14th International Conference
MIXED DESIGN
MIXDES 2007
Ciechocinek, POLAND
21 – 23 June 2007

PARAMETER EXTRACTION FOR SIMPLIFIED RF NMOSFET EQUIVALENT CIRCUIT USING SPICE

G.V. ANGELOV, M.H. HRISTOV, O.J. ANTONOVA, E.D. GADJEVA
TECHNICAL UNIVERSITY OF SOFIA, BULGARIA

KEYWORDS: Cadence PSpice, Simplified NMOSFET model, Circuit simulation, Parameter extraction

ABSTRACT: In the paper, a parameter extraction approach is proposed to a simplified small-signal NMOSFET equivalent circuit utilizing Cadence PSpice circuit simulator. A direct extraction procedure is realized based on the two-port Y-parameters. The computer realization of the developed extraction procedure is performed using parameterization of the model. Based on postprocessing in the graphical analyzer Probe utilizing corresponding macrodefinitions, the two-port Y-parameters are calculated. Verification of the extraction methodology is made by comparing the simulated and experimental results for the Y-parameters.

INTRODUCTION

A variety of compact models have been of great interest and studied for deep submicron metal-oxide-semiconductor field effect transistor (MOSFET) simulation. The model parameters extraction intrinsically characterizes properties of designed and fabricated devices. One of the challenges for the MOSFET-based RF circuit design is the prediction of the RF transistor performance characteristics. Therefore robust characterization methodology for MOSFET is crucial for RF circuit simulations. Different extraction approaches have been proposed in [1-3]. The extraction procedures encounter serious problems in deep submicron dimensions, such as time-consuming, ineffective extraction process, poor accuracy, and lack the predictive capability in applications.

In this paper, a parameter extraction methodology is proposed for a simplified small-signal NMOSFET equivalent circuit. The extraction procedure is realized using the *Cadence PSpice* circuit simulator. Parameter extraction is performed from experimental results for a 0.35 μm CMOS technology. A direct extraction procedure is realized based on the two-port Y-parameters.

PARAMETER EXTRACTION FROM Y-PARAMETER MOSFET EQUIVALENT CIRCUIT

Y-parameter definition for the equivalent circuit

Transistor impedance matching is critical in RF applications. Basic condition for a good prediction of input and output transistor characteristics is the proper definition of the intrinsic charges. These charge relations may be described in terms of capacitances that depend on bias conditions [1, 2]. At high frequencies these intrinsic and extrinsic capacitances of the device typically dominate the responses and, hence, must be accurate.

In order to verify the intrinsic capacitances, a simplified NMOSFET equivalent circuit is used. The circuit is based on an assumption of steady-state (known as the quasistatic assumption) and is shown in Fig. 1 [1]. The circuit includes the gate resistance but neglects the drain and source resistances. Further, it is assumed that the bulk and source are effectively tied together so that the bulk transconductance and the source-bulk capacitance may be omitted.

It is useful to use Y-parameter definitions in order to achieve expressions in terms of the intrinsic capacitances. The two-port Y-parameters are derived from the equivalent circuit shown in Fig. 1.

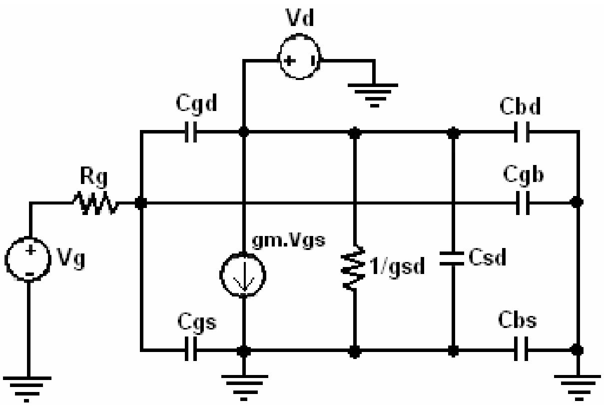

Fig. 1. Equivalent quasi-static circuit for MOSFET configured for Y-parameter extraction.

$$y_{11} = \left.\frac{I_g}{V_g}\right|_{V_d=0} = \frac{j\omega\left(C_{gs}+C_{gb}+C_{gd}\right)}{1+j\omega\left(C_{gs}+C_{gb}+C_{gd}\right)R_g} \qquad (1)$$

$$y_{12} = \left.\frac{I_g}{V_d}\right|_{V_g=0} = \frac{-j\omega\, C_{gd}}{1+j\omega\left(C_{gs}+C_{gb}+C_{gd}\right)R_g} \qquad (2)$$

$$y_{21} = \left.\frac{I_d}{V_g}\right|_{V_d=0} = \frac{y_m - j\omega\, C_{gd}}{1+j\omega\left(C_{gs}+C_{gb}+C_{gd}\right)R_g} \qquad (3)$$

where $y_m = g_m(1 - j\omega\tau)$

$$y_{22} = \left.\frac{I_d}{V_d}\right|_{V_g=0} = j\omega\left(C_{sd} + C_{db}\right) + g_{sd} + \\ + \frac{j\omega C_{gd}\left(1 + g_m R_g + j\omega\left(C_{gs} + C_{gb}\right)R_g\right)}{1 + j\omega\left(C_{gs} + C_{gb} + C_{gd}\right)R_g} \quad (4)$$

Assumptions and parameter extraction

In order to extract the model parameters in Fig. 1 it is necessary to consider two boundary frequencies: low frequency f_l, and high frequency f_h. At low frequency f_l the following assumption is valid:

$$1 >> j\omega_l\left(C_{gs} + C_{gb} + C_{gd}\right)R_g \quad (5)$$

For the transistor considered here, F300p35 [3], the low frequency approximation holds up to approximately $1 \div 2$GHz – we take f_l = 1GHz; f_h = 12GHz. Taking into account the inequality (5), a direct extraction procedure can be developed consisting of the following steps:

1) Extraction of the parameter R_g from y_{11} at f_l and f_h:
Let $C_g = C_{gs} + C_{gb} + C_{gd}$. At low frequencies

$$y_{11l} \approx \frac{j\omega_l C_g}{1} \Rightarrow C_g = \frac{|y_{11}|_l}{\omega_l}$$

At high frequencies

$$y_{11h} = \frac{j\omega_h C_g}{1 + j\omega_h C_g R_g} \Rightarrow |y_{11}|_h = \frac{\omega_h C_g}{\sqrt{1 + \left(\omega_h C_g R_g\right)^2}}$$

$$R_g = \sqrt{\frac{1}{|y_{11}|_h^2} - \frac{\omega_l^2}{\omega_h^2 |y_{11}|_l^2}} \quad (6)$$

2) Extraction of the parameter C_{gd} from y_{12} at f_h :

$$y_{12} \approx \frac{-j\omega_l C_{gd}}{1} = -j\omega C_{gd} \Rightarrow C_{gd} = \frac{|y_{12}|_l}{\omega_l} \quad (7)$$

$$y_{12h} = \frac{-j\omega_h C_{gd}}{1 + j\omega_h C_g R_g}$$

$$C_{gd} = \frac{|y_{12}|_h \sqrt{1 + \left(\omega_h C_g R_g\right)^2}}{\omega_h} \quad (8)$$

3) Extraction of the parameter $C'_{gs} = C_{gs} + C_{gb}$ from y_{11} and y_{12} at f_l:
Let $C'_{gs} = C_{gs} + C_{gb}$ since C_{gs} and C_{gb} are connected in parallel. At low frequencies

$$y_{11l} \approx \frac{j\omega_l\left(C'_{gs} + C_{gd}\right)}{1} \Rightarrow C'_{gs} = \frac{|y_{11}|_l}{\omega_l} - C_{gd}$$

Hence

$$C'_{gs} = \frac{|y_{11}|_l - |y_{12}|_l}{\omega_l} \quad (9)$$

4) Extraction of the parameters g_m and τ from y_{21} and y_{12} at f_l and f_h:

$$y_{21l} \approx \frac{g_m(1 - j\omega_l\tau) - j\omega_l C_{gd}}{1} \approx g_m \Rightarrow g_m = |y_{21}|_l \quad (10)$$

At high frequencies

$$|y_{21}|_h = \frac{|g_m(1 - j\omega_h\tau) - j\omega_h C_{gd}|}{1 + j\omega_h C_g R_g}$$

Hence

$$\tau = \frac{\sqrt{|y_{21}|_h^2\left[1 + (\omega_h C_g R_g)^2\right] - g_m^2}}{g_m \omega_h} - \frac{C_{gd}}{g_m} \quad (11)$$

5) Extraction of the parameter C'_{sd} from y_{12} and y_{22} at f_l and f_h:
Let $C'_{sd} = C_{sd} + C_{db}$ since C_{sd} and C_{db} are connected in parallel. At high frequencies [3]

$$|y_{22}|_h \approx \omega_h\left(C'_{sd} + C_{gd}\right)$$

Consequently

$$C'_{sd} = \frac{|y_{22}|_h}{\omega_h} - C_{gd} = \frac{|y_{22}|_h}{\omega_h} - \frac{|y_{12}|_l}{\omega_l}$$

$$C'_{sd} = \frac{|y_{22}|_h}{\omega_h} - \frac{|y_{12}|_l}{\omega_l} \quad (12)$$

6) Extraction of the parameter g_{sd} from y_{12} and y_{22} , y_{21} and y_{22} at f_l:

$$y_{22l} \approx g_{sd} + \frac{j\omega_l C_{gd}\left(1 + g_m R_g + j\omega_l\left(C_{gs} + C_{gb}\right)R_g\right)}{1} \approx$$

$$\approx g_{sd} - \omega_l^2 C_{gd} C'_{gs} R_g \ ; \ |y_{22}|_l = g_{sd} - \omega_l^2 C_{gd} C'_{gs} R_g$$

$$g_{sd} = |y_{22}|_l + \omega_l^2 C_{gd} C'_{gs} R_g \quad (13)$$

Taking into account $C'_{gs} = C_{gs} + C_{gb}$ and $C'_{sd} = C_{sd} + C_{db}$, the modification of the equivalent circuit in Fig. 1 is constructed as shown in Fig. 2.

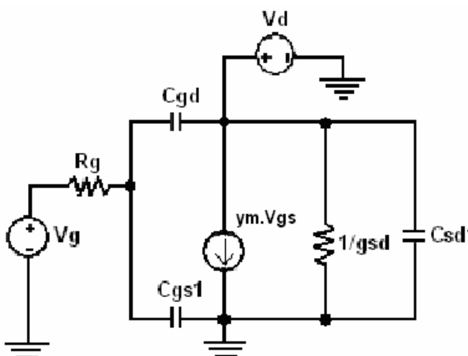

Fig. 2 Modified equivalent circuit for MOSFET parameter extraction.

The experimental data given in [3] for the required Y-parameter values are used for the extraction procedure assuming the low frequency f_l = 1GHz, and high frequency f_h = 12GHz (Table 1 and Table 2). The corresponding parameters obtained from the simulation results of the BSIM3v3 model [3] are presented in Table 3 and Table 4.

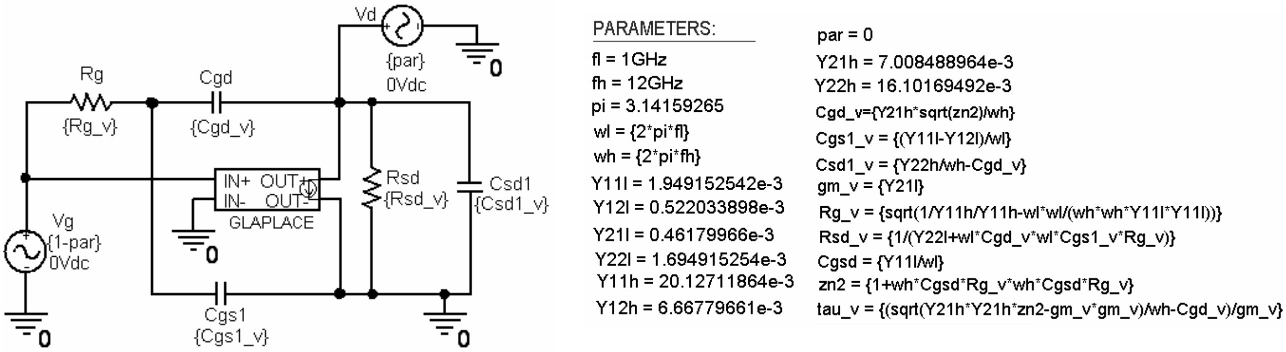

Fig 3. Parameter extraction of the simplified NMOSFET equivalent circuit using Cadence PSpice simulator.

1) F300p35 — Experimental

TABLE 1. Experimental data from [3] for f_l = 1GHz and f_h = 12GHz

Vgs = 0 V, Vds = 0 V

Y11l [mS]	1.949152542
Y11h [mS]	20.12711864
Y12l [mS]	0.46179966
Y12h [mS]	7.008488964
Y21l [mS]	0.522033898
Y21h [mS]	6.66779661
Y22l [mS]	1.694915254
Y22h [mS]	16.10169492
Wl [deg/s]	6.283185307 x 10^9
Wh [deg/s]	75.398223686 x 10^9

TABLE 2. Experimental data from [3] for f_l = 1GHz and f_h = 12GHz

Vgs = 0.8 V, Vds = 1.0 V

Y11l [mS]	2.79286927
Y11h [mS]	26.91850594
Y12l [mS]	0.320882852
Y12h [mS]	3.416808149
Y21l [mS]	1.740237691
Y21h [mS]	17.78438031
Y22l [mS]	2.003395586
Y22h [mS]	14.09168081
wl [deg/s]	6.283185307 x 10^9
wh [deg/s]	75.398223686 x 10^9

2) F300p35 — BSIM3v3

TABLE 3. Simulation results of the BSIM3v3 model [3] for f_l = 1GHz and f_h = 12GHz

Vgs = 0 V, Vds = 0 V

Y11l [mS]	1.86440678
Y11h [mS]	19.74576271
Y21l [mS]	0.379661017
Y21h [mS]	3.927118644
Y12l [mS]	0.351945854
Y12h [mS]	3.844331641
Y22l [mS]	2.118644068
Y22h [mS]	23.47457627
Wl [deg/s]	6.283185307 x 10^9
Wh [deg/s]	75.398223686 x 10^9

TABLE 4. Simulation results of the BSIM3v3 model [3] for f_l = 1GHz and f_h = 12GHz

Vgs = 0.8 V, Vds = 1.0 V

Y11l [mS]	3.208828523
Y11h [mS]	31.19694397
Y21l [mS]	2.376910017
Y21h [mS]	21.51952462
Y12l [mS]	0.344651952
Y12h [mS]	3.208828523
Y22l [mS]	1.728813559
Y22h [mS]	19.11864407
wl [deg/s]	6.283185307 x 10^9
wh [deg/s]	75.398223686 x 10^9

The computer realisation of the extraction procedure is performed using *Cadence PSpice* (Fig. 3).
The frequency dependence of the y_m parameter is represented by the GLAPLACE element with controlling coefficient

```
gm*(1-s*tau_v)
```

The frequencies f_l , f_h and the two-port Y-parameters for f_l and f_h are defined using PARAMETER statement in the computer model. The component values are calculated corresponding to the developed extraction procedure:

```
.PARAM fl=1GHz, fh=12GHz, pi=3.14159265,
+ wl={2*pi*fl}, wh={2*pi*fh},
+ Y11l=1.949152542e-3,
+ Y12l=0.522033898e-3, Y21l=0.46179966e-3,
+ Y22l=1.694915254e-3, Y11h=20.12711864e-3,
+ Y21h=7.008488964e-3, Y12h=6.66779661e-3,
+ Y22h=16.10169492e-3, par=0

.PARAM Cgsd={Y11l/wl}, gm_v={Y21l},
+ Cgs1_v={(Y11l-Y12l)/wl} , par=0,
+ Rg_v={sqrt(1/Y11h/Y11h-wl*wl/
+ (wh*wh*Y11l*Y11l))},
+ zn2={1+wh*Cgsd*Rg_v*wh*Cgsd*Rg_v},
+ Rsd_v={1/(Y22l+wl*Cgda*wl*Cgs1a*Rg)},
+ Csd1_v={Y22h/wh-Cgd_v} ,
+ Cgd_v={Y12h*sqrt(zn2)/wh}
```

The two-port Y-parameter calculation is reduced to a parametric analysis with parameter par = 0;1 [5]. The parameters y_{11} and y_{21} are obtained for par = 0 and the parameters y_{22} and y_{12} are obtained for par = 1 using the following macro-definitions in Probe:

```
Y11=-I(Vg)@1          Y12=-I(Vg)@2
Y21=-I(Vd)@1          Y22=-I(Vd)@2
```

VERIFICATION OF THE SIMPLIFIED MODEL

The verification of the extraction procedure for the parameters of the simplified model is made by comparing the results of *PSpice* simulation of the simplified RF model and the data form experiment and from the BSIM3v3 simulation. They are presented in Fig. 4.

The *PSpice* simulation results *Exper_Sim* correspond to the extracted model from experimental data for Y-parameters for f_l and f_h. The *PSpice* simulation results *BSIM_Sim* correspond to the extracted model from Y-parameters for f_l and f_h obtained by BSIM3v3 simulation.

The good agreement of the measured and simulated results confirms the validity of the proposed extraction methodology. The relative errors of *BSIM_Sim* simulations obtained for the maximal frequency 12GHz are: $\varepsilon_{11} = 2\%$, $\varepsilon_{12} = 1\%$, $\varepsilon_{21} = 16\%$ and $\varepsilon_{22} = 1.5\%$. The relative errors for *Exper_Sim* simulations obtained for the maximal frequency 12GHz are $\varepsilon_{11} = 0.5\%$, $\varepsilon_{12} = 2.5\%$, $\varepsilon_{21} = 8\%$ and $\varepsilon_{22} = 1.3\%$.

The comparison of the two-port *S*-parameters can be performed using postprocessing in the graphical analyzer *Probe,* utilizing corresponding macrodefinitions for Y- to S- parameter conversion, based on the simulated Y-parameters of the model shown in Fig. 3 [5]. The magnitude values of the measured *S*-parameters are introduced in the behavioral model using frequency-dependent elements of EFREQ type.

CONCLUSIONS

A direct parameter extraction methodology has been proposed for a simplified small-signal NMOSFET equivalent circuit. Extracted parameters are used to specify the equivalent circuit component values. The general-purpose circuit simulator *Cadence PSpice* is used for the computer realization of the extraction procedure. Verification of the parameter extraction methodology is made by comparing in the graphical analyzer *Probe* the results obtained in *PSpice* and the experiment.

THE AUTHORS

Prof. Dr. Marin Hristov, Technical University of Sofia, Department of Microelectronics, 1797 Sofia, Bulgaria, e-mail: mhristov@ecad.tu-sofia.bg

Assoc. Prof. Dr. Elissaveta Gadjeva, Technical University of Sofia, Department of Electronics, 1797 Sofia, Bulgaria, e-mail: egadjeva@tu-sofia.bg

MSc Eng. Olga Antonova, Ph.D. Student, Technical University of Sofia, Department of Microelectronics, 1797 Sofia, Bulgaria,
e-mail: antonova@ecad.tu-sofia.bg

MSc Eng. George Angelov, Ph.D. Student, Technical University of Sofia, Department of Microelectronics, 1797 Sofia, Bulgaria, e-mail: gva@ecad.tu-sofia.bg

ACKNOWLEDGEMENTS

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

REFERENCES

[1] Y. Tsividis, “Operation and Modeling of the MOS Transistor”, McGraw-Hill, Inc., ISBN 0-07-065523-5, 2nd edition, 1999.

[2] W. Liu, X. Jin, J. Chen, M.-C. Jeng, Z. Liu, Y. Cheng, K. Chen, M. Chan, K. Hui, J. Huang, R. Tu, P. K. Ko, and C. Hu, “BSIM3v3.2 MOSFET Model - Users’ Manual”, Technical report, Department of Electrical Engineering and Computer Sciences, University of California, Berkeley, California 94720, USA, August 1998.

[3] T. Kolding, “Compact Model Specification of RF MOSFET with DC and AC Evaluations”, Technical Report R99-1005, ISSN 0908-1224, RF Integrated Systems & Circuits (RISC) group, Institute of Electronic Systems, Aalborg University, Denmark, September 20, 1999.

[4] OrCAD PSpice and Basics, Circuit Analysis Software, OrCAD Inc., USA, 1998.

[5] E. Gadjeva, T. Koyumdjiev, S. Farchy, “Computer Modeling and Simulation of Electronic and Electrical Circuits Using OrCAD PSpice”, Meridian 22, Sofia, 2001 (in Bulgarian).

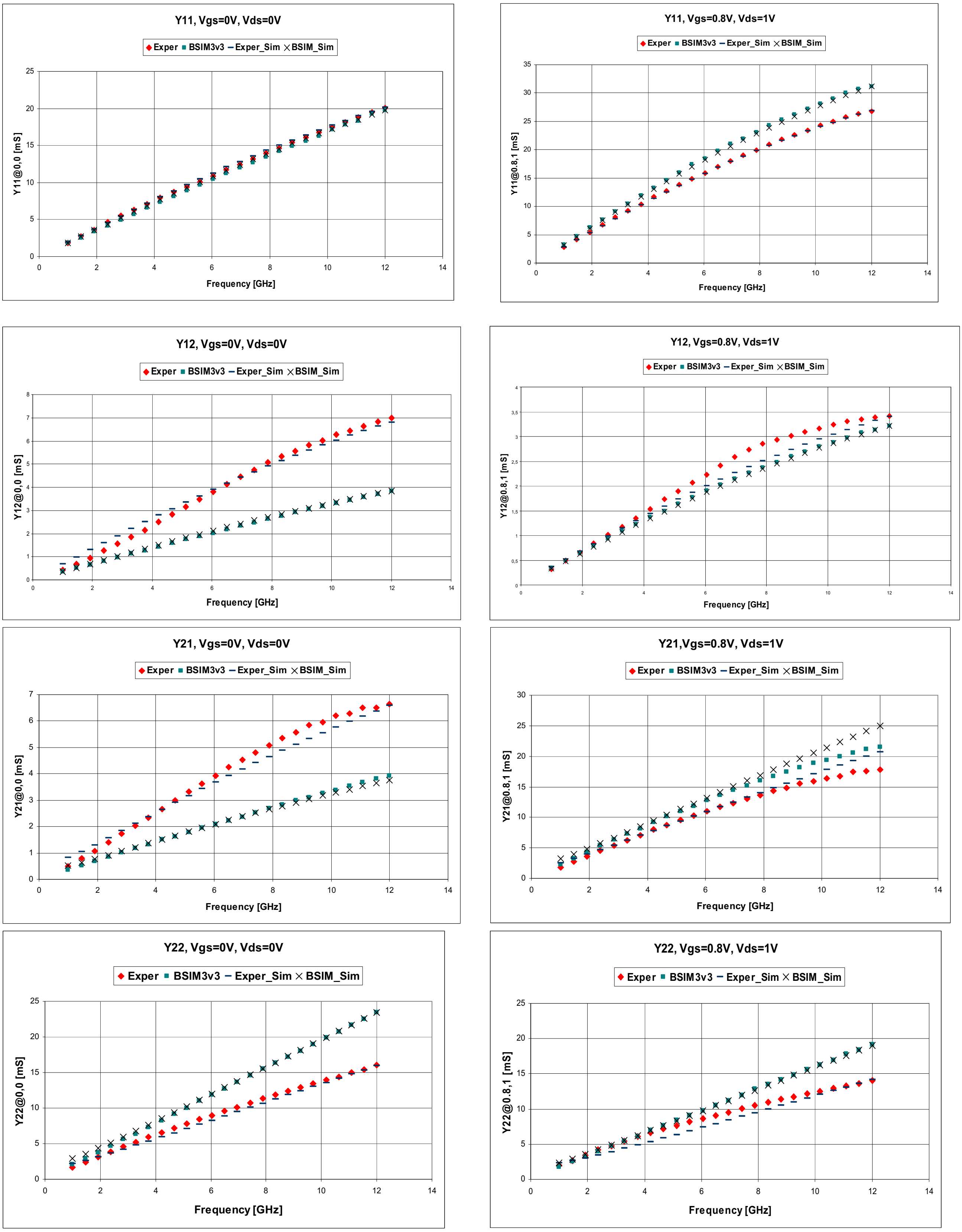

Fig.4. Verification of the extraction procedure