Macromodel for CMOS Analogue Switches Temperature Effects Sensing

Ivailo M. Pandiev[1]

Abstract – **This paper, is focused on an enhanced SPICE macromodel for CMOS analogue switches, which takes into account the second-order effects, such as the leakage current, the supply current and the on-resistance as a function of temperature. The simulation model is developed through modifying the basic macromodels employing the mechanism of controlled sources and subcircuits. Model parameters are extracted for the integrated analogue switches ADG411 and MAX4601 as examples. The accuracy of the models is demonstrated by comparison with the data sheet parameters of the real ICs.**

Keywords – **Analogue circuits, Temperature effects, CMOS analogue switches, SPICE, Modelling.**

I. Introduction

Solid-state CMOS analogue switches have become an essential component in the design of electronic systems, which require the ability to control and select a specified transition path for the analogue signal. These analogue devices are used in a wide variety of applications including multi-channel data acquisition systems, process control, communication systems, etc. [1, 2].

The temperature effects of the leakage current I_{LKG}, the switch on-resistance R_{ON} and the positive (negative) supply current I_{VDD} (I_{VSS}) are important in many analogue circuits (as programmable amplifiers, filters and oscillators) and complex mixed (analogue-digital) circuits. However, temperature effects are not presented in the available SPICE macromodels of analogue switches. The majority of published SPICE analogue switch macromodels only attempt to model I_{LKG} at $T_{nom} = 27°C$, usually by two ideal current sources connected between positive supply terminal (or ground) and input (or output) nodes [3, 4]. Also existing analogue switch macromodels simulate changing power supply current with output load current and input switching frequency but not accurately model temperature dependence of the supply current. Largely independent from various technically realized circuit structure known so far, an improved SPICE macromodel of analogue switches including temperature dependence of several parameters has been developed, which is based upon manufacturer's data. Macromodelling of temperature effects are based on those described in [5-8] and following the design procedure given in [9]. The equivalent circuit of the proposed model principally contains linear passive elements and non-linear controlled current sources. The macromodel are implemented as subcircuit and the structure of its netlist confirm to the standard SPICE format.

A full description of the enhanced macromodel is given in Section II, including the computation of the model parameters for SPICE simulators. In Section III is presented comprehensive examples, describing the use of the enhanced model, as well as a comparison between data sheet parameters of the real IC and simulation results. Conclusions and further research are presented in Section IV.

II. SPICE macromodel development

The schematic diagram of the proposed macromodel is shown in Fig. 1. It's based on a previous analogue switch macromodels [3, 4] with the additional capability to model the temperature dependence of the leakage current, the supply current and the on-resistance. The existing macromodel is represented in Fig. 1 as black-box marked '*daswitch*' and the new elements and stages, added to the original circuit, are marked in grey.

Various temperature parameters have been modelled as follows.

A. Modelling the leakage current and the supply current as a function of temperature

The temperature dependence of the leakage current and the supply current is created by two temperature coefficients of the resistor model $TC1$ and $TC2$ provided be SPICE [10]. The relation between the resistance value at the certain temperature $R_T(T)$ and the value at nominal temperature $R(T_{nom})$ is controlled with equation:

$$R_T(T) = R(T_{nom})\left[1 + TC1(T - T_{nom}) + TC2(T - T_{nom})^2\right] \quad (1)$$

The temperature-sensitive resistor $R_T(T)$ is chosen with linear function ($TC2 = 0°C^{-2}$) to simplify the calculations for the other components in the model. Since SPICE will give an error message if the resistor goes negative at any temperature, the value of the linear coefficient $TC1$ can be obtained by

$$TC1 \le \frac{10^{-1}}{T_{nom} - T_{\min}} \quad (2)$$

where $T_{\min}$ is the minimum temperature in the operating range of the real part.

In Fig. 1 the current I_T will flow through the temperature-sensitive resistor $R_T(T)$, towards the internal ground. In such

[1]Ivailo M. Pandiev is with the Faculty of Electronics from Technical University of Sofia, Kliment Ohridski 8, 1000 Sofia, Bulgaria. E-mail: ipandiev@tu-sofia.bg

a way the voltage V_{100} will depend upon the temperature. The current source I_T is fixed at $1A$ and the resistor R_T is set to 1Ω such that to provide a voltage $V_{100} = 1V$ at $T_{nom} = 27°C$. The low resistance value of 1Ω leads to minimization of the

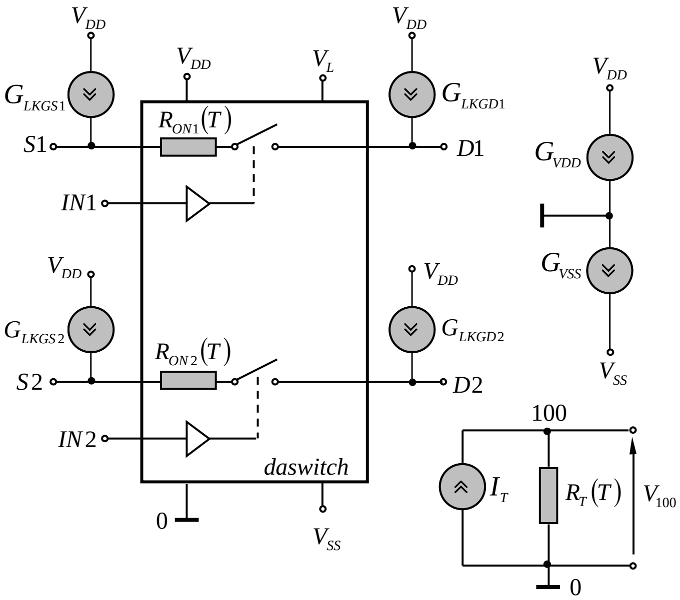

Fig. 1. Conceptual schematic of the proposed analogue switch macromodel improved with temperature effects.

generated thermal noise and to simplify the calculations for the other components in the macromodel. The voltage V_{100} works as a linear temperature-controlled input values for polynomial voltage-controlled current sources (VCCS) G_{LKGS1} (G_{LKGD1}) and G_{VDD} (G_{VSS}) in the model. As a result, both leakage current and supply current will be functions of temperature. The approximation to a non-linear behaviour is realized using appropriate polynomial coefficients of the controlled sources. A sufficient degree of accuracy for the purpose of modelling temperature dependence has been provided by choosing polynomial sources as six-order functions. The VCCS functions are given by

$$I\left(G_{LKGS1(2)}\right) = \sum_{i=0}^{6} k_{i,LKGS1(2)} V_{100}^{i} \tag{3a}$$

and

$$I\left(G_{VDD(VSS)}\right) = \sum_{i=0}^{6} k_{i,VDD(VSS)} V_{100}^{i}\,. \tag{3b}$$

The polynomial coefficients $k_{i,VDD(VSS)}$ and $k_{i,LKGS1(2)}$ in equations (3a) and (3b) are calculated using Matlab. The advantage of this approach is that parameter extraction can be performed only from data sheet parameters, even for circuits whose internal structure is unknown.

B. Modelling the switch on-resistance as a function of temperature

The switch on-resistance is an important consideration in applying analogue switches within programmable amplifiers, attenuators, etc. When the switch is closed (ON), dc performance is affected mainly by the on-resistance and leakage current. A resistive attenuator is created by the resistances of the generator, switch on-resistance and load which produce a gain error. The leakage current flows through the equivalent resistance of load in parallel with the sum of generator resistance and switch on-resistance. Not only can R_{ON} cause gain error, which can be calibrated using a system gain trim, but its variation with temperature and applied input signal (R_{ON} modulation) can introduce dc and ac distortion for which there is no calibration [1, 2]. The analogue switch macromodels described in [3, 4] only attempt to simulate on-resistance at $27°C$, but not model its temperature effects. In response to this problem here is proposed a method for simulating temperature dependence of the on-resistance in the behaviour CMOS analogue switch macromodels.

The existing macromodels for analogue switches contains ideal voltage-controlled switches. The SPICE models of the ideal switches are a special kind of voltage-controlled resistors, where the resistance between switch terminals depends on the voltage between the controlling nodes. As it is the resistance varies continuously between the RON (on-resistance) and ROFF (off-resistance) model parameters [10]. The temperature effect of the RON is developed through modifying SPICE model form of the ideal switch and by utilizing the temperature-dependent resistor model $R_{ON1}(T)$ and $R_{ON2}(T)$, as shown in Figure 1. The temperature RON equation that fit a quadratic curve is given by

$$R_{ON}(T) = RON(0°C)\left[1 + k_{LC}.TEMP + k_{QC}.TEMP^2\right] \tag{4}$$

where $R_{ON}(0°C)$ is the switch on-resistance at $0°C$, $TEMP$ is the predefined parameter using the value of the control temperature option $TNOM$ in $°C$, k_{LC} is the linear coefficient and k_{QC} is the quadratic coefficient. The equation (4) will fit a quadratic curve through three points in a temperature graph by solving three equations with three arguments.

The SPICE netlist for the temperature model form of an analogue switch is shown in Table 1. The numerical values of the model parameters are not specified, because they are determinate for an each CMOS analogue switch according to the manufacturer's data.

TABLE 1.
SPICE NETLIST FOR TEMPERATURE MODEL FORM OF AN ANALOGUE SWITCH

```
.MODEL SMOD_RONT VSWITCH +
+ (RON={RON(0°C)(1+kLC*TEMP+kQC*TEMP*TEMP)}
ROFF=_ _ _ VON=_ _ _ VOFF=_ _ _)
.PARAM  kQLTC=_ _ _ kQLTC=_ _ _
```

Therefore, the electronic circuits including analogue switches can be simulated at temperature other than default value $27°C$ by changing the $TNOM$ option of the SPICE OPTIONS control card.

III. MACROMODEL PERFORMANCE

The verification of the enhanced model with additional elements, representing temperature effects of the leakage currents is carried out by comparing simulation results with data sheet parameters of the integrated IC ADG411 [11]. The analogue switch macromodel for the ADG411 [3] was used, with the I_{LKG} section removed and the new elements and temperature stage from Fig. 1, modelling the leakage current versus temperature, inserted between input/output and positive supply voltage terminal. The test circuit for simulation (Fig. 2), of a one analogue switch (X1.SW1) is created following the test conditions given in the manufacturer's data. According to the data sheet of the selected IC the power supply voltage of the circuit are chosen ±15V and logic supply voltage is set to +5V. For the testing of the analogue switch is

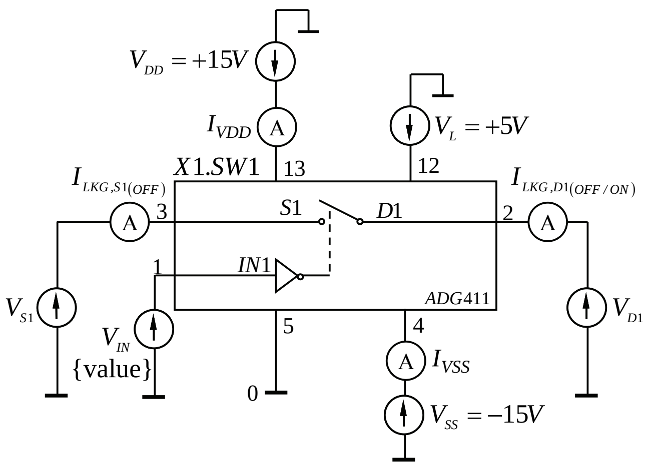

Fig. 2. Test circuit for simulation of leakage currents and supply currents

performed parametric dc sweep analysis with the following sweep parameters: start value $25°C$, end value $85°C$ and increment $1°C$. The simulations are implemented in two modes of operation: (1) closed switch (ON) and (2) opened switch (OFF). The global parameter during simulation processes is the dc value of the logic control input voltage source V_{IN}. The logic levels of the V_{IN}, according to the data sheet of the IC steps from 0V ("ON" condition) to 3V ("OFF" condition). The dc voltage sources connected at the nodes 3 (S1) and 2 (D1) are set to $V_{S1} = -14{,}5V$ and $V_{D1} = +14{,}5V$. The simulation testing is implemented within OrCAD PSpice.

In Fig. 3a, Fig. 3b and Fig. 3c are presented simulation output, experimental results for the leakage currents and error in percents ($\delta = \left[\left(I_{LKG,M} - I_{LKG,DS}\right)/ I_{LKG,DS}\right]100\%$) versus temperature, where the solid lines show the behaviour of the SPICE macromodel. Notice that the simulated parameters closely match of the actual device ($\delta < 3\%$).

The verification of the enhanced model with additional elements, representing temperature effects of the supply current is carried out by comparing simulation results with data sheet parameters of the integrated MAX4601 CMOS analogue switch [12]. The analogue switch macromodel for the MAX4601 [4] was used, with the I_{VDD} (I_{VSS}) section removed and the new elements and temperature stage from Fig. 1, modelling the supply current versus temperature, inserted between positive and negative supply terminals. The test circuit for simulation is presented in Fig. 2. During the process of simulation is specified and performed dc sweep analysis with the temperature range from 25°C to 85°C. According to the manufacturer's data the supply voltage is chosen ±15V and logic supply voltage is set to +5V. In Fig. 4 is shown simulation output (solid lines) and experimental results (data pointed) for the positive and negative supply current. As shown in diagram (Fig. 4), the curves of the supply current versus temperature give a response close to that of the real

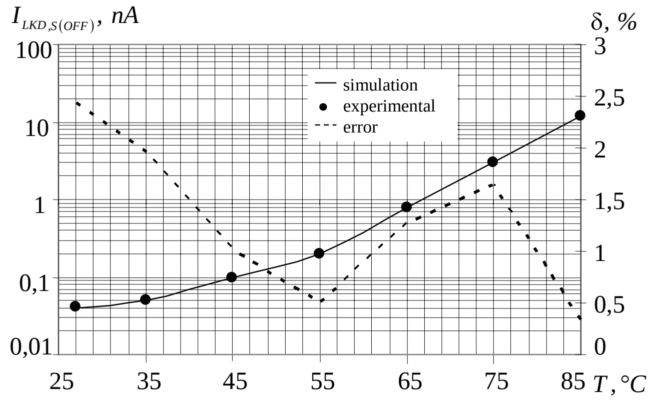

Fig. 3a. Source OFF leakage current as a function of temperature for ADG411

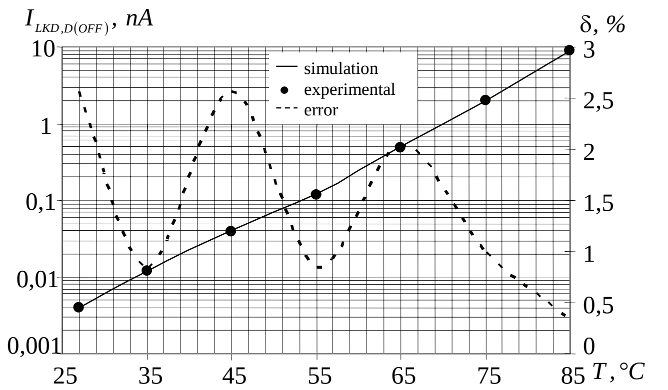

Fig. 3b. Drain OFF leakage current as a function of temperature for ADG411

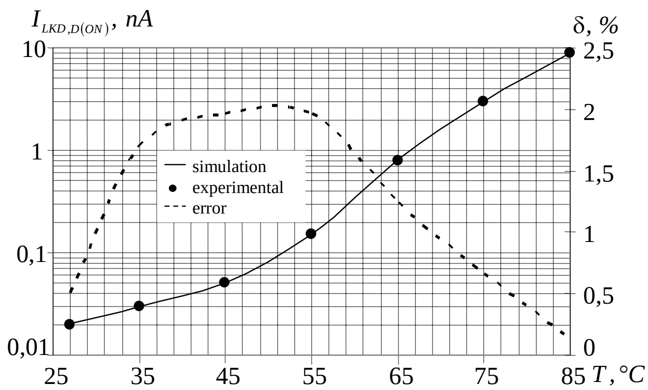

Fig. 3c. Drain ON leakage current as a function of temperature for ADG411

analogue switch. The resulting error between simulation and data sheet parameters is being not higher than 5%.

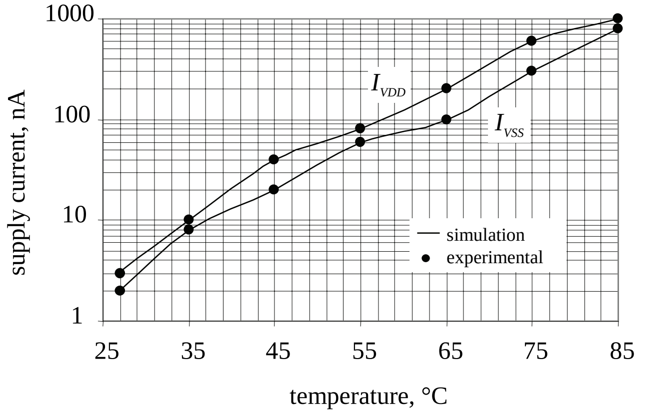

Fig. 4. Positive and negative supply current as a function of temperature for MAX4601

The verification check of the temperature effect for the on-resistance is performed by comparing simulation results with data sheet parameters of the MAX4601. The simulation testing is implemented following the test conditions given in the manufacture's data. The single supply voltage for the circuit is chosen $+12V$, logic supply voltage is set to +5V and logic control input is fixed at +3V. The switch on-resistance is measured for the input voltage $+6V$. The simulation testing is performed within temperature range from -40°C to +85°C. In Fig. 5 is given simulation output, experimental results for the on-resistance and error in percents versus temperature. The solid line shows the behaviour of the SPICE macromodel, using the model parameters in the netlist of Table 2. The data points plotted shows temperature dependence of the on-resistance for the real-world device. As can be seen, the goal of a 2% match between the simulation model and the actual device is achieved.

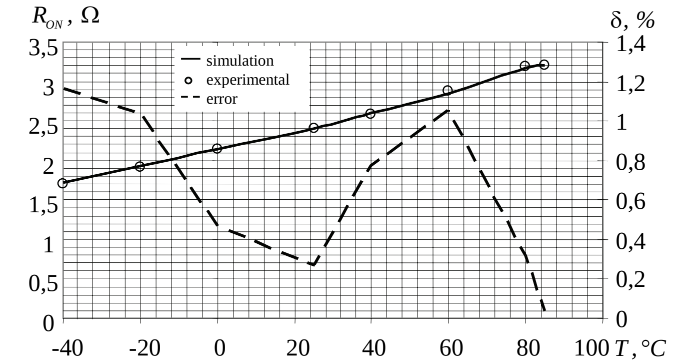

Fig. 5. Switch on-resistance as a function of temperature for MAX4601

TABLE 2.
SPICE NETLIST FOR TEMPERATURE MODEL FORM OF AN ANALOGUE SWITCH MAX4601

```
.MODEL SMOD_RONT VSWITCH
(RON={2.11025641025641*(1+0.00524908869988*TEMP+
+ 9.720534628068047e-006*TEMP*TEMP)} ROFF=0.5E6
+ VON=0.8 VOFF=2.4)
```

IV. CONCLUSION

An improved SPICE macromodel for CMOS analogue switches that incorporates temperature effects of the leakage current, the power supply current and the on-resistance has been developed. It's providing accurate model for dc, ac and transient analyses. The macromodel is independent from actual technical realizations and the model parameters can be obtained only from manufacturer's data of the real IC. The efficiency of the model was proved by comparison of simulation results with data sheet parameters of the integrated CMOS analogue switches ADG411 and MAX4601. The maximum error between simulation and experimental results are not higher than 5%, which guarantee the sufficient degree of accuracy. Despite the addition all these new features, the macromodel's simulation speed and computer resources needed are still equal to those of the SPICE standard libraries models. However, further research is required to develop appropriate macromodels with more complete temperature response.

ACKNOWLEDGEMENT

The author wish to thank Dr. G. Nikolov from the Automated measurement and testing laboratory, Technical University of Sofia, for the many valuable discussions.

REFERENCES

[1] W. Kester, "Data converters", Analog Devices, Norwood, MA, USA, pp. 7.62-7.66, 2005.
[2] *Op Amp Applications*, Analog Devices, Norwood, MA, USA, pp. 6.128-6.129 and pp. 3.26-3.27, 2002.
[3] "Switches/Multiplexers - SPICE Models", Analog Devices 2005.
[4] "Analog Switches and Multiplexers Macromodels", Maxim Integrated Products 2005,http://www.maxim-ic.com/tools/spice.
[5] H. Chenmin, D. Leach and Sh. Chan, "An improved macromodel for operational amplifiers," Int. J. Circ. Theory Appl., vol. 18, pp. 190-203, 1990.
[6] U. Jörges, G. Jummel and G. Müller, "A macromodel of sample-and-hold circuits," Int. J. Circ. Theory Appl., vol. 25, pp. 483-502, 1997.
[7] "Development of an extensive SPICE macromodel for current-feedback amplifiers," National Semiconductor, Application Note AN 840, 1992.
[8] M.E. Brinson and D.J. Faulkner, "SPICE macromodel for operational amplifier power supply current sensing," Electronics Letters, Vol. 30, No. 23, pp. 1911-1912, 1994.
[9] I.M. Pandiev, "Model building and testing procedure - An analogue multiplexer SPICE macromodel improved with temperature effects," ICEST 2006, Bulgaria, Conference Proceedings, pp. 274-277.
[10] *MicroSim PSpice A/D Reference Manual*, MicroSim, Irvine, California USA, Chapter 2, 1996.
[11] "LC2MOS Precision Quad SPST Switch ADG411 - data sheet", Analog Devices, Norwood, MA, USA, 2004.
[12] "CMOS Analog Switch MAX4601 - data sheet", Maxim Integrated Products, 2005.