New PSPICE Modeling and Simulation Method for Multiple Mode Oscillation for ECAM

D. N. Vizireanu[1], I. Pirnog[2], R. M. Udrea[3]

Abstract - **This paper presents new methods of simulating the algebraic functions as well as the solving of non-linear differential equations using PSPICE, one of the most well known circuit analysis program. A new negative resistance oscillator model was used for the single and multiple mode LCR networks of ECAM. The results obtained by numerical integration of the differential equations using a new PSPICE method are compared with the analytic approximations.**

Keywords - **PSPICE, nonlinear circuits, ECAM.**

I. INTRODUCTION

This paper presents new methods of simulating the algebraic functions as well as the solving of non-linear differential equations using PSPICE.

In this paper we present the following aspects: integrated circuits simulation, PSPICE sub circuits achievement for simulating algebraic functions and solving non-linear differential equations.

A new negative resistance oscillator model was proposed by Walker and Connelly [1], [2]. In this paper an analytic approximation to the periodic solutions for LCR networks of Emitter Coupled Astabil Multivibrator (ECAM) is obtained.

These predictions are compared with the results obtained by numerical integration of the differential equations using a new PSPICE method.

II. PSPICE ALGEBRAIC FUNCTIONS SIMULATION

The circuit from Fig. 1 is used for showing the way that PSPICE works in simulating algebraic functions.

And we will also use the equations:

$$E_3 = E_2 \quad , \ V_4 = \ln V_i \,, \tag{1}$$

We make the presumption that the voltage commanded source E_4 is:

$$E_4 = 10^{12}\left(E_3 - E_2\right), \tag{2}$$

[1]Dragos Nicolae Vizireanu is with the Faculty of Electronics, Telecommunications and Information Technology of “Politehnica” University, Iuliu Maniu 1-3, 7000 Bucharest, Romania, E-mail: nae@comm.pub.ro.

[2]Ionut Pirnog is with the Faculty of Electronics, Telecommunications and Information Technology of “Politehnica” University, Iuliu Maniu 1-3, 7000 Bucharest, Romania, E-mail: ionut@comm.pub.ro.

[3]Radu Mihnea Udrea is with the Faculty of Electronics, Telecommunications and Information Technology of “Politehnica” University, Iuliu Maniu 1-3, 7000 Bucharest, Romania, E-mail: mihnea@comm.pub.ro.

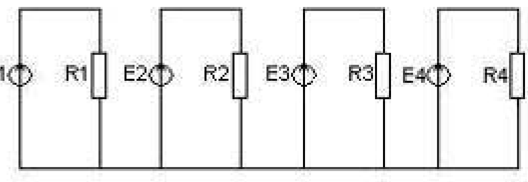

Fig. 1. Function generation sub circuit.

then:

$$E_3 - E_2 = \frac{E_4}{10^{12}} \tag{3}$$

Also, if we consider that E_2 and E_3 are much bigger than $E_4/10^{12}$, we obtain:

$$E_3 = E_2 \,. \tag{4}$$

Using PSPICE there can be generated a set of algebraic functions using the circuit from Fig. 1 and Eq. (5). This idea is based on the following presumptions:

a) E_2 and E_3 have a polynomial dependence for V_i and E_4;

b) $E_4 = F\left(V_i\right)$;

c) it is possible that an equilibrium is obtained in the conditions of Eq. (6).

Example, if the output voltage E_4 must be:

$$V_4 = \ln V_i \text{, or } V_i = e^{V_4} \tag{5}$$

where V_i is the input voltage, if we impose:

$$E_3 = V_i \,, \tag{6}$$

then, in the conditions of Eq. (5), we obtain:

$$V_i = e^{V_4} \text{, or } V_4 = \ln V_i \tag{7}$$

Using the same method there can be generated also other algebraic functions that can be represented as a polynomial decomposition.

Integration simulation is based on the dependence between the voltage and the current of a capacitor.

The circuit that corresponds to this function is represented in Fig. 2. R_4 is used in parallel with the capacitor to allow achieving of static points at the start of the PSPICE algorithm. Its value must be as big as possible in order to have no influence in the function of the circuit.

The loading current of the capacitor simulated by the current source F_4, commanded by the voltage source V_i (where: $R_3 = 10k\Omega$) is:

$$F_4 = \frac{V_i}{R_3} \tag{8}$$

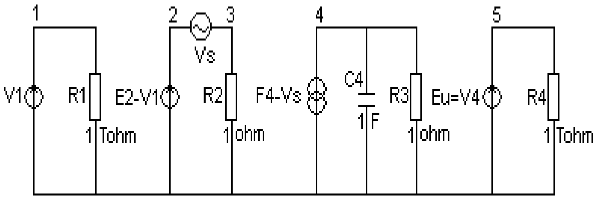

Fig. 2. The integration subcircuit.

The V_4 voltage can be written:

$$V_4 = \frac{1}{C_4}\int F_4 dt \ . \qquad (9)$$

If $E_5 = V_4$ and $C_4 = 100\mu F$, Eq. (8) becomes:

$$V_5 = \int V_i dt \ . \qquad (10)$$

With the help of the C_4 capacitor we can control the initial conditions for integration.

III. Solving the Nonlinear Differential Equations for the Emitter-Coupled Astable Multivibrator (ECAM)

In the scientific literature there are a multitude of algorithms for solving the nonlinear differential equations. An approach needs two aspects:
a good knowledge of the mathematical algorithm, for choosing the most appropriate algorithm to use;
the knowledge of a programming language for the implementation of the algorithm.

The purpose of this paragraph is to present a new method, simple and fast, to solve the nonlinear differential equations using PSPICE.

A. Single-Mode E.C.A.M.

In [1], [2] Walker and Connelly proposed a new negative resistance oscillator model for a class of current, negative resistance oscillators.

The equivalent circuit is presented in Fig. 3.

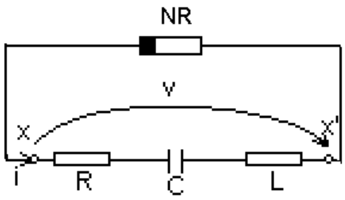

Fig. 3. The single-mode ECAM

The model [1] between x and x' is:

$$v = -a\cdot i + \frac{b}{2}\ln\left(\frac{1-c\cdot i}{1+c\cdot i}\right) \qquad (11)$$

where:

$$a = 2RC \ , \ b = 2V_R \ , \ c = \frac{1}{L} \qquad (12)$$

The nonlinear differential equation for the single-mode oscillator circuit in Fig. 3 is:

$$0 = \frac{d^2 i}{dt^2} + \frac{R-a}{L}\left(1 + \frac{bc}{R-a}\cdot\frac{1}{1-c^2 i^2}\right)\frac{di}{dt} + \frac{i}{LC} \qquad (13)$$

With the notation,

$$\omega^2 = \frac{1}{LC} \ , \ \beta = \frac{bc}{a-R} \ , \ \varepsilon = \frac{a-R}{\sqrt{\frac{L}{C}}} \qquad (14)$$

and the changing of variables:

$$i \to \frac{X}{C} \ , \ t \to \frac{t}{\omega} \qquad (15)$$

the Eq. (13) becomes:

$$\frac{d^2 x}{dt^2} - \varepsilon\left(1 - \frac{\beta}{1-x^2}\right)\frac{dx}{dt} + x = 0 \qquad (16)$$

The Eq. (16) can be written:

$$\frac{d^2 x}{dt^2} = E_a + E_b + E_c \ , \ E_a = -x^2\frac{d^2 x}{dt^2} \qquad (17)$$

$$E_b = \varepsilon\left(1-\beta-x^2\right)\frac{dx}{dt} \ , \ E_c = x\left(1-x^2\right) \qquad (18)$$

B. N-Mode E.C.A.M.

The equivalent circuit is presented in Fig. 4.

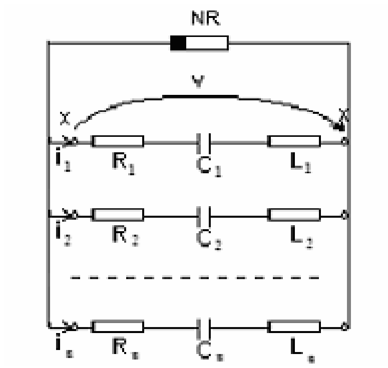

Fig. 4. The n-mode ECAM

The differential equations for the n-mode ECAM circuit were derived [2]:

$$\frac{d^2 i_j}{dt^2} - \frac{a-R_j}{L_j}\left[\frac{di_j}{dt} + \frac{a}{a-R_j}\sum_{\substack{k=1\\k\neq j}}^{n}\frac{di_k}{dt} + \right.$$

$$\left. + \frac{bc}{R_j - a}\frac{\sum_{\substack{k=1\\k\neq j}}^{n}\frac{di_k}{dt}}{1-c^2\left(\sum_{k=1}^{n} i_k\right)^2} + \frac{i_j}{L_j c_j}\right] = 0 \qquad (19)$$

$$j = \overline{1,n}$$

with the conditions: $-1 < c\left(\sum_{j=1}^{n} i_j\right) < 1$.

The transformation of variables:

$$i_j \to \frac{x_j}{C_j};\ j=\overline{1,n}\ ,\ x_j = \frac{a-R_j}{L_j};\ j=\overline{1,n} \qquad (20)$$

and $\omega_1^2 = \frac{1}{L_1C_1};\ \omega_2^2 = \frac{1}{L_2C_2}\ ,\beta_j = \frac{bc}{a-R_j};\ j=\overline{1,n}\ ,$ (21)

$$\tau_j = \frac{a}{a-R_j};\ j=\overline{1,n}\ , \qquad (22)$$

allows us to obtain the dimensionless forms:

$$\frac{d^2x_j}{dt^2} - \pi_j\left[1-\beta_j\frac{1}{1-\left(\sum_{k=1}^{n}x_k\right)^2}\right]\frac{dx_j}{dt} +$$

$$+\omega_j^2 x_j - \pi_j\left[\tau_j - \frac{\beta_j}{1-\left(\sum_{k=1}^{n}x_k\right)^2}\right]\sum_{\substack{k=1\\k\neq j}}^{n}\frac{dx_k}{dt} = 0 \qquad (23)$$

$$j=\overline{1,n}$$

We investigate these equations with a new computational method using PSPICE and to determine an analytical approximation for periodic solutions.

The interest for these equations is based on the fact that they represent a model of equations that describe a new class of nonlinear differential equations. Enforcing quasi-linear with:

$$\pi_1 << \omega_1,\ \pi_2 << \omega_2,\dots,\pi_n << \omega_n \qquad (24)$$

permits approximate solutions for Eq. (23):

$$\begin{gathered} x_1 = \alpha_1(t)\sin(\theta_1),\quad x_2 = \alpha_2(t)\sin(\theta_2) \\ x_j = \alpha_j(t)\sin(\theta_j),\quad j=\overline{1,n} \end{gathered} \qquad (25)$$

with:

$$\begin{gathered} \theta_1 = \omega_1 t + \Phi_1(t),\quad \theta_2 = \omega_2 t + \Phi_2(t), \\ \theta_j = \omega_j t + \Phi_j(t),\quad j=\overline{1,n} \end{gathered}\ . \qquad (26)$$

The method of equivalent linearization [4] will permit construction of the time derivatives of (23). Applying the linearization method to (23) with (24) assumed gives:

$$\frac{d\alpha_1}{dt} = \frac{\alpha_1 x_1}{2}\left[1-\frac{2\beta_1}{c^2\alpha_1^2}\left(1-\sqrt{1-c^2\alpha_1^2-\sum_{\substack{k=1\\k\neq j}}^{n}\frac{c^2\alpha_k^2}{2}}\right)\right] \qquad (27)$$

$$\frac{d\alpha_j}{dt} = \frac{\alpha_j x_j}{2}\left[1-\frac{2\beta_j}{c^2\alpha_j^2}\left(1-\sqrt{1-c^2\alpha_2^2-\sum_{\substack{k=1\\k\neq j}}^{n}\frac{c^2\alpha^2}{2}}\right)\right] \qquad (28)$$

With: $$\frac{d\Phi_1}{dt}=0,\frac{d\Phi_2}{dt}=0,\dots,\frac{d\Phi_n}{dt}=0 \qquad (29)$$

The steady-state amplitudes of oscillation may be obtained by equating Eqs. (27) and (28) to zero.

We can have analytic approximation to the periodic solutions. These predictions are compared with the results obtained by PSPICE numerical integration of the differential equations.

C. Example: Double-Mode E.C.A.M.

The equivalent circuit is presented in Fig. 5.

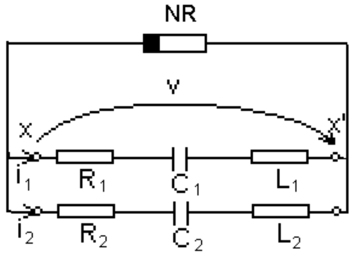

Fig. 5. The double-mode ECAM

Applying the analytical method from the multiple mode ECAM for the double mode ECAM we obtain :

$$\frac{d\alpha_1}{dt} = \frac{\alpha_1 x_1}{2}\left[1-\frac{2\beta_1}{c^2\alpha_1^2}\left(1-\sqrt{1-c^2\alpha_1^2-\frac{c^2\alpha_2^2}{2}}\right)\right] \qquad (30)$$

$$\frac{d\alpha_2}{dt} = \frac{\alpha_2 x_2}{2}\left[1-\frac{2\beta_2}{c^2\alpha_2^2}\left(1-\sqrt{1-c^2\alpha_2^2-\frac{c^2\alpha_1^2}{2}}\right)\right] \qquad (31)$$

Using the following definitions:

$$Y_1 = C_2\alpha_1^2 > 0, Y_2 = C_1\alpha_2^2 > 0\ , \qquad (32)$$

we can rewrite Eqs. (30) and (31):

$$\frac{Y_1^2}{4\beta_1^2} + \left(1-\frac{1}{\beta_1}\right) + \frac{Y_2}{2} = 0 \qquad (33)$$

$$\frac{Y_2^2}{4\beta_2^2} + \left(1-\frac{1}{\beta_2}\right) + \frac{Y_1}{2} = 0\ . \qquad (34)$$

We have the conditions:

$$0<\beta_1<1;\ 0<\beta_2<1 \qquad (35)$$

The next step is the comparison of the analytical approximations for the periodic solutions and the results obtained by PSPICE numerical integration of the differential equations.

IV. THE PSPICE METHOD TO SOLVE NONLINEAR EQUATIONS FOR E.C.A.M.

PSPICE has become the standard computer program for most electrical simulation.

Higher-level abstraction and hierarchy can be modeled using controlled sources and subcircuits blocks. The nonlinear function applies only to the time domain. PSPICE supports the polynomial sources.

A functional model for single mode is presented in Fig. 6 and for double mode in Fig. 7.

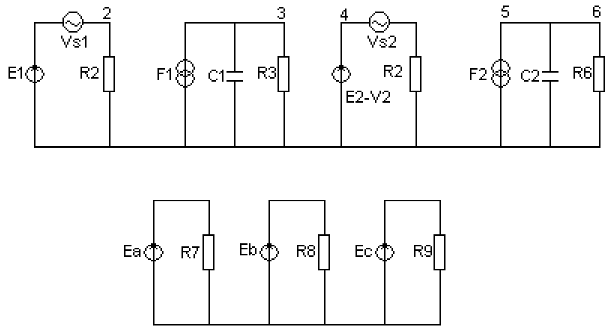

Fig. 6. PSPICE equivalent scheme for single mode nonlinear differential equation

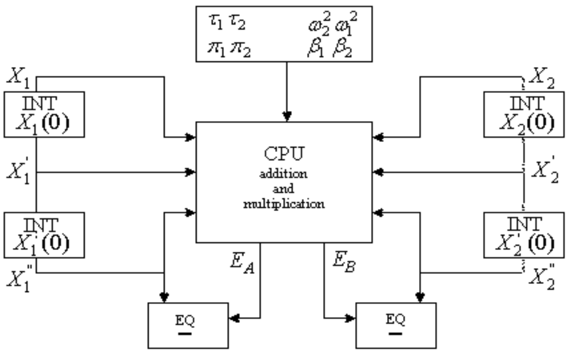

Fig. 7. PSPICE blocks (double mode)

Addition and multiplication (CPU block in Fig. 7) can be achieved with polynomial voltage-controlled current source (VCCS):

EA 3 0 $POLY(2)$ 1 0 2 0 0 1 $*V(3)=V(1)+V(2)$

EB 4 0 $POLY(2)$ 1 0 2 0 0 0 0 0 1 $*V(3)=V(1)+V(2)$

where $V(1), V(2), V(3), V(4)$ are voltages at nodes 1, 2, 3 and 4, in reference to ground (node 0).

The integrator (INT blocks in Fig. 7):

$$V_c(t)=\frac{1}{C}\int i_c(t)dt+v_\infty \tag{36}$$

is used in PSPICE to model the capacitor. If

$$i_c(t)=\frac{V_i}{R} \tag{37}$$

with $R=10k\Omega$ and $C=100\mu F$:

$$V_c(t)=\int V_i dt+v_\infty . \tag{38}$$

If we have (EQ blocks in Fig. 7):

$$E_3-E_2=\frac{E_4}{10^{12}} \tag{39}$$

and:

$$E_{2,3}<<E_4 , E_4=10^{12}(E_3-E_2) \tag{40}$$

giving:

$$E_3=E_2 \tag{41}$$

The PSPICE program has 4 block levels.

INPUT DATA block: using controlled sources we have eight parameters:

$\pi_1, \tau_1, \omega_1, \beta_1, \pi_2, \tau_2, \omega_2, \beta_2$.

INT blocks: compute the integrator blocks, here are the initial conditions.

CPU block: here we have multiplication, addition and return E_{c1} and E_{c11} .

EQ blocks: close the loops, with relations $X_1=E_{c1}$ and $X_2=E_{c11}$.

The dimensionless forms for the PSPICE simulation are:

$x_1''=E_{c1}$

$$\begin{aligned}E_{c1}&=x_1''(x_1+x_2)^2+\pi_1\left[1-\beta_1-(x_1+x_2)^2\right]x_1'-\\&-\omega_1^2x_1+\pi_1\left[\tau_1-\beta_1-\tau_1(x_1+x_2)^2\right]x_2'\end{aligned} \tag{42}$$

$x_2''=E_{c11}$

$$\begin{aligned}E_{c11}&=x_2''(x_1+x_2)^2+\pi_2\left[1-\beta_2-(x_1+x_2)^2\right]x_2'-\\&-\omega_2^2x_2+\pi_2\left[\tau_2-\beta_2-\tau_2(x_1+x_2)^2\right]x_1'\end{aligned} \tag{43}$$

V. Results of Numerical Integration

Using the PSPICE program we compare the theoretical results with the result of numerical computation.

Over 200 runs with several initial conditions prove that we can have a stable oscillation for double and multiple-mode ECAM. The general prediction error is 10 %.

VI. Conclusions

In this paper were introduced a new and improved PSPICE method for simulating linear and nonlinear equations.

An analytic approximation to the periodic solutions for the single and double-mode LCR networks of E.C.A.M. is obtained.

A PSPICE method has been proposed to solve the nonlinear differential equations.

Over 200 sets of initial conditions and parameters prove that the obtained results were in a good agreement with theoretical predictions.

References

[1] S. S. Walker, J. A. Connelly, “A new negative resistance oscillator model”, Circuits, Systems and Signal Processing 2, pp. 213-238, 1983

[2] “Investigation of the mathematical properties of a new negative resistance oscillator model”, Circuits, Systems and Signal Processing 2, pp. 187-205, 1992

[3] ***, PSPICE, Microsim Corporation, Irvine, SUA, 1992

[4] N. Minorsky, “Introduction to Non-Linear Mechanics”, J.W.Edwards, Ann Arbor, Michigan, 1947.

[5] “Nonlinear ordinary differential equations”, G. P. Flessas, P. G. L. Leach, 2005.