Published at the 23rd IEEE Israel Convention, pp. 338-341, Tel-Aviv, 2004

A SPICE BEHAVIORAL MODEL FOR CURRENT-CONTROLLED MAGNETIC INDUCTORS

*Evgeny Rozanov and Sam Ben-Yaakov**

Power Electronics Laboratory, Department of Electrical and Computer Engineering
Ben-Gurion University of the Negev, P.O. Box 653, Beer-Sheva 84105, ISRAEL.
Phone: +972-8-646-1561; Fax: +972-8-647-2949; Email: sby@ee.bgu.ac.il
Website: www.ee.bgu.ac.il/~pel

ABSTRACT

A new SPICE-compatible behavioral model for magnetic controlled inductors was developed and verified experimentally. The new model is based on the gyrator-capacitor scheme in which the nonlinear permeance is emulated by a non-linear capacitor that is built by using a 'transformer' circuit. The gyrator-capacitor approach can be used to evaluate an arbitrary magnetic circuit, and the non-linear 'transformer' circuit makes possible the modeling the nonlinearity of the ferromagnetic core. The core saturation and hysteresis are reproduced by applying the core manufacturer's data. A current-controlled inductor was used to test the proposed model. Close agreement was obtained between the simulation and experimental results.

1. INTRODUCTION

SPICE simulation of electronic circuits that include controlled magnetic devices is a nontrivial task. The difficulty stems from the fact that SPICE is primarily oriented toward electronic circuits (not magnetic circuits) simulations. Because of this, SPICE libraries do not offer built-in models for simulating complex magnetic elements beyond a simple inductor or transformer. Consequently, devices such as a controlled inductor [1] in which the outer legs of an E core are forced into saturation by bias windings can not be simulated by PSPICE. One possibility to overcome this obstacle is to represent such devices by SPICE compatible equivalent electric models.

The objective of the present study was to develop a SPICE compatible model of integrated magnetic devices where the core properties can be described either analytically or by data tables that are based on manufacturers' data.

2. SPICE-COMPATIBLE MAGNETIC MODEL

The required equivalent circuit model has to imitate the storing, dissipation, and interchanging of energy flow in the electronic circuit. Thus, to define the equivalent electric model of an arbitrary magnetic circuit, it is necessary to complete the two following tasks.

The first task is to develop a magnetic circuit model that can imitate the energy storing, while keeping the correct relationships between the electric equivalents of flux, magnetomotive force and magnetic core properties.

The second task is to model energy interchange between the model of the magnetic circuit and the rest of the original electric circuit. To do this, it is necessary to evaluate the Ampere and Faraday laws for the electric equivalents of the magnetic properties, on the one hand, and for the voltages and currents of the windings, on the other hand.

An elegant solution to the above tasks is the gyrator-capacitor-model (GCM) [2,3]. It suggests employing a capacitor for emulation the magnetic energy storage element. The magnetic energy E_{mf} stored by any branch of magnetic circuit, can be expressed in terms of the flux Φ and the permeance P:

$$E_{mf} = \frac{\Phi^2}{2P} \qquad (1)$$

On the other hand, the storage of electric field energy E_{ef} capacitor can be expressed in terms of the electric charge Q and capacitance C:

$$E_{ef} = \frac{Q^2}{2C} \qquad (2)$$

The analogy between electrical and magnetic parameters in (1) and (2) is evident. The rest of equivalency can be found from the set of the following well-known equations:

$$P = \frac{\Phi}{mmf} = \frac{\int \frac{\partial \Phi}{\partial t} dt}{mmf} \qquad (3)$$

where $mmf = H \cdot l_e$ is the magnetomotive force, H is the magnetic field strength and l_e is effective magnetic branch length.

* Corresponding author

The analogous electrical equation is:

$$C = \frac{Q}{V_C} = \frac{\int \frac{dQ}{dt} dt}{V_C} \quad (4)$$

where Q and V_C are the charge and voltage of the capacitor C, respectively.

To emulate electromagnetic link between the voltage, current and magnetic field of the windings the model must reflect the Faraday and Ampere laws:

$$\frac{\partial \Phi}{\partial t} = -\frac{V}{n} \quad (5)$$

$$I = \frac{mmf}{n} \quad (6)$$

Based on the above, the electrical equivalent circuit of a magnetic circuit can be developed as follows. From (1), (2):

$$C \equiv P \quad (7)$$

$$Q \equiv \Phi \quad (8)$$

From (3)

$$mmf = P \cdot \Phi \quad (9)$$

$$V_C = C \cdot Q \quad (10)$$

and hence (7)-(10)

$$V_C \equiv mmf \quad (11)$$

Defining

$$I_C \equiv \frac{\partial \Phi}{\partial t} \quad (12)$$

one can translate (5) into

$$I_C = -\frac{V}{n} \quad (13)$$

and from (6) and (11)

$$I = \frac{V_C}{n} \quad (14)$$

Consequently, equations (7), (13), (14) can be used to emulate the magnetic circuit of Fig. 1a by a gyrator that is loaded by a capacitor of value P (Fig. 1b).

3. MODELING OF NON-LINEAR MAGNETIC PROPERTIES

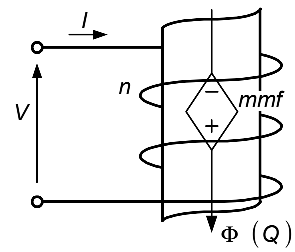
A prerequisite for simulating a magnetic circuit that applies practical magnetic cores, is the ability to take into account the non linearity of the permeances.

From (3):

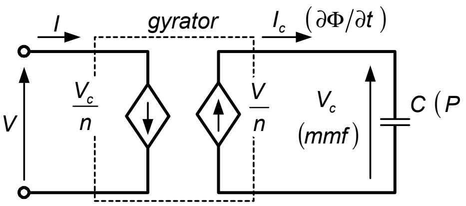
$$P_{ei} = \frac{\Phi_i}{mmf_i} = \frac{\Phi_i}{H_i \cdot l_e} \quad (15)$$

Eq. (15) implies that the permeance at a given operating point is a function of the non linear ratio B_i / H_i. This non ideality is emulated in the proposed approach by making the capacitor of the equivalent circuit (Fig. 1b) non-linear.

The proposed work suggests a simple approach



(a)



(b)

Fig. 1. A simple magnetic branch (a) and its equivalent gyrator-capacitor model (b).

modeling a controlled non-linear capacitor. This model utilizes a non-linear 'transformer' [4] with a capacitive load (Fig. 2a), where the input terminals of 'transformer' emulate a non-linear capacitor behaviour.

The capacitor value reflected to the primary C_{eq} can be evaluated as follows. Since :

$$C = \frac{I}{\frac{dV}{dt}} \quad (16)$$

Then:

$$C_{eq} = \frac{I}{\frac{d(kV)}{dt}} = \frac{1}{k}\frac{I}{\frac{dV}{dt}} \quad (17)$$

where k is the 'transformer' scale factor.

From (17):

$$C_{eq} = \frac{1}{k} C \quad (18)$$

If C is unity then:

$$C_{eq} = \frac{1}{k} \quad (19)$$

In order for the proposed model of controlled capacitor C_{eq} to emulate the behaviour of the magnetic branch permeance the scale factor k_i (Fig. 2b) needs to be fitted to the core data:

$$P_{ei} = \frac{1}{k_i} \quad (20)$$

From (15) and (20):

$$k_i = \frac{\{H_i(\Phi_i / A_e)\}}{\Phi_i} \quad (21)$$

where the notation $\{H_i(\Phi_i/A_e)\}$ implies that H_i is a function of B_i and hence of (Φ_i/A_e). Therefore this

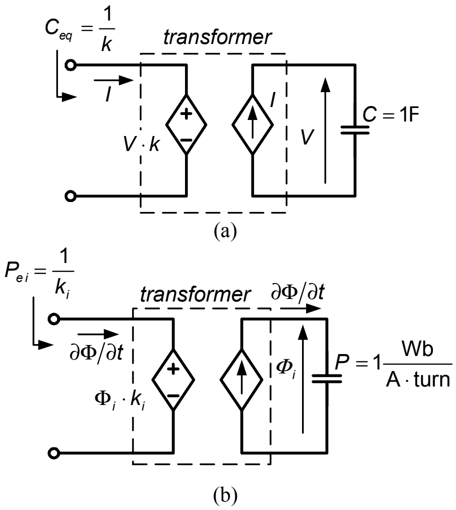


Fig. 2. A variable capacitor model (a) and its application to emulate a non-linear permeance (b).

relationships can be obtained from branch's effective parameters (A_e , l_e) and the manufacturers data (in form of table or fitted experimental equation) of the magnetic field strength H_i vs. magnetic flux density B_i of the core.

Applying (11), (15), (21) the input transformer voltage source $\Phi_i k_i$ (Fig. 2b) can be replaced by:

$$V_C = \frac{\Phi_i}{P_{eq}} = \Phi_i k_i = \{H_i(\Phi_i / A_e)\} \cdot l_e = \{H_i(B_i)\} \cdot l_e \quad (22)$$

since $B_i = \Phi_i / A_e$.

The table $\{B_i, H_i\}$ is identical for all branches that are built from the same magnetic material and can be obtained from this material manufacturer's data.

To realize the controlled capacitor C_{eq} in PSPICE environment, the input 'transformer' voltage source can be partitioned into two dependent voltage sources (Fig. 3). The first one (ETABLE) samples the capacitor charge Q_{eq} which is equal to V_{FLUX} since $P = 1\,\mathrm{Wb/A \cdot turn}$. That is, V_{FLUX} represents Φ_i. The expression of the ETABLE V_{FLUX}/A_e converts Φ_i to B_i. The stored look up table then converts B_i into H_i which is represented by the voltage V_H. The second source (EVALUE) samples the voltage V_H and multiplies it by l_e to produce voltage V_C (i.e. *mmf*).

The complementary part of the "transformer" that feeds the input current the capacitor P is realized by a behavioral dependent current source GVALUE.

Fig. 3 is thus a representation of a non linear permeance of a single magnetic branch. To model a non-linear arbitrary magnetic circuit one would need the values of effective parameters of all branches and a single

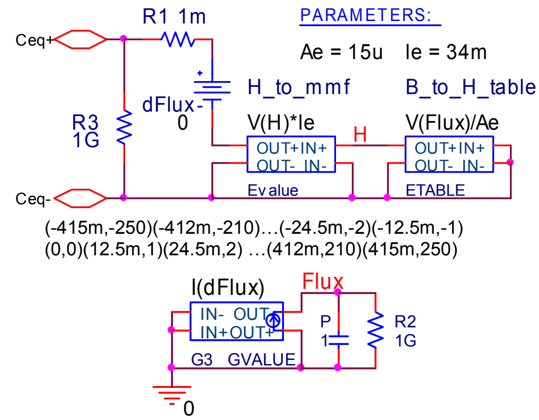


Fig. 3. PSPICE (CADENCE Eval. Version 9.2) model of non-linear capacitor, which emulates relationships between a magnetic flux Φ (an electric charge of capacitor) and the mmf (the input terminals voltage). Branch parameters: $A_e = 15\,\mu\mathrm{m}^2$ and $l_e = 34\,\mathrm{mm}$. Core material: 3F3 [5].

H vs. B look-up-table of the magnetic material (assuming that all branches are made from same material).

To eliminate a convergence problem in SPICE simulations, the model incorporates some extra resistances R1, R2 and R3. R2 provides a galvanic path to ground while R1 allows the connection of the node to a voltage source. It should be noted that by proper sizing R1 one can implement hysteretic behavior [2,3].

4. THE CURRENT-CONTROLLED INDUCTOR

A current-controlled inductor [1] was used to verify the proposed model. The inductor is built around a double 'E' core structure (Fig. 4). The basic idea of this device is to control the center-arm-wound-inductor by altering the side arms permeances. For this purpose, the side arm windings are driven by the dc current source I_{BIAS} .

The proposed equivalent SPICE model (Fig. 5) of the current-controlled inductor includes three sub-circuits (Left_leg, Right_leg, Center_leg), which emulate the non-linear behavior of magnetic core arms. The structure of each sub-circuit is equal to the circuit depicted in Fig. 4. The permeance of air gap of the center arm is emulated by a constant capacitor C4. The value of this capacitance is found from (3). The excitation of the center-arm-wound-inductor is modeled by a gyrator that incorporates two dependent current sources (G1 and G2). The gyration resistance n emulates the number of turns of the controlled inductor. The bias current drive of the side arm is modeled by adding to side arms the appropriate magnetomotive forces taking into account their number of turns n_bias . This is accomplished by two dependant voltage sources, which reflect the input bias current into appropriated magnetomotive forces of the side arms.

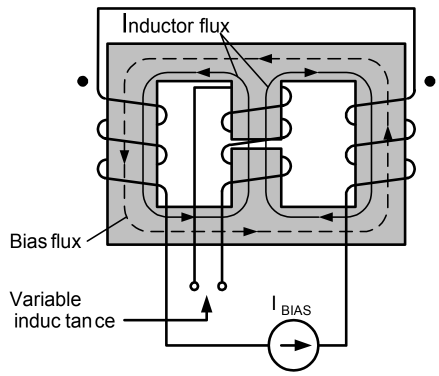


Fig. 4. Winding arrangements of the a current-controlled inductor [1].

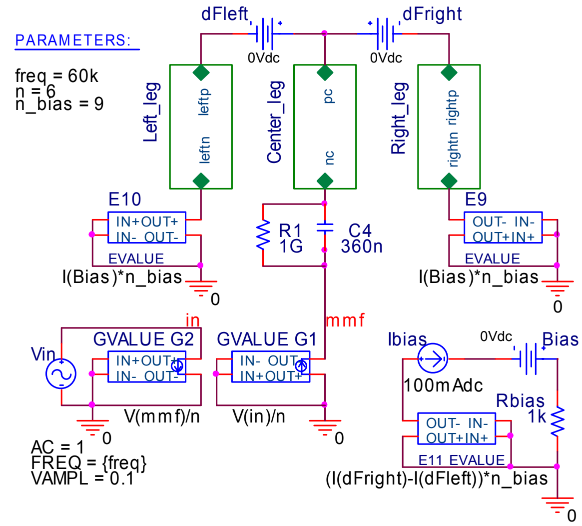


Fig. 5. PSPICE model of the current-controlled inductor of Fig. 4.

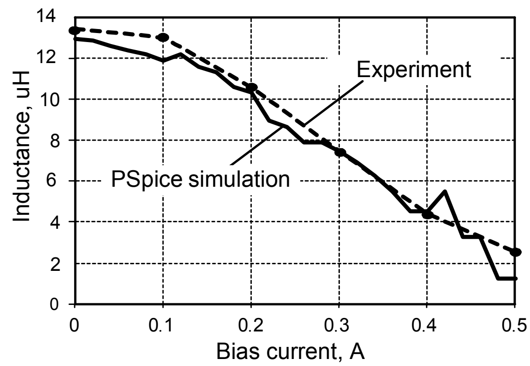


Fig. 6. Simulated and measured inductance of the current–controlled inductor versus bias current.

The proposed model was verified by experimental measurements of inductance vs. bias current of a current-controlled inductor built around an EFD15. The initial center arm inductance (no bias) was core $L_0 = 13\mu H$; the number of turns were: $n = 6,\ n_{bias} = 9$. Close agreement was obtained between the simulated and experimental results (Fig. 6).

5. DISCUSSION AND CONCLUSIONS

The main advantage of the proposed approach is that it makes possible to represent any given relationship between charge (flux) and capacitance (permeance). For close approximation of permeance vs. magnetic flux, the look-up-table can be fitted to the magnetic core manufacturer data. As a result, a good agreement between the simulations and experiments can be achieved.

The discrepancy between the measured and simulated results is probably due to the inaccuracy of the data points and the fact that they do not represent a smooth curve. The experiment and simulation involved small signal response, that is, measuring and simulating the incremental inductance for given bias currents. The AC analysis needed to be carried out by PSPICE requires the simulator to carry out linearization around the operating point. Since any irregularity in the H-B table will manifest itself as a large error in the derivative, a non smooth data may result in relatively large errors. This could be overcome by fitting the data to a smooth experimental curve.

Notwithstanding the above, the proposed modeling and simulation approach is potentially a viable tool to study the behavior of complex integrated magnetics structures.

ACKNOWLEDGMENT

This research was supported by THE ISRAEL SCIENCE FOUNDATION (grant No. 113/02) and by the Paul Ivanier Center for Robotics and Production management.

REFERENCES

[1] D. Medini and S. Ben-Yaakov, "A current controlled variable inductor for high frequency resonant power converters", in *IEEE Applied Power Electronics Conference, APEC'94*, pp. 219-225, Orlando, 1994.

[2] D. C. Hamill, "Gyrator-capacitor modeling: a better way of understanding magnetic components", in *IEEE Applied Power Electronics Conference, APEC'94*, pp. 326-332, Orlando,1994.

[3] D. Cheng, L. P. Wong, and Y. S. Lee, "Design, modeling, and analysis of integrated magnetics for power converters", in *IEEE Power Electronics Specialists Conference, APEC'00*, pp.320-325, New Orleans, 2000.

[4] S. Ben-Yaakov and M. Peretz, "Simulation bits: a SPICE behavioral model of non-linear inductors", *IEEE Power Electronics Society NEWSLETTER*, Fourth Quarter 2003.

[5] "Data handbook soft ferrites and accessories, Ferroxcube", Available via the Internet link: http://www.ferroxcube.com/appl/info/HB2002.pdf