# Application of Statistical Simulation Using SPICE to Parameter Extraction of RF Transistor Models

Elissaveta D. Gadjeva[1], Vladislav P. Durev[2]

***Abstract*** **–In the present paper, the possibilities of the PSpice simulator for statistical analysis are used to parameter extraction of RF transistor models. The measured S-parameters are introduced using frequency-dependent tables of EFREQ type. The error function is calculated using macro-definitions in the graphical analyzer Probe. The model parameters, which ensure a minimal error function, are also obtained in Probe. The implementation of the extraction procedure in the standard circuit simulator such as PSpice allows to use the extended possibilities of the input language for the RF model description, the powerful statistical simulation tool, as well as the goal function formulation and assessment in the graphical analyzer Probe.**

***Keywords*** **– High frequency transistor models, parameter extraction, S-parameters, PSpice simulation**

## I. Introduction

The extended possibilities of general-purpose circuit simulators such as *PSpice* with respect to the input language, as well as the existence of powerful simulation tools, allow the application of such standard simulators to adequate modeling and simulation of electronic circuits. Behavioral SPICE models of high accuracy are developed, based on measured two-port parameters [1,2,3], as well as on datasheet parameters [4,5].

The high frequency transistor models are of great importance in computer-aided design of high frequency circuits. They allow the application of general-purpose circuit analysis programs for the RF circuit simulation in the frequency and in the time domains. A number of approaches are developed for the model parameter extraction using the measured two-port *S*-parameters, based on direct procedures [2,3,6,7], as well as on optimization parameter extraction procedures [8,9].

In the present paper, the possibilities of the *PSpice* simulator for statistical analysis are used to parameter extraction of RF transistor models. The measured *S*-parameters are introduced using frequency-dependent tables of **EFREQ** type. The error function is calculated using macro-definitions in the graphical analyzer *Probe*. The model parameters, which ensure a minimal error function, are also obtained in *Probe*. The implementation of the extraction procedure in the standard circuit simulators such as *PSpice* allows to use the extended possibilities of the input language for the RF model description, the powerful statistical simulation tool, as well as the goal function formulation and assessment in the graphical analyzer.

In this paper, a parameter extraction methodology is proposed for a simplified small-signal transistor equivalent circuit. The extraction procedure is realized using the *OrCAD PSpice* circuit simulator. The extraction procedure is realized based on the two-port *S*-parameters.

## II. Extraction Procedure Using Monte Carlo Simulation

The extraction procedure is realized using statistical analysis used as an optimization tool. The goal function is the difference between the measured *S*-parameters and the modeled *S*-parameters of the RF transistor equivalent circuit. The simplified small-signal equivalent circuit shown in Fig. 1 is used, which is characterized by a good accuracy [8].

The manufacturer's *S*-parameters are measured at the corresponding bias point $V_{CE}$ and $I_C$. The initial values for the parameters $G_m$ , $R_{pi}$ and $C_{pi}$ are calculated using the transistor parameters $\beta$, $f_T$, and the current $I_C$ using the equations [8]:

$$G_m = \frac{I_C}{V_T} \; ; \; R_{pi} = \frac{\beta}{G_m} \; ; \; C_{pi} = \frac{G_m}{2\pi f_T} \qquad (1)$$

The phasors of the measured *S*-parameters are introduced in the *PSpice* model using frequency dependent sources of **EFREQ** type (Fig. 2).

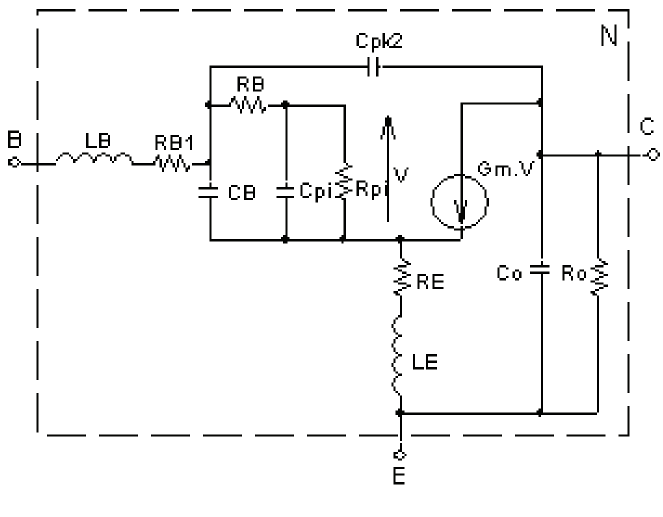



Fig. 1. Simplified small-signal RF transistor model

[1]Elissaveta D. Gadjeva is with the Faculty of Electronic Engineering and Technologies, Technical University of Sofia, Klimenrt Ohridski Blvd. 8, 1000 Sofia, Bulgaria, E-mail: egadjeva@tu-sofia.bg

[2]Vladislav P. Durev is with the Faculty of Electronic Engineering and Technologies, Technical University of Sofia, Klimenrt Ohridski Blvd. 8, 1000 Sofia, Bulgaria, E-mail: v_p_durev@yahoo.com

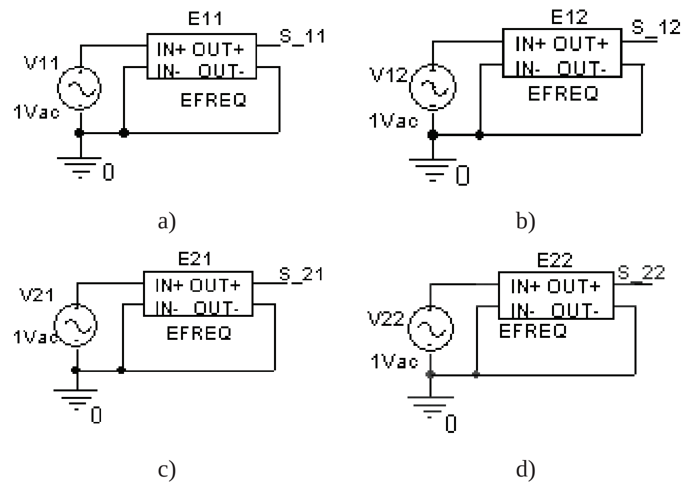



Fig. 2. Introducing the measured S-parameters in Capture

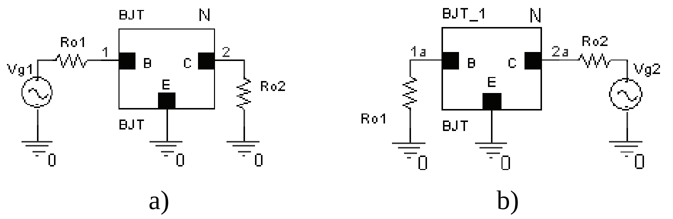



Fig. 3. *S*-parameter determination of the RF transistor model

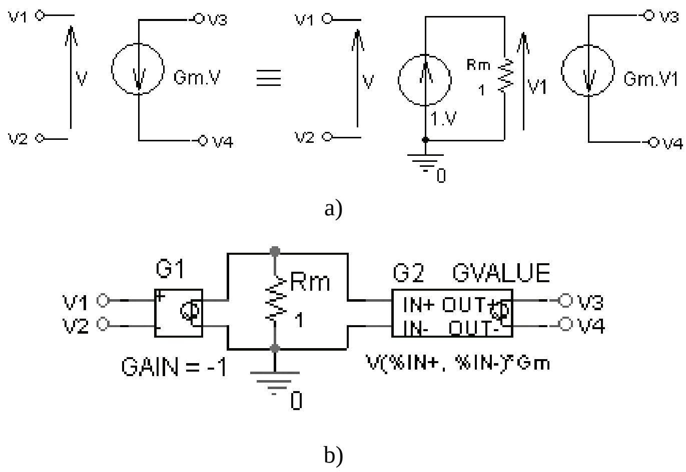



Fig. 4. Equivalent circuit for tolerance definition of the $G_m$ parameter

The frequency dependences of the magnitude $\left|\dot{S}_{ij}\right|$ and the phase $\arg(S_{ij}), i, j=1,2$ are defined in the form of frequency-dependent tables:

(frequency$_1$, magnitude$_1$, phase$_1$)
+ …
+ (frequency$_n$, magnitude$_n$, phase$_n$) ,

where *n* is the number of frequency points.

For example, the measured data for $S_{21}$ of the transistor BFY90 in the frequency range 100MHz$\div$500MHz at $V_{CE}$ = 10V and $I_C$ = 8mA, given in [8], are introduced in the model using the **EFREQ** element (Fig. 2c). The property **TABLE** has the form:

(100Meg,10.65,127) (200Meg,7.01,105) (300Meg,4.44,97) (400Meg,3.62,92) (500Meg,3.02,88)

and the attribute MAGUNITS=MAG is selected.

The *S*-parameters of the model are obtained in the form of node voltages of the circuit shown in Fig. 3, where $V_{g1}=V_{g2}$=1V [10]:

$$S_{11}=V(1)\ ;\ S_{21}=V(2)\ ;\ S_{12}=V(1a)\ ;\ S_{22}=V(2a) \qquad (2)$$

The blocks BJT contain the small-signal equivalent circuit **N** shown in Fig. 1.

The determination of the model parameters, which ensure the best fit of the simulated *S*-parameters to the measured data, is reduced to statistical optimization using *Monte Carlo* analysis of the *OrCAD PSpice* simulator.

The parameters of the model elements are defined by tolerances and *Monte Carlo* simulation is performed. The difference between the measured and the simulated *S*-parameters, is calculated in the graphical analyzer *Probe* and the error function *ERR* is obtained in the form:

$$ERR=\sum_{i,j} w_{ij}\left|\left(S_{ij}^{(s)}-S_{ij}^{(m)}\right)\right| \ ,\ i,j=1,2 \qquad (3)$$

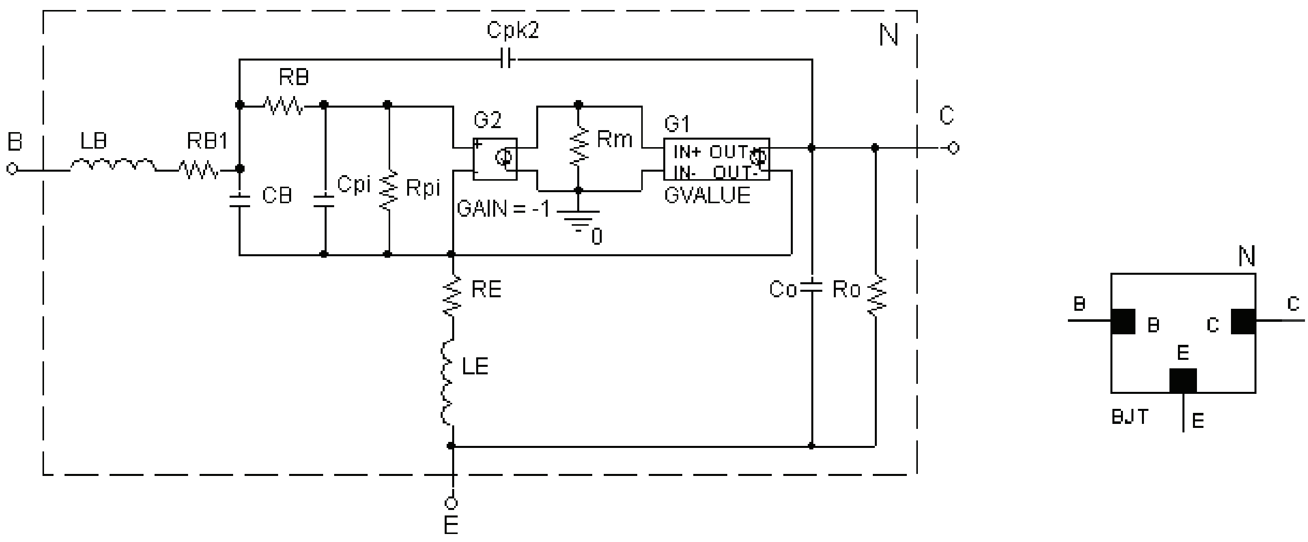
$$S_{ij}^{(m)}=\left|\dot{S}_{ij}^{(m)}\right|;\ S_{ij}^{(s)}=\left|\dot{S}_{ij}^{(s)}\right|,$$

where $S_{ij}^{(m)}$ and $S_{ij}^{(s)}$ are the magnitudes of the corresponding measured and simulated *S*-parameters.

The variant, which is characterized by a minimal *RMS* value of the error function *ERR*, calculated for the investigated frequency range, ensures the best fit of the model *S*-parameters to the measured data:

$$EPS=\min(RMS(ERR)). \qquad (4)$$

It is obtained using post-processing of the simulation results in the graphical analyzer *Probe*. In order to obtain all parameters $S_{ij}^{(m)}$, *i*,*j*=1,2 in *Probe*, the circuits in Fig. 3a and Fig. 3b are analyzed simultaneously. The corresponding parameter values of the model elements, participating in the blocks, are to be the same during *Monte Carlo* simulation. This is achieved by defining correlated (**LOT**) deviations for the parameter values of the model. The *PSpice* simulator allows to define tolerance deviations for the passive (R,L,C) elements, but not for the controlling parameters of the dependent sources. In order to define tolerance deviation for the parameter $G_m$ of the model in Fig. 1, the voltage controlled current source (**VCVS**) is represented equivalently by the circuit shown in Fig. 4a. The tolerance of model parameter $G_m$ is applied to the resistor $R_m$. The corresponding *PSpice* model of **VCCS** is shown in Fig. 4b. The *PSpice* realization of the small-signal equivalent transistor circuit is presented in Fig. 5a. It is defined in the form of block (Fig. 5b).




a) b)

Fig. 5. *PSpice* realization of the small-signal RF transistor model

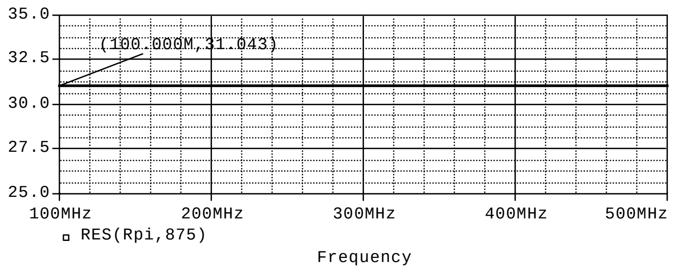



Fig. 6. Determination of $R_{pi}$ value in *Probe*

## III. Determination of Error Function in the Graphical Analyzer Probe

The minimal value *EPS* of the error function (4) is obtained using post-processing of the simulation results in the graphical analyzer Probe.

The following macro-definitions are used fir this purpose:

*Determination of S-parameters
S11 = 2*V(1)-1
S22 = 2*V(2a)
S21 = 2*V(2)
S12 = 2*V(1a)

*Determination of the error function
E11 = RMS(M(S11)-M(V(S_11)))
E21 = RMS(M(S21)-M(V(S_21)))
E12 = RMS(M(M(S12)-V(S_12)))
E22 = RMS(M(S22)-M(V(S_22)))
EPS=E11+E12+E21+E22

After selecting the best variant *N*, the extracted optimal parameter values can be obtained in *Probe* using predefined macros [10]. For the resistor element it is in the form:

RES(RR,N)=M(V(RR:1,RR:2)@N/I(RR)@N)

A *Monte Carlo* simulation is performed with 2000 runs. The best fit corresponds to *N*=876. The optimal value for $R_{pi}$ is calculated using the macro-definition: RES(Rpi,875)

The obtained value for $R_{pi}$ in the graphical analyzer *Probe* is presented in Fig. 6. The predefined macro for the capacitor element is in the form

CAP(CC, N)=I(CC)@N/(V(CC:1,CC:2)@N*2*pi*frequency))

The macro-definition for the controlling parameter of **VCCS** $G_m$ has the form:

GM(GG,N)=M(I(GG)@N/V(v1,v2)@N)

where GG is the VCCS name and V(v1,v2) is the controlling voltage.

As a result, the extracted parameter values of the model are: $R_{pi}$= 31.04Ω , $C_{pi}$=6.42pF, $G_m$=1mho, $R_B$ =10Ω, $R_{B1}$=20Ω, $C_B$=2pF, $R_E$=1.78Ω, $C_{pk2}$=2.2pF, $L_E$=1nH, $L_B$=0.04nH, $C_o$=0.06pF, $R_o$=9.3Ω.

The simulation results for $S_{21}$ of the model shown in Fig. 1 with initial parameter values of the circuit elements are shown in Fig. 7 together with the measured data. The simulation results for $S_{21}$ with extracted parameter values of the model after optimization are shown in Fig. 8. It is seen that a good approximation is obtained applying the extraction procedure using statistical *PSpice* optimization.

## IV. Conclusion

An extraction procedure for RF transistor model parameter determination has been proposed in the paper. The parameter extraction is reduced to statistical analysis using standard circuit simulators such as *OrCAD PSpice*. The error function and the extracted model parameter values are obtained in the graphical analyzer *Probe*.

## V. Acknowledgements

This research is in the framework of the BY-TH-115/2005 project.

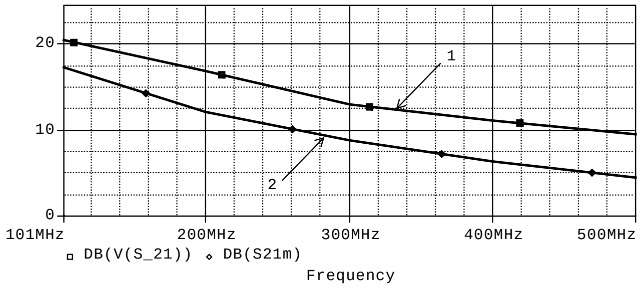



Fig. 7. Simulation results for $S_{21}$ with initial parameter values:
1- measured 2 - simulated

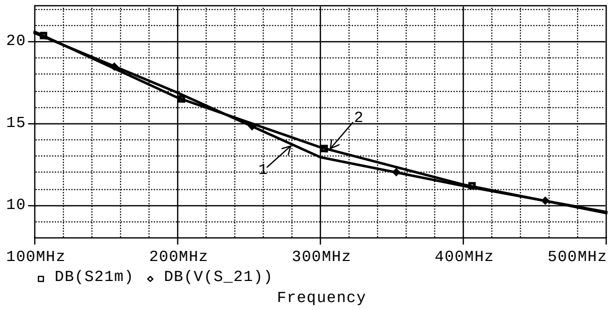



Fig. 8. Simulation results for $S_{21}$ with extracted parameter values after optimization
1- measured 2 - simulated

## REFERENCES


[1] M. Hristov, E. Gadjeva, D. Pukneva, Computer Modelling and Geometry Optimization of Spiral Inductors for RF Applications Using Spice, *The 10th International Conference Mixed Design of Integrated Circuits and Systems*, MIXDES'2003, 26-28 June 2003, Lodz, Poland.

[2] G. Angelov, M. Hristov, O. Antonova, E. Gadjeva, "Parameter Extraction for Simplified RF NMOSFET Equivalent Circuit using SPICE", 14th International Conference Mixed Design of Integrated Circuits and Systems, Ciechocinek, 21-23 June 2007, ISBN: 83-922632-9-4, pp. 464-468.

[3] E. Gadjeva, V. Durev, M. Hristov, "Parameter Extraction of Planar Inductor and Transformer Models Using SPICE", *14th International Symposium on Power Electronics* - EE 2007, Novi Sad, Republic of Serbia, November 7 - 9, 2007.

[4] B. Nikolova, G. Nikolov, M. Hristov, "Simulation of Integrated Sensors Using Analog Behavioural Models", *International Scientific Conference Computer Science'2006*, Istanbul, 2006, pp 279-284

[5] B. Nikolova, G. Nikolov, M. Hristov, "Analogue Behavioural Modelling of Integrated Sensors", *The 2006 International Conference on Computer Engineering & Systems* (ICCES'06), Cairo, Egipt, November 5-7, 2006

[6] R. Rudolph, Doerner and P. Heymann. "Direct Extraction of HBT Equivalent-Circuit Elements", *IEEE Transactions of Microwave Theory and Techniques*, vol. 47, No. 1, Jan. 1999, pp. 82-83.

[7] E. Gadjeva, E. Gjerassi, B. Mihova, "Computer-Aided Extraction of HBT Small-Signal Parameter Values Using General-Purpose Analysis Programs", *International Conference on Fundamentals of Electrotechnics and Circuit Theory* – IC-SPETO'2002, 22-25 May, 2002, Gliwice, Poland, pp.413-416.

[8] M. A. Firas,"Microwave Transistor Modeling for Time Domain Simulation", *Summit Technical Media*, LLC, Sept. 2007, pp.52-60.

[9] E. Gadjeva, V. Durev, M. Hristov, D. Pukneva, "Optimization Of Geometric Parameters Of Spiral Inductors Using Genetic Algorithms", *Proceedings of the International Conference "Mixed Design of Integrated Circuits and Systems" MIXDES 2006*, 22-24 June, 2006, pp. 518- 521

[10] E. Gadjeva, S. Farchy, T. Kouyoumdjiev , "Computer modeling and simulation of electronic and electrical circuits using OrCAD PSpice" Meridian 22, Sofia, 2005.

[11] PSpice User's Guide, Cadence Design Systems, 2003.