made previously [5], but only for coupled devices in very small circuits.

In our implementation the connection of MINIMOS and SPICE works in the following way: MINIMOS is added as a subroutine to SPICE. MINIMOS calculates every operating point of each transistor needed by SPICE. The input for MINIMOS is a set of parameters on the .MODEL card of SPICE, that means that the geometry and technology parameters for each transistor in the circuit can be different.

To illustrate the advantages of our approach, we performed simulations with the conventional level 3 MOS model and with our "MINIMOS model". In Fig. 1 the static I-V curve of a particular transistor is shown, the dotted lines are simulations with level 3, the solid lines are MINIMOS results. For comparison we choose the ten transistor differential amplifier circuit from Fig. 2 where most transistors have different channel lengths and widths. In Fig. 3 the results of the AC-analysis with SPICE - MINIMOS (solid lines) and level 3 (dotted lines) are shown. The DC-operating point is nearly the same for both simulations, the different voltage gains and critical frequencies are due to less accurate values of conductances and capacitances in the level 3 MOS model. In transient analyses these differences are of the same order of magnitude. This new approach gives highest possible accuracy regarding device models for circuit simulation.

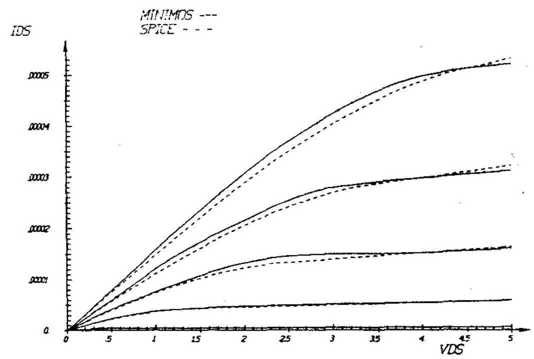

Fig 1: static I-V curve of a particular transistor (dotted lines = SPICE level 3 model, solid lines = MINIMOS results)

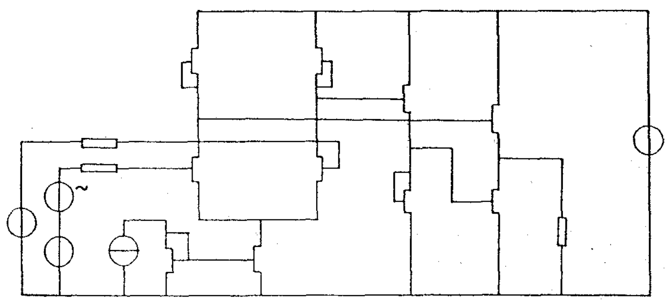

Fig. 2: differential ampifier curcuit

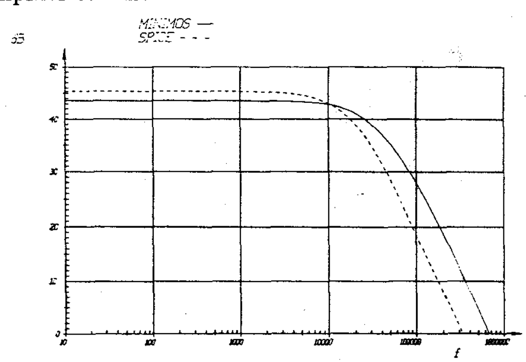

Fig. 3: AC-analysis (dotted line = SPICE level 3 model, solid line = SPICE - MINIMOS result)

[1] Meyer, J.E.; MOS models and circuit simulation, RCA Review, Vol. 32, March 1971

[2] Ward, D.E.; Dutton, R.W.; A charge oriented model for MOS transistor capacitances, IEEE Journal of Solid-State Circuits, Vol. SC-13, No. 5, pp. 703-707 (1978)

[3] Cirit, M.; The meyer model revisited: why is charge not conserved? IEEE Trans. on Computer-Aided Design, Vol. 8, No. 10, pp. 1033-1037 (1989)

[4] S.E. Laux; Application of sinusoidal steady - state analysis to numerical device simulation, J.J.H. Miller (ed), New problems and new solutions for device and process modelling, Dublin: Boole Press (1985)

[5] Mock, M.S.; Time - dependent simulation of coupled devices, Proceedings of the Nasecode II conference 17-19 June 1981, Boole Press Limited (1981)