International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-11, Issue-04, July 2025

Spice Models of Time Varying Resistances Under Control Voltage

Purajit Mishra, Principal, Spintronic Technology and Advance Research, Bhubaneswar, INDIA, purajitmishra@gmail.com

Brahmananda Das, HOD EEE, Spintronic Technology and Advance Research, Bhubaneswar, INDIA, Brahman315@gmail.com

Abstract—Spice models of controlled resistances that change over time according to a given law under the influence of a control voltage are proposed. A SPICE model is a text-description of a circuit component used by the SPICE Simulator to mathematically predict the behavior of that part under varying conditions. Models can operate in the modes of a given current or a given voltage. The presented models of controlled resistances can also be used to create models of controlled capacitances and inductances. Models of controlled elements can be part of models of multi-element electrical circuits. Model inclusion diagrams and simulation results are presented.

Key words—Spice models of controlled resistances, modes of given current and given voltage, simulation.

I. INTRODUCTION

Dynamic measurements of physical quantities, the size of which changes during measurement are increasingly used. The dynamic measurement, is a method in which data is recorded over a specified time interval, allowing the change in the value of the measured quantity to be studied at different points in time, but in static measurement, the data is obtained about a physical quantity at a specific point in time. The data obtained from dynamic measurement can provide greater insight into a process or phenomenon. The range of sensors with characteristics allowing for dynamic measurements is expanding. Examples include various pressure sensors [1], which convert changes in time of a physical quantity - pressure - into changes in time of sensor parameters: electrical resistance, capacitance or inductance. Using measuring transducers based on operational amplifiers [2], [3], time-varying sensor parameters are converted into electrical signals. In this case the parameters of the signals, for example, the instant values of frequency or voltage amplitude, are proportional to the time varying physical quantity. The time-varying resistance can be included in theinput circuit of the operational amplifier of the measuring transducer or in its negative feedback circuit, so models of time-varying resistances operating in different energy modes: a given voltage or a given current are required. To reproduce the required laws of change in resistance over time, it is necessary to generate the appropriate control voltages. The purpose of this work is to create Spice models

II. METHOD

The controlled resistance model R2 (t) in the given current mode (in Figure 1 is highlighted by the dotted line) is included in the negative feedback circuit of the operational amplifier, represented by the macro model X1. Model R2 (t) is obtained as a result of a series connection of the source resistance and a voltage source controlled by voltage E1. (Voltage-controlled voltage source).

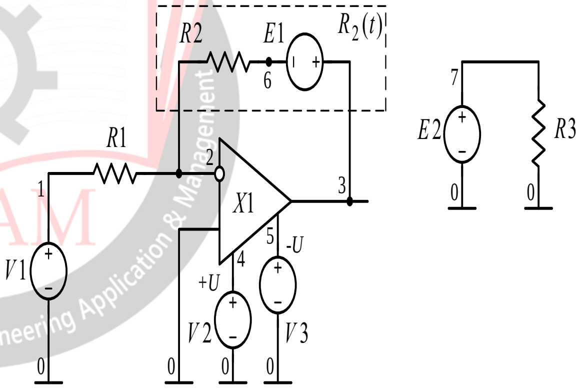

Fig. 1: Model of an operational amplifier with controlled resistance in given current mode.

The output voltage of the operational amplifier $U_{out}(t)$ is determined by the voltage drop across the resistance $R_2(t)$, which changes over time as the voltage $U_{E1}(t)$ of the source $E1$ changes under the influence of the control voltage $U_{E2}(t)$ of the source $E2$. The required law of change in resistance over time can be reproduced by specifying the corresponding analytical description of the change in time of the control voltage $U_{E2}(t)$:

 DOI : 10.35291/2454-9150.2025.0133

$R_2(t) = R2 \cdot U_{E2}(t)$ ....(1)

Therefore, the controlled resistance can be considered as an Impedance Converter in voltage [6]. The output voltage of the operational amplifier:

$$U_{out}(t) = \frac{U_{R_2}}{R2} \cdot R_2(t) = U_{R2} \cdot U_{E2}(t) = U_{R2} - U_{E1}(t) \quad (2)$$

Output voltage of the source E1:

$$U_{E1}(t) = U_{R2} - U_{R2} \cdot U_{E2}(t) \ldots\ldots \quad (3)$$

The source E2 in the program PSpice [5] is described by:
E2 7 0 VALUE= < expression>}(4)

As follows from Figure 1 and formula (3), the voltages controlling the source $E1$ are the voltages U_{R2} between nodes 2 and 6 and $U_{E2}(t)$ between nodes 7 and 0.

$$U_{E1} = P_0 + P_1 \cdot V_{2,6} + P_2 \cdot V_{7,0} + P_3 \cdot V_{2,6}^2 + P_4 \cdot V_{2,6} \cdot V_{7,0} \quad (5)$$

At, $P_0 = 0$; $P_1 = 1$; $P_2 = 0$; $P_3 = 0$; $P_4 = -1$:

$$U_{E1} = V_{2,6} - V_{2,6} \cdot V_{7,0} \quad (6)$$

The source $E1$ is described by the sentence:

E1 3 6 POLY(2) (2,6) (7,0) 0 1 0 0 –1. (7)

from zero to 1ms. The As an example, the model of a resistor $R_2(t)$ is considered, the resistance of which varies linearly from 1 kΩ to 10 k Ω in a time interval

$$R_2(t) = 1 \cdot 10^3 + 9 \cdot 10^6 \cdot t$$

Let the initial resistance be R_2 = 10 Ω. Then

$R_2(t) = (0.1 + 0.9 \cdot 10^3 \cdot t) \cdot R2$; $U_{E2}(t) = 0.1 + 0.9 \cdot 10^3 \cdot t$.

Figure 2 shows the results of simulation of the considered circuit when a pulse of negative polarity with an amplitude of 5 V is applied to the input; $R1 = 10$ kΩ; operational amplifier – AD823. The dotted line shows the output voltage of the amplifier when the initial resistance R_2 = 10 kΩ is connected to the negative feedback circuit; solid line – when the controlled resistance $R_2(t) = (0.1 + 0.9 \cdot 10^3 \cdot t) \cdot R2$is turned on

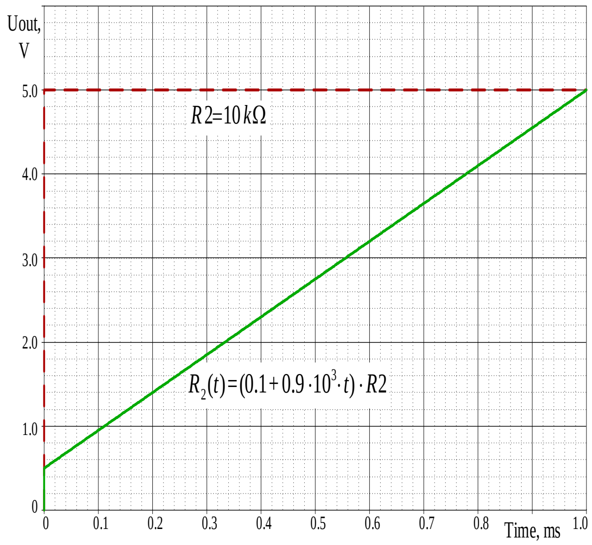

Figure 2 – Results of simulation of an amplifier with a controlled resistance in the given current mode

The control voltage $U_{E2}(t)$ can be specified not only by an analytical expression, but also obtained at the output of a controlled source, the transfer function of which is specified using the Laplace transform. The output voltage of such a source according to the .TRAN directive is calculated as the convolution of the input effect with the impulse response of the source (Duhamel integral)[8],[10]. To form the input actionin theform of a unit step function, an additional voltage source is introduced.

The model of controlled resistance $R_1(t)$ in thegivenvoltage mode (in Figure 3 is highlighted with a dotted line) is included in the input circuit of the operational amplifier. The model is obtained as a result of parallel connection of the original resistance and the current-controlled current source, F1 [7],[9].

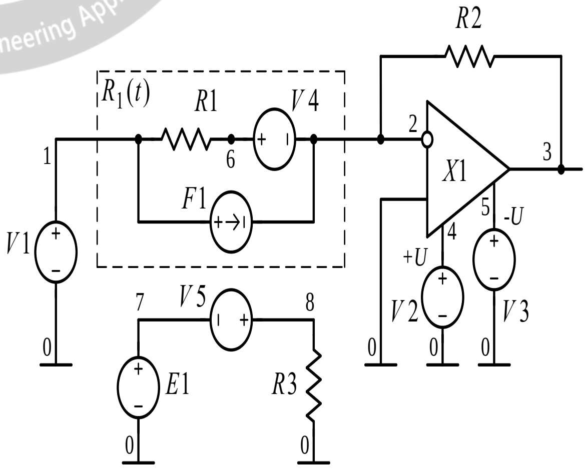

Figure 3 – Model of an amplifier with controlled resistance in given voltage mode

The output voltage of the operational amplifier $U_{out}(t)$ is determined by the current through the resistance $R_1(t)$, which changes over time as the current

 DOI : 10.35291/2454-9150.2025.0133

$I_{F1}(t)$ of the source F1 changes under the influence of the current $I_{E1}(t)$ of the control voltage source E1. The required law of change in time of resistance $R_1(t)$ can be reproduced by specifying the corresponding analytical description of the change in time of control current $I_{E1}(t)$

$$R_1(t)=\frac{R1}{I_{E1}(t)} \qquad (8)$$

Therefore, the controlled resistance $R_1(t)$ can be considered as an Impedance Converter for current [6].

Output current of the source F1:

$$I_{F1}(t)=I_{R1}\cdot I_{E1}(t)-I_{R1} \qquad (9)$$

The control current $I_{E1}(t)$ is generated using a voltage source *E*1 loaded onto a unit resistance *R*3. Current-controlled current source *F*1 can only be controlled by currents from independent voltage sources [5]. Therefore, to implement (9), the model includes independent voltage sources *V*4 and *V*5 with zero EMF, used as "current sensors": $I_{V4}=I_{R1}$ and $I_{V5}=-I_{E1}$.Output current of the source*F*1:

$$I_{F1}=-I_{V4}-I_{V4}\cdot I_{V5} \quad \ldots\ldots\ldots\ldots(10)$$

Source F1 is described by the sentence:

F1 1 2 POLY(2) V4 V5 0 –1 0 0 –1 (11)

Let us consider the model of a resistor $R_1(t)$, the resistance of which is described by the same expression as in the previous example:

$$R_1(t)=1\cdot 10^3+9\cdot 10^6\cdot t$$

Let the initial resistance be 10 kΩ. Then the output voltage of source *E*1 should be described by the expression:

$$U_{E1}(t)=\frac{R1\cdot R3}{R_1(t)}=\frac{1}{0.1+0.9\cdot 10^3\cdot t}$$

Figure 4 shows the results of simulating the circuit presented in Figure 3 under the same conditions as in the previous example.

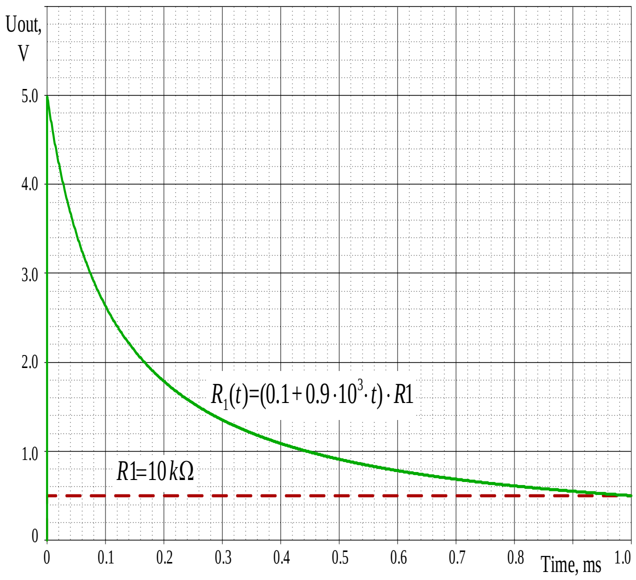

Figure 4 – Results of simulation of operational amplifier with a controlled resistance in the given voltage mode.

The presented models of controlled resistors can also be used to create models of controlled capacitances and inductances operating in the modes of a given current or a given voltage. As an example, let us consider a model of a capacitance converter during the period of electrical oscillations (Figure 5). The circuit is a relaxation self-oscillator, which includes an integrator and a comparator.

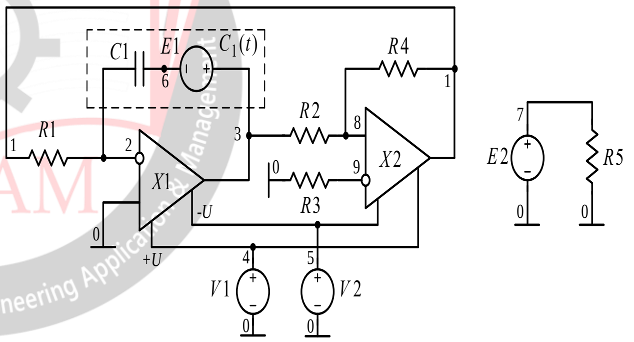

Figure 5 – Model of capacitance converter during the period electrical vibrations

The capacitance of a capacitor connected in the negative feedback circuit of the integrator operational amplifier changes over time under the influence of the control voltage $U_{E2}(t)$:

$$C_1(t)=\frac{C1}{U_{E2}(t)} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots(12).$$

The period of electrical oscillations in the circuit changes in proportion to the change in capacitance $C_1(t)$. At :

$$R2=R4$$

$$T(t)=2\cdot R1\cdot C_1(t) \qquad (13)$$

 DOI : 10.35291/2454-9150.2025.0133

ISSN : 2454-9150 Vol-11, Issue-04, July 2025

functional dependence of resistance on time is described by the expression:

Figure 6 shows the timing diagrams of the output voltages of the integrator (macromodelX1) and comparator (macromodel X2), built on AD823 operational amplifiers. At $R1=R2=R4=$20kΩ, $R3=3$ kΩ, $C1=10\cdot10^{-9}$ F, $U_{E2}(t)=0.2+0.2\cdot10^{3}\cdot t$ in the time interval from zero to 6 ms, the controlled capacitance $C_1(t)$ decreases from $50\cdot10^{-9}$ F to $7.143\cdot10^{-9}$ F.Accordingly, the period of electrical oscillations in the circuit decreases.

Models of controlled elements can be part of models of multi-element electrical circuits.

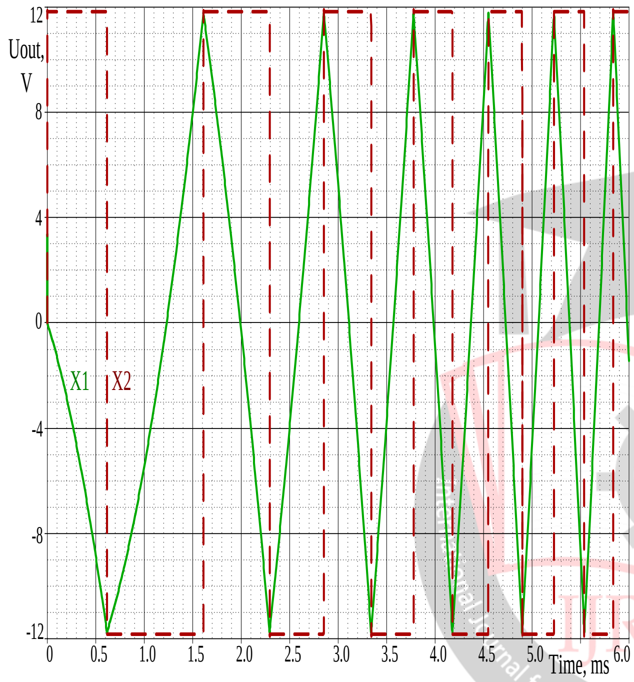

Figure 6 – Simulation results of capacitance converter during the period of electrical oscillations

III. CONCLUSION AND FUTURE SCOPE

The given examples demonstrate the possibility of using the proposed Spice models of controlled elements to simulate measuring transducers of object parameters that change over time according to a given law under the influence of a control voltage .The models of controlled resistors can also be used to create models of controlled capacitances and inductances operating in the modes of a given current or a given voltage.

IV. REFERENCES

[1]. Jan Hjelmgren. Dynamic Measurement of Pressure - A Literature Survey. SP Swedish National Testing and Research Institute.Report2002:34. http://www.vibrationdata.com/tutorials/2002_34.pdf.

[2]. Bruce Carter and Ron Mancine. Op Amps for Everyone. Third Edition. – Newnes, Texas Instruments Incorporated, 2008.

[3]. Leon O. Chua, Charles A. Desoer, Ernest S. Kuh. Linear and Nonlinear Circuits. – McGraw-Hill Book Company, 1987.

[4]. John Keown. OrCADPSpice and Circuit Analysis. – Prentce Hall. Upper Saddie River, New Jersey, 2001.

[5]. OrCAD PSPICE A/D. User’s Guide. Reference Manual. OrCAD, Inc.,1998.

[6]. J.-C. Marchais. L’AmplificateurOperationnel et Ses Applications Masson et Cie, Editeurs. Paris. 1971.

[7].Kazakov, V.A.; Svetlov, A.V. Modelling measuring transducers of current consumption for active components installed on printed circuit boards. Measurement Techniques, 2004, Vol. 47, 9, pp 932-935.

[8] ershin & Di Ventra’s 2012 paper **“SPICE model of memristive devices with threshold”**

[9] Krasnosel'Skii MA, Pokrovskii AV. *Systems with Hysteresis*, 1st edn. Springer-Verlag: Berlin Heidelberg, 1989.

[10] Hong DS, Chen YS, Sun JR, Shen BG. Ternary synaptic plasticity arising from memdiode behavior of TiOx single nanowires. *Advanced Electronic Materials* 2016; **2**(4): 1-7.

 DOI : 10.35291/2454-9150.2025.0133