Microwave Large-Signal PHEMT Model for SPICE-based simulators

J. M. ZAMANILLO, H. INGELMO, B. COBO, C. PEREZ-VEGA AND A. MEDIAVILLA

Communication Engineering Department (DICOM)
University of Cantabria, Laboratorios I+D Telecomunicación
Plaza de la Ciencia s/n, 39005 Santander (Cantabria)
SPAIN
Phone: 34-942-202219; Fax: 34-942-201488;

Abstract — A comprehensive large-signal HEMT model that provides a realistic description of measured characteristics over all operating regions for different PHEMTs is presented. The model was previously tested in harmonic-balance based simulators [1] and for the first time it has been implemented inside the time domain SPICE simulator. In order to do that, a new set of routines and libraries has been developed. The procedure introduced here can be extended to model other kind of devices described in terms of equivalent circuit. DC and scattering simulation results show very good agreement with the experimental measurements.

1. Introduction

The potential of microwave MESFETs and PHEMTs active devices in actual communications systems have stimulated the research efforts of our group with the development of different large-signal models for harmonic balance based simulators. These models offer results totally in concordance with the experimental measurements [1]-[4], DC and pulsed I-V curves, scattering parameters and CW laser optical stimulation of the device. Commercial simulators have the disadvantage of high cost, which affordable only by certain manufactures and universities, because of the discount policy by educational licenses. Bearing in mind this idea, and in a trial to popularize the models developed by our group, emerges the following question: Why do not to use SPICE as an microwave devices simulator?

This software is a well tested tool and universally known. In the 70’s of the past century, SPICE was developed by the University of Berkley, and nowadays several commercial simulators use the original code of SPICE for time domain simulations. This is one of the reasons that makes SPICE one of the most famous and popular electrical simulators used by the scientific community and electronic manufacturers. All the active devices models included in PSPICE, including the GaAs MESFET, are based on physical characteristics [5] and therefore the parameters that define them depend on the transistor production technology used, as well as geometric factors, usually, known only by the manufacturer or the designer of the device. Unfortunately, SPICE lacks of a built-in HEMT physical model. When SPICE has been used in the past to simulate GaAs MESFET and HEMT circuits, the standard JFET model was often employed in place of a true physical GaAs FET model. However, this approach has been shown [6]-[8] to yield considerable error in the various computed circuit responses. In this paper, we describe the implementation of our AlGaAs model of PHEMTs, previously reported [1]-[2] into PSPICE (commercial version for PC of the original Berkeley SPICE 2G code). This task has been divided into two parts: the first one, which is the aim of the present paper, will show how the microwave PHEMT model has been introduced in the SPICE simulator and the results obtained.

The simulated characteristic I/V curves and scattering parameters has been validated with experimental measurements carried out in our laboratory. In a second stage, which is now under development, and whose results will be shown in future communications, intermodulation effects including high order derivatives of the non-linear Ids current source must be simulated with the model.

Fig. 1  DC Large Signal Model for PHEMT devices.
2. The Microwave Model

Among the different existing classical models in the bibliography, our large-signal model [1]-[2] was chosen because it is adequate to represent with fidelity the behavior of these devices, and furthermore the model is valid for HEMT and MESFET microwave transistors.

The DC large-signal equivalent circuit model proposed is shown in Fig. 1. Classical models offer, in general, good results for low-power transistors; however they are not capable of adequate way for power HEMT devices. The model implemented here, is adequate for this kind of devices and is also capable to simulate the trasconductance compression phenomena in HEMT devices.

The expression for the current source $I_{ds}$ is given by (1) and it uses the internal voltages of the transistor $V_{gi}$ and $V_{di}$ as variables, these voltages are shown in Fig.1. The physical mean of the internal parameters used by this expression are shown in Table 1. The relation between external and internal control voltages is shown in expressions (2) and (3).

The rest of the equations that compose the complete model have been omitted, since they have been previously reported and a detailed description of them can be found in [1]-[2]. It is very important to emphasize the accuracy in the extraction procedure of the parasitic resistance values because of their implication in the value of the intrinsic voltages. It is very important to emphasize the accuracy in the extraction procedure of the parasitic resistance values because of their implication in the value of the intrinsic voltages. The model is valid for MESFET and HEMT microwave transistors. It be noted that being a DC model have been omitted parasitic inductances $L_g$, $L_d$, $L_s$, as well as the parasitic capacitances $C_{pgi}$ and $C_{pdi}$.

\[
I_{ds} = I_{dss} \cdot \left(1 - \frac{V_{gi}}{V_t}\right)^{(E+K_e V_g)} \cdot \left(1 + \frac{S_e V_{di}}{I_{ds}}\right) \cdot \tanh \left(\frac{S \cdot V_{di}}{1 - K_g \cdot V_{gi}}\right) 
\cdot \left(\frac{V_{di} - V_{gs}}{\mu}\right)
\]

Where

\[
V_t = V_{to} + \gamma_{Ids} \cdot I_{di}
\]

\[
V_{gi} = V_{gs} - I_d \cdot R_s
\]

\[
V_{di} = V_{ds} - I_d \cdot (R_d + R_s)
\]

3. PSPICE implementation

To guarantee the compatibility with all versions of the SPICE simulator, schematics had not been used in the description of the model components, and the old style commands from SPICE 2G written in human-readable ASCII text files has been used to integrate the model within the SPICE simulator. The same procedure has been used to implement all the libraries needed for a correct microwave simulation. This task has been structured in three different tasks which will be commented as follows:

- Task 1: Develop a Mathematical Library.
- Task 2: Definition of necessary subcircuits.
- Task 3: Built a main program that calls the above mentioned library and subcircuits.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Physical Mean</th>
<th>S.I. units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{dss}$</td>
<td>Saturation drain current for $V_{gs}=0$ V.</td>
<td>A</td>
</tr>
<tr>
<td>$V_{to}$</td>
<td>Pinch-off Voltage</td>
<td>V</td>
</tr>
<tr>
<td>$E$</td>
<td>Parameter of the saturation current variation.</td>
<td>-</td>
</tr>
<tr>
<td>$K_e$</td>
<td>Parameter of the saturation current variation.</td>
<td>V^-1</td>
</tr>
<tr>
<td>$S_e$</td>
<td>Saturation drain current slope</td>
<td>Ω^-1</td>
</tr>
<tr>
<td>$S_t$</td>
<td>Drain current slope in the lineal zone for $V_{gi}=0$ V.</td>
<td>Ω</td>
</tr>
<tr>
<td>$K_g$</td>
<td>Parameter that describes de lineal zone</td>
<td>V^-1</td>
</tr>
<tr>
<td>$V_{pf}$</td>
<td>Voltage at which transconductance degradation begins</td>
<td>V</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Transconductance adjustment parameter</td>
<td>-</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Transconductance adjustment parameter</td>
<td>V^-1</td>
</tr>
</tbody>
</table>

**TABLE 1**

IDS CURRENT PARAMETERS
Fig. 2 Description of the mathematical functions contained in the library MATH.LIB (the subcircuits have been omitted here because its extension).

The first step necessary is to develop a standard mathematical library called MATH.LIB which contains a series of sub-circuits that implement typical mathematical functions used to define the model equations (addition, subtraction, potentiation, hyperbolic tangent, exponentiation, etc.).

The second task consists of the definition of the necessary sub-circuits that compose the complete model, one for each non linear current source (Ids, Igs and Igd), as well as the necessary components to properly model the non-linear capacitance Cgs, and the linear capacitance Cgd.

To illustrate this, Table 2 shows how the non-linear current source has been implemented in PSPICE using equation (1). The source code that describes the behavior of Ids has been named as: HEMT_IDS.LIB. It has six nodes, four input nodes and two output nodes, as is shown in Table 2.

### TABLE 2

**EXPLANATION OF IDS CURRENT INTERNAL NODES USED IN SUB-CIRCUIT HEMT_IDS**

The necessary task consists of the definition of the necessary sub-circuits that compose the complete model, one for each non linear current source (Ids, Igs and Igd), as well as the necessary components to properly model the non-linear capacitance Cgs, and the linear capacitance Cgd.

To illustrate this, Table 2 shows how the non-linear current source has been implemented in PSPICE using equation (1). The source code that describes the behavior of Ids has been named as: HEMT_IDS.LIB. It has six nodes, four input nodes and two output nodes, as is shown in Table 2.
The library *HEMT_GSD.LIB* is very similar to *HEMT_IDS.LIB*, and contains the sub-circuits related to the sources $I_{gs}$ and $I_{gd}$, as well as the capacities $C_{gs}$ and $C_{gd}$.

Finally, a main program that calls the libraries and sub-circuits, written in SPICE language is needed to run the simulation. The content of this file it is shown in figure 4. This procedure can be generalized to simulate any type of electronic device that can be characterized by an electrical equivalent circuit.

![Fig 5 - Simulation results of OMMIC PHEMTs in PSPICE.](image)

(a) Simulation of 4x30µm PHEMT transistor within PSPICE. $I_{ds}$ vs. $V_{ds}$.

(b) Simulation of 4x30µm PHEMT transistor within PSPICE. $I_{ds}$ vs. $V_{gs}$.

With the necessary equations for the correct operation of the circuit proposed selected has proceeded to the integration of the same in the simulator PSPICE student’s version of ORCAD.

### 4. Results

In order to test the behavior of the model implemented in PSPICE simulator, two different size PHEMT devices from OMMIC foundry, have been used. The first transistor is a low power device, 4*30µm (4 fingers, and 30µm of gate periphery) and the other is a medium power transistor 6*150µm (6 fingers, and 150µm of gate periphery). Fig. 2 (a) and Fig. 2 (b) show the PSPICE output by means of the graphic post-processor PROBE for the 4*30µm device. Because SPICE is unable to import and plot external measurement data files, a small program written in MATLAB language shows a comparison between the SPICE results read directly of the output file .OUT and the experimental DC measures performed in our laboratory for the 6*150µm, transistor, taking the $V_{gs}$ and $V_{ds}$ sweeps in the same experimentally measured values.

![Fig 6 - Simulation results of OMMIC PHEMTs in PSPICE.](image)

(a) Comparison between simulated and experimental I-V characteristic curves measurements for a 4*30µm device (4 fingers, and 30µm of gate perifery) PHEMT transistor.

(b) Comparison between simulated and experimental I-V characteristic curves measurements for a 6*150µm device (6 fingers, and 150µm of gate perifery) PHEMT transistor.

### 5. Conclusion

An electrical large-signal model for PHEMT and MESFET devices has been implemented in the PSPICE simulator, extending its utilization to this type of microwave devices. The method presented here can be extended to other types of non-linear devices using the same procedure. Simulation
results show good experimental agreement, being similar to those harmonic balance type simulators, including the transconductance compression phenomena. So it maybe concluded that using a free tool as is the student version PSPICE accurate results for microwave devices can be obtained.

As future work lines, we continue with the development of new routines that allow to get the high order derivatives for this type of devices, and the introduction of the electro-optical MESFET/HEMT transistors model [9,11] in the same way the model here presented has been introduced.

6. Acknowledgement

This work has been supported by the Spanish Ministry of Science and Innovation thru the project TEC2005-07985-C03-01.

References:


