BEHAVIORAL COMPUTER MODELING OF A/D CONVERTERS USING GENERAL PURPOSE ANALYSIS PROGRAMS

Assoc. Prof. Dr. Katya Krusteva Stefanova*
Assoc. Prof. Dr. Elissaveta Dimitrova Gadjeva**
Department of Electronics, Technical University of Sofia, Branch Plovdiv
Department of Electronics, Technical University of Sofia,
e-mail: egadjeva@vmei.acad.bg

ABSTRACT: Computer behavioral models of analog-to-digital converters (**ADC**) are developed in the work which are applicable to the general purpose analysis programs of the *PSpice* type. The models allow to investigate the nonideality and to assess the error of the convertion of **ADC** for the logarithmic nonlinear sensor characteristics. The total error of the whole device is determined combining the converter model with the input amplifier noise model. The **ADC** investigation is reduced to the *DC* analysis in the defined range of the input signal alteration. The model is universal and is realised as a library block – a module of the *OrCAD PSpice* program. This provides the possibility to use it for modeling and error estimation of different types of **ADC** by defining the corresponding parameters.

I. INTRODUCTION

Recently digital signal processing and filtering have a growing influence in all areas of signal processing. This substantiates the need of CAD software for an accurate and efficient modeling and simulation of corresponding analog-to-digital converters (**ADC**) and digital-to-analog converters (**DAC**)[1-4]. Besides the specialized software, widely used for this purpose, CAD tools for the general purpose simulators are also developed. Some of the possibilities which characterize these simulators as a flexible and efficient tool for investigation of analog-to-digital converters, are:

- the enhanced possibilities of the input language for description of behavioral models;
- the subcircuit and block definitions, the accurate computer library models of the components;
- the ability for parametrization and for statistical modeling of the nonlinear behavior.

The present work considers the developed computer behavioral **ADC** models which are applicable to the general purpose analysis programs of the *PSpice* type. The models allow to investigate the nonideality and to assess the error of the convertion of **ADC** for the logarithmic nonlinear sensor characteristics.

The total error of the whole device can be determined combining the converter model with the input amplifier noise model. The **ADC** investigation has been reduced to the *DC* analysis in the defined range of the input signal alteration. The results are obtained using the graphical *Probe* analyzer.

II. CONSTRUCTING OF THE MODEL

When developing devices for converting and measuring of physical quantities by sensors with nonlinear characteristics, it is important to choose a suitable converting method meeting the technical requirements for accuracy, noise resistance, speed and universality. For example, logarithmic polynomials are used to approximate the characteristics of sensors for cryogenic temperatures:

$$T(R_x) = \sum_{j=1}^{k} \sum_{i=0}^{p} a_{ij} . \lg\left(\frac{R_x}{R_{0j}}\right)^i , \tag{1}$$

where R_x is the sensor resistance, k is the number of measuring ranges (one range corresponds to R_x variation within one decade), p is the polynomial power, a_{ij} are coefficients of the approximating polynomial, R_{oj} is the initial state of the sensor in the j^{th} range.

The developed behavioral model is connected with the used method for interpolation of the logarithmic and exponential functions.

Sufficient accuracy and speed are obtained when applying the method with parallel correction of the results [4]. The $\log_2(x)$ characteristics is approximated with a ssufficient accuracy in the interval $x \in [1,2]$ by the dependence:

$$y = f(x) = \log_2(x) \approx x - 1. \tag{2}$$

The following variable correspondence is introduced for the construction of the behavioral **ADC** model:

- a node voltage V_x of node n_x corresponds to the input signal (the argument) x;
- a node voltage V_y of node n_y corresponds to the function $y=f(x)=\log_2(x)$.

The change of x in the given interval is defined by an independent voltage source of the **VSRC** type connected to the n_x node using the *DC Sweep* option.

The corresponding instruction generated in the *PSpice* simulator input file has the form:

$$.\mathrm{DC\ LIN} \left\langle \begin{matrix}\text{initial}\\ \text{value}\end{matrix} \right\rangle \left\langle \begin{matrix}\text{end}\\ \text{value}\end{matrix} \right\rangle \left\langle \begin{matrix}\text{incre-}\\ \text{ment}\end{matrix} \right\rangle$$

The dependence $f(x)=\log_2(x)$ obtained by the *PSpice* simulator is visualized by the *Probe* graphical analyzer (Fig. 1).

The error Δy is determined as the difference between the initial $y=f(x)$ and the approximating $y_a=f_a(x)$ function:

$$\Delta y = \Delta f(x) = f(x) - f_a(x) \tag{3}$$

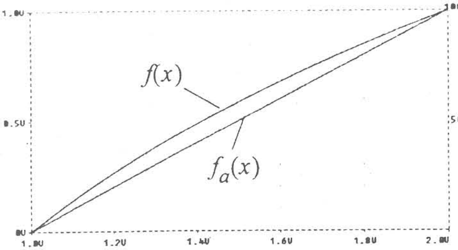


Fig. 1

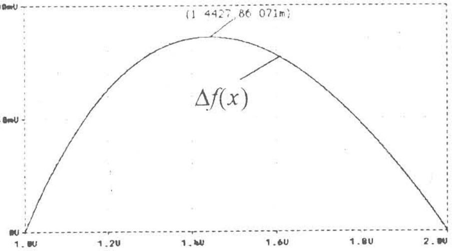


Fig. 2

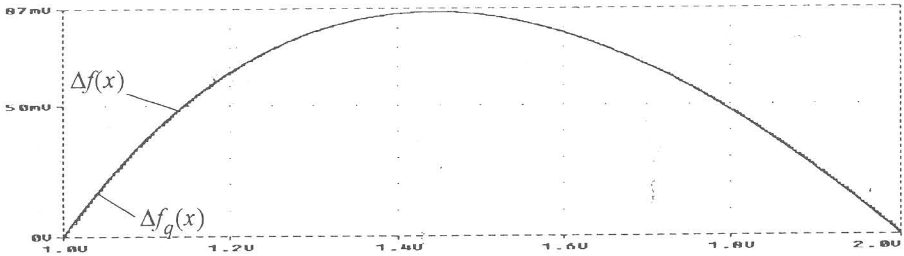


Fig. 3

The plot of Δy obtained by the *Probe* analyzer is shown in Fig. 2. The maximal error is $\Delta y_{max}=86.071\text{x}10^{-3}$.

When the correcting function is used [4] the approximating function $\Delta y_q=\Delta f_q(x)$ of the error Δy is obtained:

$$\Delta f_q(x) = \Delta f(x_q) \tag{4}$$

where

$$x_q = \text{mod}\left[x, \frac{\Delta x}{n}\right] \tag{5}$$

The function $\Delta f_q(x)$ is the approximating quantized function when the interval Δx is divided in n subintervals.

The function mod(a,b) yielding the integer part of the division a/b is used to calculate the correcting function computed for n points. It can be realized in *PSpice* by the dependence [6]

$$\text{mod}[a,b] = \frac{b}{\pi}\cdot arctg\left[tg\left(\frac{a.\pi}{b} - \frac{\pi}{2}\right) + \frac{\pi}{2}\right] \tag{6}$$

According to the input language of *PSpice* the relationship (5) is defined in the expression field of the dependent voltage-controlled sources of **EVALUE** or **GVALUE** type:

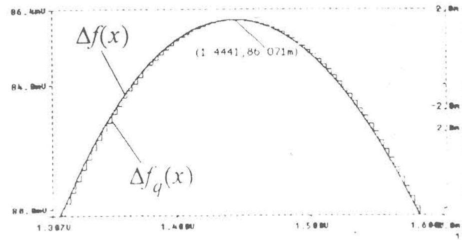
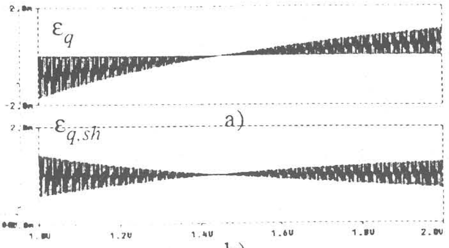


Fig. 4



b)

Fig. 6

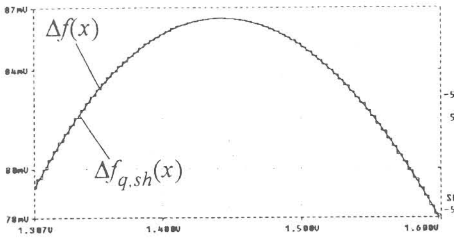
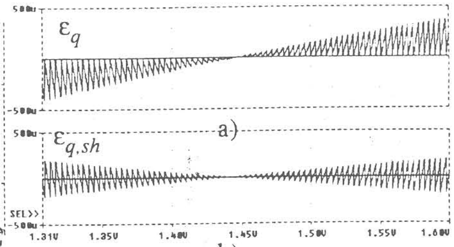


Fig. 5



b)

Fig. 7

```
(b)*(atan(tan(((a*v(in)-1)/(b))*pi-pi/2))+pi/2)/pi
```

where V_{in} is the voltage modeling the input signal.

When the correcting function is used for n=512 the approximating quantized function $\Delta f_q(x)$ as shown in Fig. 3 is obtained for the whole interval $x \in [1,2]$. A zoomed view of the function $\Delta f_q(x)$ is shown in Fig. 4 for clarity.

The accuracy is increased when the approximating function is calculated for the quantized values of the argument, shifted by a half quant [4]:

$$\Delta f_{q,sh}(x) = \Delta f\left(x_q + \frac{\Delta x}{2n}\right) \qquad (7)$$

The error due to quantizing is reduced twice in absolute value and is symmetrical with respect to the zero level. The results for the approximating function $\Delta f_{q,sh}(x)$ obtained using the *PSpice* simulator, are shown in Fig. 5 for a section of the investigated interval.

The resulting error ε_q using the correcting function $\Delta f_q(x)$ is:

$$\varepsilon_q = \Delta f(x) - \Delta f_q(x). \qquad (8)$$

The simulated error ε_q using the *PSpice* program is presented in Fig. 6a. Similarly, the resulting error $\varepsilon_{q.sh}$ using the correcting function $\Delta f_{q,sh}(x)$ is:

$$\varepsilon_{q,sh} = \Delta f(x) - \Delta f_{q,sh}(x)\,, \qquad (9)$$

as shown in Fig. 6b.

Zoomed views of the errors ε_q and $\varepsilon_{q,sh}$ are presented in Fig. 7a and Fig. 7a correspondingly.

The deternination of the ε_q and $\varepsilon_{q,sh}$ errors using the *PSpice* simulator allows an easy assessment of the influence of the number of points n on the **ADC** conversion accuracy. For n=512 the error $\varepsilon_k \leq 1.085 \times 10^{-3}$, and $\varepsilon_{k,sh} \leq 0.862 \times 10^{-3}$. When correcting function calculated for 4096 points is used, the error $\varepsilon_k \leq 0.16 \times 10^{-3}$, and $\varepsilon_{k,sh} \leq 0.086 \times 10^{-3}$.

III. COMPUTER SIMULATION OF THE TOTAL ERROR

The so developed block for approximation of the function $\Delta f(x)$ can be investigated together with the block of the input amplifier in order to assess the error resulting from the temperature drift which is defined by the temperature coefficient of the input voltage of asymmetry. The temperature drift modeling is performed by introducing a voltage source $\Delta v_{in} = T_{KUi} \cdot \Delta t^{o}$, where Δt^{o} is the temperature interval, T_{KUi} is the temperature coefficient of the input voltage of asymmetry.

Δv_{in} is represented in the input block model by a voltage source connected in-series in the input circuit. The error Δv_{in} is introduced as a statistical value in the range

$$[-T_{KUi} \cdot \Delta t^{o}\,,\, T_{KUi} \cdot \Delta t^{o}] \qquad (10)$$

by defining a uniform distribution. In this way, the worst case is modeled which corresponds to the maximal error due to the temperature drift. Thus the total error is obtained for the whole device including the **ADC** and the input amplifier, using statistical (*Monte Carlo*) analysis. The data for the total error are generated in the Summary section of the *Monte Carlo* analysis in the output file of the *PSpice* simulator.

The approach is illustrated by investigation of the block for the following specified parameters:

$$T_{KU_i} = 0.1\mu V/^{\circ}C\,,\ \Delta t^{o} = \pm 10^{\circ}C\,, A_o = 100 \times 10^3,\ K_C = 10,$$

where A_o is the open-loop amplifier gain and K_C is the closed-loop amplifier gain.

The obtained total error as a result from the *Monte Carlo* analysis is 0.8633×10^{-3}. The total error for $TKU_i = 1\mu V/^{\circ}C$ is 0.8635×10^{-3}.

The developed behavioral **ADC** models are included in specialised symbol and model libraries intended for the *PSpice* simulator. The model parameters are easily accessed by the user and allow to assess the accuracy of conversion for different user-defined input parameter values.

IV. CONCLUSIONS

Computer behavioral **ADC** models have been developed, which are applicable to the general purpose analysis program *PSpice*. The models allow to investigate the nonideality of **ADC** and to assess the error of the convertion of **ADC** for the logarithmic nonlinear sensor characteristics. The total error of the whole device has been determined combining the converter model with the input amplifier noise model. The possibilities of the *PSpice* simulator for behavioral modeling, *DC Sweep* and statistical analysis are used to estimate the total error. The computer models are universal and is realised as a library block – a module of the *OrCAD PSpice* program.

REFERENCES

[1] Gielen, G., J. Franca, CAD Tools for Data Converter Design: An Overview, *IEEE Trans. on Circuits and Systems - II: Analog and Digital Signal Prcessing*, vol. 43, No. 2, Febr. 1996, pp. 77-89.

[2] Liu, E., G. Gielen, H. Chang, A. Sangiovanni-Vincentelli and P. Gray, "Behavioral Modeling and Simulation of Data Converters", in *Proc. Intern. Symp. Circuits and Systems (ISCAS)*, 1992, pp. 2144-2147.

[3] Ruan, G., "A Behavioral Model od A/D Converters Using a Mixed-Mode Simulator", *IEEE J. Solid-State Circuits*, vol. 26, pp. 283-290, March 1991.

[4] Cidelko, V., N. Nagaets, J. Hohlov, A. Gladkov, V. Teslenko, "Design of microprocessor measuring devices and systems", *Tehnika*, 1984.

[5] PSpice, Circuit Analysis User's Guide, *Microsim Co.*, Irvine, 1991.

[6] Application Notes, *Microsim Corp.*, USA, 1996.

Suppoted by the National Foundation "Scientific Investigations" under Grant VRP-I-2/1999