8th International Conference on DEVELOPMENT AND APPLICATION SYSTEMS
Suceava, Romania, May 25 – 27, 2006

SPICE AND VHDL-AMS CO-SIMULATION FOR THE POWER SEMICONDUCTOR DEVICE

Alin MOSCALU[1], Mihaela MOSCALU[2], Petre-Daniel IONESCU[3]
RAJAC Iasi
University of Medicine and Pharmacy "Gr.T.Popa" Iasi
S.C. FORTUS S.A. Iasi
[1]alinm2004@yahoo.com

Abstract. *While the design of digital circuits is fully automated and is performed extremely fast, the analogical parts of these circuits are still manually designed. An essential role within the speed designing of digital circuits is the use of a HDL (hardware description language).*
This paper is discussing the use of VHDL models for power semiconductor devices.
In fact, within analogical circuits made with SPICE models for components, we will insert VHDL models for semiconductor devices.
The advantages of this inserting are the increasing of simulation speed as well as the possibility to use some complex models for semiconductor devices by modeling all phenomena that are influencing the device's functioning.
Keywords: *VHDL-AMS, SPICE simulation*

Introduction

Before physically manufacturing an electronic circuit, an important stage is the simulation of its behaviour in order to check its optimal functioning. Simulation processes require complex models for its components, models that can be achieved on basis of their behaviour [equivalent models] or on basis of state equations.

SPICE is one of the most used simulators for analogic circuits which uses equivalent models for semi-conductive devices. In order to build models on basis of the equations it is possible to use VHDL-AMS, one of the languages that support the analogic circuits' description on basis of ordinary differential algebric equations (ODAE) in order to complete the support needed to describe and simulate discrete [1].

VHDL-AMS and its capability to provide communication between discrete events and continuous time systems, allows the creation of very complex models for the functioning of analogic, digital and mixed signals electronic circuits.

Models presented in this paper, described in VHDL-AMS, are based on a SPICE model for semi-conductive devices, the validation of these models being performed on basis of SPICE simulations, with use of MULTISIM program.
MULTISIM, manufactured by Electronics Workbench, is a program conceived for the simulation of circuits that allow users to simulate circuit components, simultaneously with SPICE models and VHDL-AMS models.
First we will use MULTIVHDL program for writing and checking the VHDL code for the modeled device. After validating this code we will create a new MULTISIM component, by using a link of this code. Hence, any modification of the VHDL code will be reflected in the functioning of the model used in the MULTISIM simulation.

Models for the semi-conductive device

In order to create models susceptible to soundly describe the functioning of the semi-conductive device some rules are to be observed:
A necessary, but not sufficient condition, for the model's solvability, is to make sure that the total number of unknown quantities within the model

is equal to the total number of the characteristic equations within the model.
The total number of the characteristic equations within the model is determined by the total number of scalar situations within the model.
Initial conditions play a crucial role in the solving of a set of equations which represent a continuous-time system.
The device's model, in most of the cases, use some forms of depending sources [controlled sources] in order to represent its functioning. These controlled sources may lead to "divide by zero" situations, within the model, situations which are to be avoided.
Model's parameters are very important in order to characterize the model. Non-realistic parameters may lead to the non-solving of the model.

The VHDL model

The VHDL model is to be built for the power diode, starting from the device's equivalent model.

Depending on the operating frequency we will have the following equivalent models for the semi-conductive device: the static model, the low signal model and the high signal model.

By static model or direct current model it is understood that device's model which operates at low frequencies (<=1 kHz) or in direct current, being independent of signal's level.
By high signal model it is understood that semiconductor device's model which operates at high frequencies and can admit huge signal variations.

By low signal model it is understood a model resembling the high frequency model, but which can stand only slight signal variations around the direct current functioning point.

In SPICE this low signal model is represented by the linearization of high signal model [high frequency] around the direct current functioning point.

Therefore we will study the semi-conductive device's high frequency model and on its basis we will develop a VHDL model, able to operate at all frequencies and signal levels.

The semi-conductive device's high frequency model at low frequency or in direct current, a realistic approximation of a power diode is shown in Figure 1 that is a non-linear power source serially connected to a resistor.

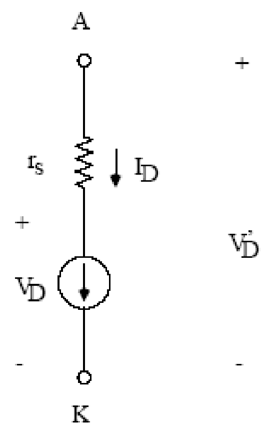

Figure 1. Diode equivalent model.

The value of a non-linear power source is given by equation:

I-V characteristic

$$i_D = \begin{cases} I_s\left(e^{\frac{v_D}{nV_T}} - 1\right) + v_D G_{min} & ; \quad -5V_T \le v_D \\ -I_s + v_D G_{min} & ; \quad -BV < v_D < -5V_T \\ -I_{BV} & ; \quad v_D = -BV \\ I_s\left(e^{-\frac{BV+v_D}{nV_T}} - 1 + \frac{BV}{V_T}\right) & ; \quad v_D \le BV \end{cases}$$

Model parameters:

n - **emission coefficient** (empirical, $1 \le n \le 2$), I_s - **saturation current**,
BV - **break down voltage**, I_{BV} - **break down current**,
G_{min} - **minimum conductance** (introduced to facilitate numerical calculations).

and iar V_D is the voltage applied to diode's terminals, The theoretical I-V characteristic, drafted on basis of the above shown equations looks like it follows:

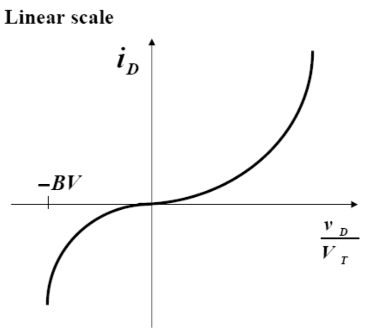

Figure 2. The theoretical I-V characteristic.

This paper tries to validate proposed models by drafting these characteristics using the SPICE and VHDL models.

In order to represent the semi-conductive device's parameters in VHDL we will use several IF instructions.

```
entity diode is
library IEEE;
use IEEE.std_logic_1164.all;
use IEEE.electrical_systems.all;

port(terminal din,dout: electrical);
end diode;
--
architecture complete of diode is
quantity Vd across di through din to
dout;
constant VT: real := 0.026;
constant BV: real := 100.0;
constant IBV: real := 0.001;
constant ISS: real := 1.0E-12;    --
saturation current
constant NISS: real := -1.0E-12;  --
negative saturation current
constant n: real := 1;
begin
  if (Vd >=(-5.0*VT*n)) use
      di == ISS*(exp(Vd/(VT*n)) -
            1.0))+(gmin*Vd);
elsif ((Vd >=(-5.0*VT)) and (Vd > -
       BV)) use
      di == ((-1.0*ISS)+(gmin*Vd));
  elsif (Vd = BV) use
      di == -IBV;
  else
      di == NISS*(exp(-(BV+Vd)/VT)-
            1.0+BV/VT);
  end use;
end complete;
```

Next to the use of VHDL model the next transfer characteristic (I_V) is obtained:

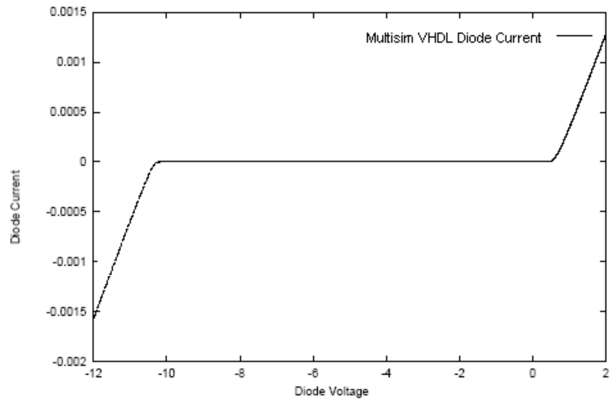

Figure 3. DC Characteristic of Diode.

Then, in order to validate proposed model, we will use a diagram for a mono-alternance rectifier in which we will implement successively the proposed SPICE and VHDL models.

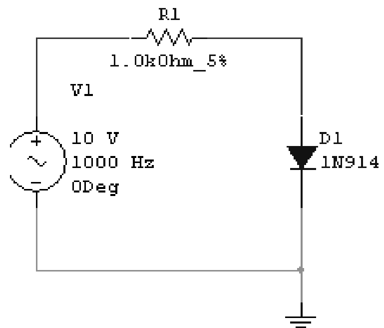

Figure 4. Half wave rectifier.

By means of an oscilloscope we will visualize the voltage on diode's terminals, which will have the shape below:

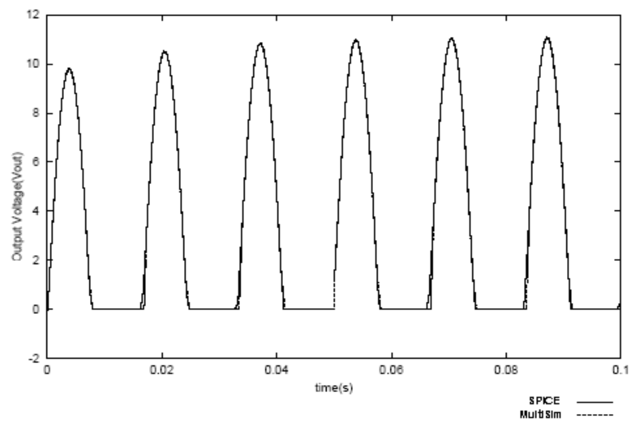

Figure 5. Transient characteristics of rectifier circuit.

These are the wave shapes obtained following the use of a simple model for the power diode. Starting from this simple model complex models can be built, models comprising all the factors that could influence the diode's functioning. Hence, we can take into account the variation of heat dissipated on the pn junction [3] obtaining this way accurate data concerning the functioning of this circuit at various temperatures.

Or it is possible to create the model for a reversed recovery diode.

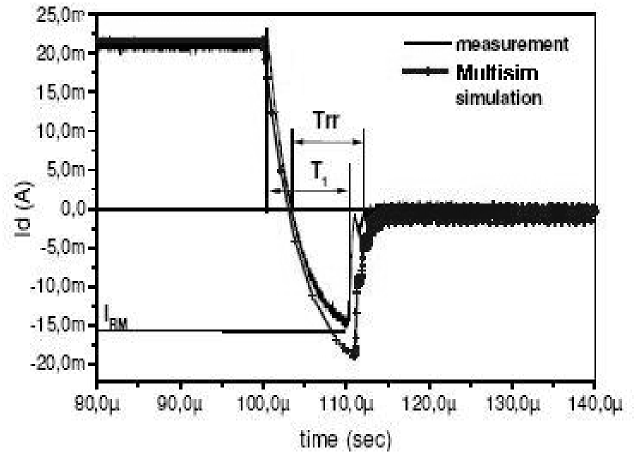

Figure 6.

It is noticed in this case also that the wave shapes obtained when using the VHDL model are identical to those obtained when using the SPICE model.

Conclusions

The simulation and the modeling of power semi-conductive devices is crucial for developing complex models for power circuits. The model's portability is also very important. As it can be noticed in this paper, compared to the SPICE model which requires extra voltage and current sources in order to model each phenomenon that can influence the functioning of the semi-conductive device, by using a VHDL model it is easy to obtain complex models only on basis of its state equations.

References

[1] P. Ashenden, G. Peterson, D. Teegarden (2003) *The system designer's guide to VHDL-AMS*, Morgan Kaufmann Publishers.

[2] Peter O. Lauritzen et Cliff L. Ma (1991) *A simple diode model with reverse recovery,* IEEE Transaction on power electronics, vol. 62, no.3.

[3] L.Starzack, M.Zubert, A.Napieralski, *The new approach to the power semiconductor device modeling.*

[4] Lyse-Aline Coyitangiye, Richard Grisel, *Model of the diode with VHDL-AMS including reverse recovery*

[5] Electronics Workbench (2000) *Multisim-Schematic Capture, Simulation & Programmable Logic - Getting Started & Tutorial*, Canada.

[6] Patricia Desgreys, Mohamed Karray, Jean-Jacques Charlot, Yannick Herve, *Modele thermo-opto-electrique dúne diode VCSEL en VHDL-AMS.*