Proceedings of the 5th Small Systems Simulation Symposium 2014, Niš, Serbia, 12th-14th February 2014

SPICE Model of a Linear Variable Capacitance

Miona Andrejević Stošović, Marko Dimitrijević, and Vančo Litovski

Abstract - A simple model of the linear capacitance being function of circuit, mechanical or environmental variable will be introduced. That will enable effective SPICE simulation of circuits containing this kind of capacitances such as sensors, actuators, and, generally, circuits with time varying linear capacitances. Illustrative example will be elaborated related to MEMS capacitive pressure sensor.

Keywords – Modeling, MEMS, SPICE Simulation.

I. INTRODUCTION

Capacitors with variable capacitances may be grouped into two categories: the ones controlled by their own voltage, and the ones controlled by a variable that may or may not be electrical. In the first case we in fact have a nonlinear capacitance which may be treated as described in [1]. These are out of the scope of this paper. Here we will consider linear capacitors only whose capacitance is properly controlled by a circuit variable or force, pressure, light, temperature or some other environmental variable.

To our knowledge the first capacitor with variable capacitance was patented by Nikola Tesla in 1896 [2]. It was the "vacuum variable capacitor". The variation of the capacitance was achieved by rotating one of the capacitor`s plates so changing the overlapping area. This component went into series production in 1942. Much more frequently implemented capacitor of this kind was the one used in the heterodyne to allow for selection of the radio station by the wireless receiver. It was built as "a group of semicircular metal plates on a rotary axis ("rotor") that are positioned in the gaps between a set of stationary plates ("stator") so that the area of overlap can be changed by rotating the axis" [3].

Today we have many versions of capacitors whose capacitance is controlled by some environmental or circuit variable. For example, MEMS pressure sensors are most frequently capacitive [4,5]. In addition, one may use the capacitive properties of some semiconductor components as a light sensor [6] or one may synthesize an electronic circuit where the capacitance will be controlled by some circuit variable [7].

Finally, temperature is always a controlling variable to a capacitance and especially for supercapacitors which are now emerging for everyday use [8]. In any case, for design of a system based on such component one needs a model that will accurately encompass the devices properties and, in the same time, will be easily implementable in the most popular electronic circuits simulation program SPICE [9].

In the next we will first expose the problem. Then a solution will be proposed followed by an illustrative example.

II. MODELLING THE LINEAR CAPACITOR CONTROLLED BY A VARIABLE PARAMETER

The capacitor is introduced into the circuit description by two equations constituting its model no matter what specific properties it has:

$$q_c = f(v_c, \underline{p}) \tag{1}$$

$$i_c = \mathrm{d}q_c/\mathrm{d}t. \tag{2}$$

Here q_c is the charge captured by the capacitor, v_c is the voltage at its terminals, $\underline{p}$ is a vector of controlling variables, i_c is the capacitor`s current and t is the time variable. The parameter(s), $\underline{p}$, may be time varying which means that, implicitly, we presume that the capacitor has a time varying capacitance. The main difference with usual capacitors having time varying capacitance here is in that in our case the capacitance is a controlled (not independent) variable and it will be treated as such.

In the next, for simplicity, we will consider $\underline{p}$ to be one-dimensional, i.e. we will use p.

Modeling of this kind of component was addressed and solved in [1] and implemented in [10,11]. Implementation of that solution, however, needs intervention into the simulator`s code what was done with the Alecsis simulator [12] in [10,11]. Here, we will go for a modelling procedure that will be based on circuit elements being normally recognized by the SPICE`s input language.

For a linear capacitor (1) may be represented as follows:

$$q_c = C(p) \cdot v_c = C_0 \cdot g(p) \cdot v_c \tag{3}$$

where $C(p)=C_0 \cdot g(p)$ was introduced. C_0 is a conveniently chosen constant. Substitution in (2) leads to the following:

$$i_c = C_0 \cdot g(p) \cdot \frac{dv_c}{dt} + C_0 \cdot v_c \frac{dg(p)}{dp} \cdot \frac{dp}{dt}. \tag{4}$$

Note, the dependence $C(p)$ [or $g(p)$] and its derivative

Miona Andrejević Stošović and Marko Dimitrijević are with the, University of Niš, Faculty of Electronic Engineering, Aleksandra Medvedeva 14, 18000 Niš, Serbia, E-mail: {miona.andrejevic, marko.dimitrijevic}@elfak.ni.ac.rs.

Vančo Litovski is with NiCAT - Niš Cluster of Advanced Technologies, 18000 Niš, Serbia (vanco.litovski@elfak.ni.ac.rs)

$\dfrac{dC(p)}{dp}$ [or $\dfrac{dg(p)}{dp}$] are considered known functions representing the capacitance`s properties.

To get a circuit representation of (4) it will be rearranged in the following way:

$$i_c = \left(C_0 \cdot \frac{dv_c}{dt}\right) \cdot g(p) + \left(C_0 \cdot \frac{dp}{dt}\right) \cdot v_c \cdot \frac{dg(p)}{dp}. \qquad (5)$$

That may be seen as a parallel connection of two current sources:

$$i_{cv} = \left(C_0 \cdot \frac{dv_c}{dt}\right) \cdot g(p) \qquad (6a)$$

and

$$i_{cp} = \left(C_0 \cdot \frac{dp}{dt}\right) \cdot v_c \cdot \frac{dg(p)}{dp}. \qquad (6b)$$

The first one, i_{cv}, represents a product of a current of a linear constant capacitance C_0 and a voltage of a nonlinear controlled source $g(p)$. The second represents a product of three quantities: the quantity $\left(C_0 \cdot \dfrac{dp}{dt}\right)$ whose nature will be considered later on, the capacitor`s voltage v_c, and the known derivative represented as a controlled source: $\dfrac{dg(p)}{dp}$. It is important to note that p may be a circuit variable or a controlling (excitation) one defined outside of the circuit. The modelling procedure will have to take that into account by developing two variants of the model.

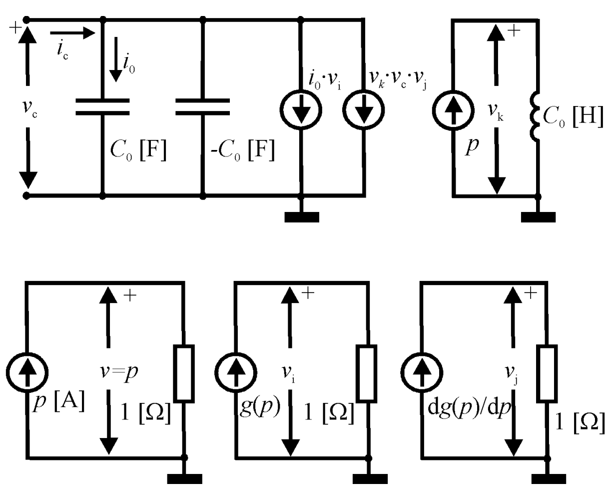

Fig. 1 Circuit representation of (5) for the case when p is controlling (independent) variable

A. p is an independent controlling variable

In this case two controlled sources are available in advance: $v_i=g(p)$ and $v_j=\dfrac{dg(p)}{dp}$. The quantity $i_0 = \left(C_0 \cdot \dfrac{dv_c}{dt}\right)$ will be related to a constant capacitor while $v_k = \left(C_0 \cdot \dfrac{dp}{dt}\right)$ will be treated as a voltage. That means that p will be considered a current and we will use an inductance of value C_0 to get that voltage.

The complete circuit modelling (5) is depicted in Fig. 1.

B. p is a circuit variable

Again, $v_i=g(p)$ and $v_j=\dfrac{dg(p)}{dp}$ are available in advance. The quantity $i_0 = \left(C_0 \cdot \dfrac{dv_c}{dt}\right)$ will be related to a constant capacitor while $\left(C_0 \cdot \dfrac{dp}{dt}\right)$ will be treated depending on the nature of p. If p is a node voltage, $i_k = \left(C_0 \cdot \dfrac{dp}{dt}\right)$ will represent a current. This situation is depicted in Fig. 2.

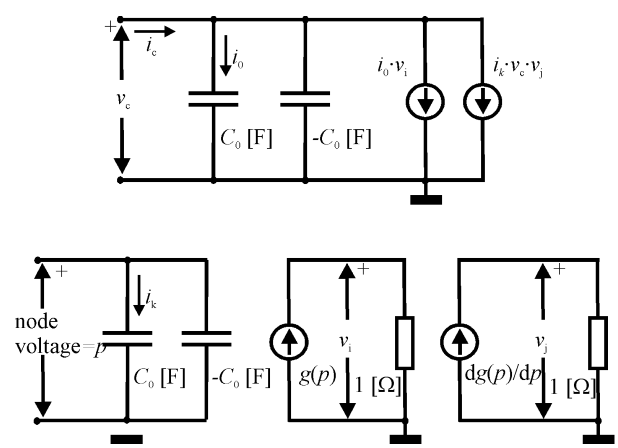

Fig. 2 Circuit representation of (5) for the case when p is circuit variable - voltage

C. p is a branch current

Finally, if p is a branch current the circuit of Fig. 3 may be used.

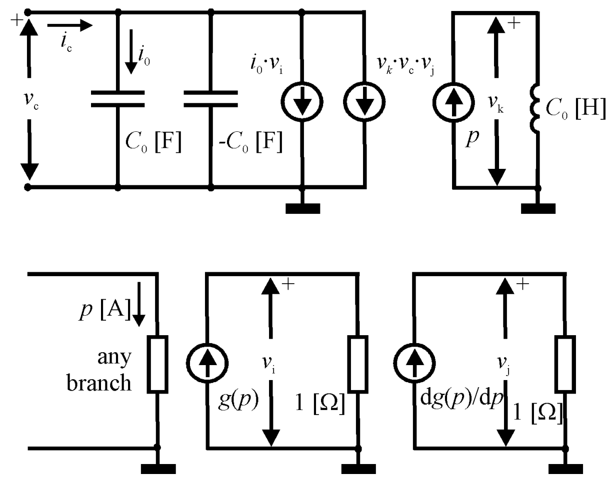

Fig. 3 Circuit representation of (5) for the case when p is circuit variable - current

III. IMPLEMENTATION EXAMPLE

To illustrate we will implement the above modeling procedure to a capacitive pressure sensor followed by a sampling circuit as described in [10,11] and depicted in Fig. 4a. The dependence of the sensing capacitance (here denoted as C_S) on the outside pressure is depicted in Fig. 4b. It was approximated by a polynomial:

$$C(p)=(8.2623-4.799p+67.256\cdot p^2-213.97\cdot p^3+306.2\cdot p^4-200.83\cdot p^5+49.709\cdot p^6)\ [\text{pF}] \quad (7)$$

Least squares approximation was used to get the coefficients in (7). Fig. 4c represents the approximation error.

The iterative process was stopped at $R^2=1-\frac{SS_{\text{res}}}{SS_{\text{tot}}}=0.998$. Here $SS_{\text{tot}}=\sum_{i=1}^{n}[\hat{C}(p_i)-\bar{C}]^2$ is the total sum of squares; $SS_{\text{res}}=\sum_{i=1}^{n}[\hat{C}(p_i)-C(p_i)]^2$ is the sum of squares of residuals based on (7); $\bar{C}=\frac{1}{n}\sum_{i=1}^{n}\hat{C}(p_i)$; $\hat{C}(p_i)$ are samples taken from Fig. 4b, and n=30.

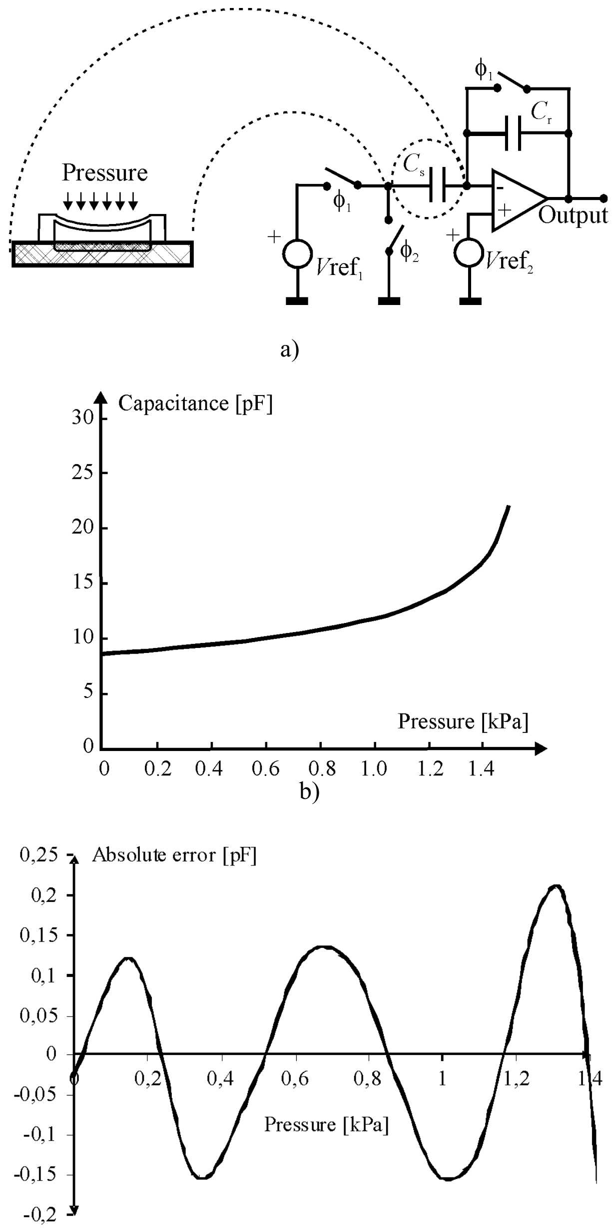

Fig. 4. a) Capacitive pressure sensing circuit, b) characteristic of the pressure to capacitance conversion device, and c) the approximation error

For verification, a sinusoidal excitation was used: $p(t)=0.7+0.7\cdot\sin(\omega t)$ [kPa], where $\omega=2000\cdot\pi$ s^{-1}. $C_0=C(p_0=0)=8.2623$ pF. Fig. 5 represents the simulation results based on the use of Fig. 1 as the circuit model. The following parameters were used: $V_{\text{ref}}=0.1$V and $C_r=C_0$. The period of the sampling signal was $T_s=30\mu$s.

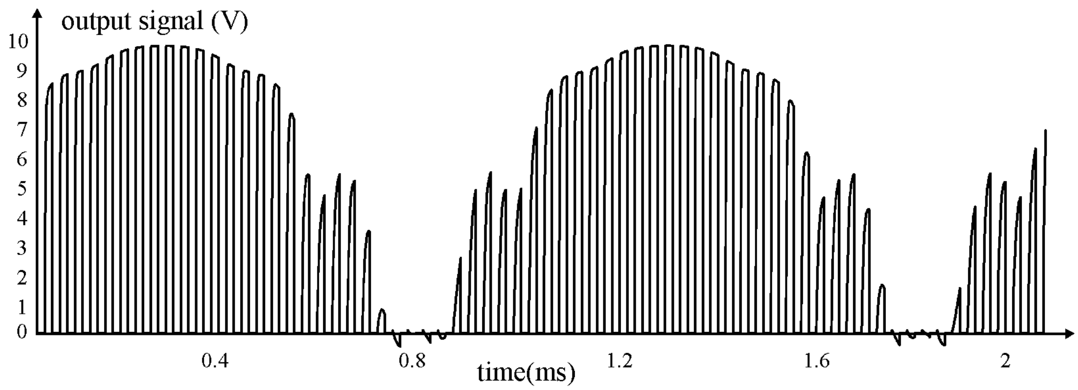

Fig. 5. Output signal obtained by SPICE simulation of the circuit of Fig. 4a.

IV. CONCLUSION

The subject of modeling a linear capacitance controlled by some dependent or independent variable was revisited. In our previous research we implemented the model in a way that was asking for intervention into the simulation software`s code. Here, we express an idea of how the same model may be implemented using the SPICE input language and SPICE predefined circuit elements, only. The results obtained confirm the feasibility of the idea while asking in the same time for better approximation i.e. approximation that includes both the function and its derivative.

ACKNOWLEDGEMENT

This research was partially funded by The Ministry of Education and Science of Republic of Serbia under contract No. TR32004.

REFERENCES

[1] Litovski, V., and Zwolinski, M., "VLSI Circuit Simulation and Optimization." Chapman and Hall, London, 1997.

[2]http://en.wikipedia.org/wiki/Vacuum_variable_capacitor

[3] http://en.wikipedia.org/wiki/Variable_capacitor

[4] Mrčarica, Ž., Litovski, V., and Detter, H., "Modeling and Simulation of Microsystems Using Hardware Description Language", Research Journal on Microsystem Technologies, Vol. 3, No. 2, Feb., 1997, pp. 80-85.

[5] Juillard, J., Arcamone, J., Arndt, G., and Colinet, E., "Influence of excitation waveform and oscillator geometry on the resonant pull-in of capacitive MEMS oscillators", Published in: Design, Test, Integration and Packaging of MEMS/MOEMS (DTIP 2013), Barcelona, Spain, 2013, http://hal.archives-ouvertes.fr/docs/00/83/05/33/PDF/DTIP_auteur.pdf.

[6] Haidar, S., " Light-controlled oscillator uses solar cell junction capacitance", EDN Network, August 8, 2013, http://www.edn.com/design/analog/4419444/Light-controlled-oscillator-uses-solar-cell-junction-capacitance.

[7] Abbadi, M.I. and Jaradat, A.-R. M., "Artificial Voltage-Controlled Capacitance and Inductance using Voltage-Controlled Transconductance", World Academy of Science, Engineering and Technology, International Science Index 20,Vol. 2, No. 8, pp. 782 - 786.

[8] Kötz, R., Hahn, M., and Gallay, R., "Temperature behaviour and impedance fundamentals of supercapacitors", J. of Power Sources, Vol. 154, No. 2, 2006, pp. 550–555.

[9] Nagel, L. W, and Pederson, D. O., "SPICE (Simulation Program with Integrated Circuit Emphasis)", Memorandum No. ERL-M382, University of California, Berkeley, Apr. 1973.

[10]V. Litovski, M. Andrejevic, M. Zwolinski, "ANN Based Modeling, Testing, and Diagnosis of MEMS," IEEE 7th Seminar on Neural Network Applications in Electrical Engineering, NEUREL 2004, Sept. 2004, Belgrade, pp. 183-188.

[11]Litovski, V., Andrejević, M, Zwolinski. M. "Behavioural modelling, simulation, test and diagnosis of MEMS using ANNs", Proc. Intl. Symp. Circuits and Systems, ISCAS`05, Kobe, Japan, 2005, pp. 5182-5185.

[12]Mrčarica, Ž, Ilić, T., Glozić, D., Litovski, V., and Detter, H., "Mechatronic Simulation Using Alecsis: Anatomy of the Simulator", Proc. of the Eurosim'95, Vienna, Austria, Sept. 1995, pp. 651-656.