A NON-LINEAR SEMICONDUCTOR COUPLED TO FEM

Silvio Ikuyo Nabeta* Kay Hameyer** Herbert De Gersem**

*Escola Politécnica da Universidade de São Paulo, Brazil
**Katholieke Universiteit Leuven, Belgium

***Abstract* — This paper presents a non-linear model for a diode coupled to the Finite Element Method (FEM). This coupling is applied to a 2D time-stepping analysis of an inductor and the result is compared to those from the *HSpice* package.**

I. INTRODUCTION

The coupling of electric circuits in Finite Element Analysis was a major development for the electromagnetic fields computation. One of the greatest achievements of this coupling is the consideration of semiconductors in the analysis.

In general, there are two different approaches to couple the Finite Element Method (FEM) with electric circuits. In the first approach, namely Direct Coupling, the equations from the FEM and the electric circuit analysis are assembled together and the system is solved simultaneously. In the second approach, known as Indirect Coupling, the FEM and the electric circuit systems are separate and the solution is achieved by an iterative method.

The aim of this work is to couple a non-linear model of a diode with the Finite Element Method by an Indirect Coupling.

With the aid of this coupling a Time-Stepping Finite Element Analysis of an inductor and its circuit is accomplished and the results are compared to those of an on-off model and also to those issued from the *HSpice* package.

II. DIODE MODELLING

In this work two models for a diode are presented namely the *on-off model* and the *non-linear model*. For each model a Time-Stepping FEA is carried out and the results of the current are compared.

On-Off Model

In the on-off model the diode is considered as a perfect switch as presented in Fig. 1. No voltage drop is considered in the switch.

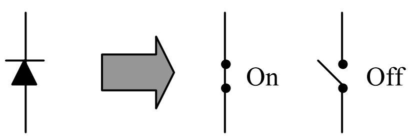
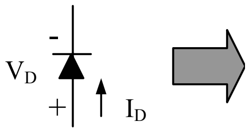


Figure 1: On-Off Model

This work was partially supported by FAPESP-Fundação de Amparo à Pesquisa do Estado de São Paulo (Project n° 00/08790-0).

Non-Linear Model

The non-linear model takes into account the non-linear relation between the voltage and the current in the diode as presented in (1) [1].



$$V_D = V_T \ln\left(\frac{I_D}{I_S} + 1\right) + R_S I_D \qquad (1)$$

Where: V_T is the thermodynamic voltage;
I_S is the reverse saturation current;
R_S is the internal resistance.

III. THE FINITE ELEMENT PACKAGE

The finite element package used was the *Olympos* [2] an Unix based software developed by the K.U. Leuven team.

Olympos provided the 2D time-stepping analysis with electric circuit coupling.

The generalized time-stepping scheme with fixed time step is given by (2) [2]

$$\left(\alpha K + \frac{R}{\Delta t}\right)A_k + \left((1-\alpha)K - \frac{R}{\Delta t}\right)A_{k-1} - \left(\alpha T_k + (1-\alpha)T_{k-1}\right) = 0 \qquad (2)$$

Where: K is the element coefficient matrix;
R is the stiffness matrix;
T_{k-1} and T_k are the source vectors at $t=t_{k-1}$ and $t=t_k$, respectively;
α is a weighting factor

IV. PROBLEM UNDER ANALYSIS

The problem under analysis consists of an inductor fed by a voltage source through a rectifier as shown in Fig.2.

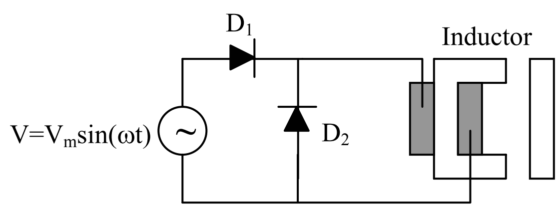


Figure 2: FEM and Circuit Coupling Problem

The inductor has a coil with 100 turns and a linear ferromagnetic core with $\mu_r = 1000$.

The diodes D_1 and D_2 are modeled, in the non-linear case, by (1) with parameters of the commercial type MBR4015 CTL [4].

The voltage source has a sinus waveform and the time step was fixed to 0.5 ms.

V. DIODE MODEL AND FINITE ELEMENT COUPLING

The coupling of the *Olympos* package and the diode model was accomplished by using the *Matlab* [3] software as a manager.

On-Off Model

In this model the diodes D_1 and D_2 were considered as an ideal switch, i.e., without voltage drops on them.

Figure 3 presents schematically the coupling and the manager function.

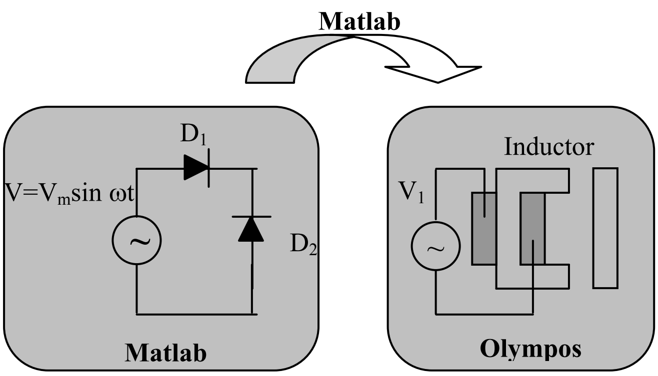


Figure 3: On-Off Model and *Olympos* Coupling

For the *On-Off Model* the Matlab verified the conducting conditions of D_1 and D_2 and applied the corresponding voltage in the terminals of the inductor (V_1) as presented in Table I.

Table I

	Value of V_1
D_1 On & D_2 Off	$V_m \sin \omega t$
D_1 Off & D_2 On	0

Non-Linear Model

An iterative method was applied due to the non-linearity of the diodes model.

Figure 4 presents the iterative scheme used for the non-linear model.

For each time step the currents for both circuits (rectifier and inductor) were calculated until a convergence was achieved.

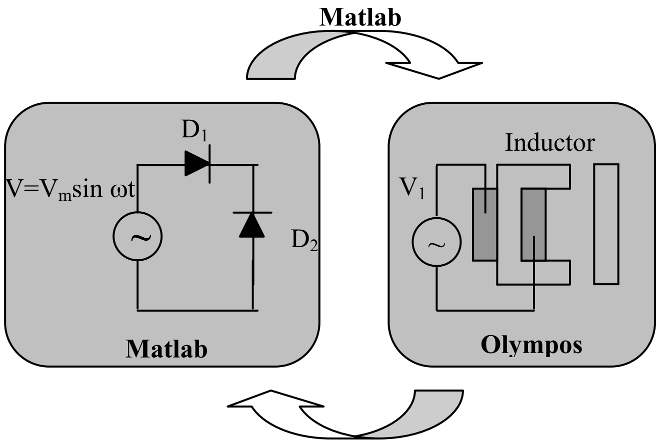


Figure 4: Non-Linear Model and *Olympos* Coupling

Table II presents the value of V_1 for each time step.

Table II

	Value of V_1
D_1 On & D_2 Off	$V_m \sin \omega t - V_D$
D_1 Off & D_2 On	$-V_D$

And the drop-voltage V_D was calculated by using (1).

VI. NUMERICAL SIMULATIONS

The proposed problem consists of an inductor fed by a voltage source through a rectifier represented by the diodes D_1 and D_2, as shown in fig.2.

The inductor core was considered linear with a relative permeability $\mu_r = 1000$. The coil had 100 turns and the voltage source was $V=10.\sin \omega t$, with f=50 Hz.

The diodes D_1 and D_2 were the commercial type MBR4015 CTL from ON Semiconductors which parameters were found in [4].

The following time-stepping analysis were carried out in the problem above:

1. FEM and ON-Off model for D_1 and D_2;
2. FEM and Non-Linear model for D_1 and D_2;

The discretization of the inductor geometry provided a mesh with 342 nodes.

A linear magnetodynamic FEM simulation was previously carried out on the inductor in order to determine its values of resistance *R* and inductance *L*. Figure 5 shows the inductor geometry coupled to the electric circuit used in this simulation.

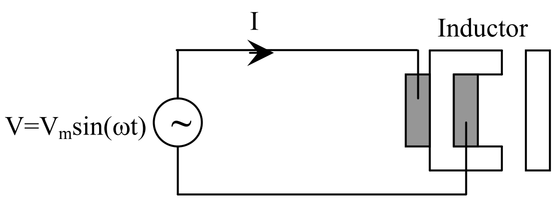


Figure 5: Scheme used for the magnetodynamic simulation

Assuming $\overline{V}$ and $\overline{I}$ as the fasor notation of V and I, the impedance $\overline{Z} = (R + j\omega L)$ can be found easily by (3).

$$\overline{Z} = \frac{\overline{V}}{\overline{I}} \qquad (3)$$

And the obtained values are L=13.96 mH and R=0.17 Ω for its inductance and resistance, respectively. Those values were used in the *Hspice* simulation as described in the next section.

VII. *HSPICE* SIMULATION

In order to compare the results of the diode and FEM coupled simulations, a *Hspice* analysis was carried out in the circuit shown in fig.6.

The values of the resistance R and the inductance L were determined by a linear magnetodynamic simulation by using *Olympos*.

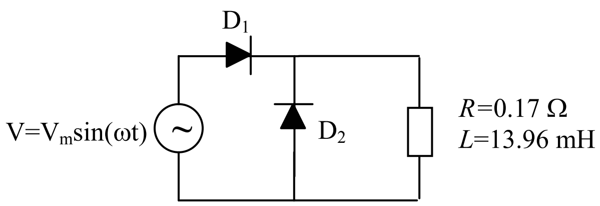


Figure 6: Circuit used in *HSpice*

The diodes D_1 and D_2 were modeled by using the Spice model given by the manufacturer in the web site [4] for the diode MBR4015 CTL.

VIII. RESULTS

The quantity analyzed is the current in the inductor.

Figure 3 presents the results comparison for the current in the inductor for the three analyses mentioned above.

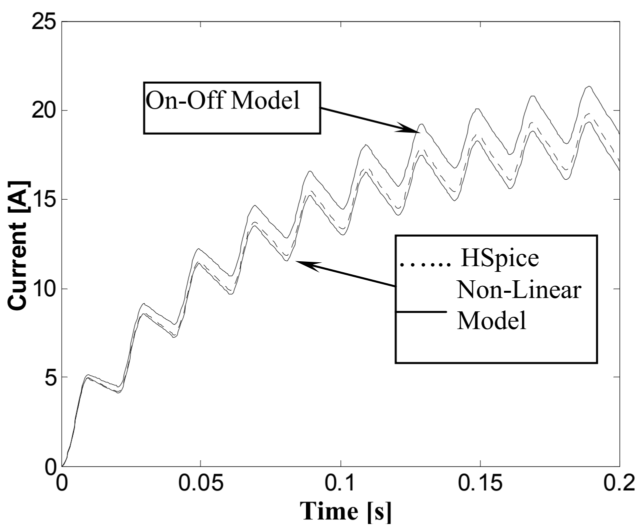


Figure 3: Results Comparison

CONCLUSIONS

One can note that the non-linear model simulation provided values in good agreement with the *HSpice* results. It suggests that the equation (1) represents fairly well the diode performance.

Considering the *HSpice* simulation as the reference for the comparisons, one can observe that the on-off model leaded to a non-satisfactory modeling for the diodes whereas the non-linear model provided a good approach.

Although the results are promising it is important to accomplish a more extensive simulations and further analysis considering different values for the voltage source, non-linear cores, etc.

REFERENCES

[1] P. Antognetti, G. Massobrio, Semiconductor Device Modeling with SPICE, New York, McGraw Hill, 1987.

[2]*Olympos*, Finite Element Package, ESAT/ELEN, 2001, http://www.esat.kuleuven.ac.be/elen/research/software-development/

[3] *Matlab* v. 6.0, Mathworks, 2000.

[4] ON Semiconductors, http://www.onsemi.com

[5] *Star-HSpice*, HP platform, 2000