MICROWAVE CIRCUIT ANALYSIS USING GENERAL-PURPOSE ANALYSIS PROGRAMS

Assoc. Prof. Dr. Emanuil Nisim Djerassi*,
Assoc. Prof. Dr. Elissaveta Dimitrova Gadjeva**

**Central Laboratory of Parallel Signal Processing,
Bulgarian Academy of Sciences, e-mail: djerassi@bol.bg*
***Department of Electronics, Technical University of Sofia
e-mail: egadjeva@vmei.acad.bg*

ABSTRACT: A macromodel approach to the microwave circuit analysis is developed for the frequency analysis of microwave circuits consisting of arbitrarily connected multiport subcircuits, characterized by scattering (S) parameters. The S-parameters of the whole circuit are obtained related to the external ports. The possibilities of the general-purpose analysis program *OrCAD PSpice* for the description of elements with frequency-dependent parameters are used. The developed unified approach provides the ground for investigation of additional characteristics of microwave circuits and enhances the *PSpice* possibilities in the field of microwave circuit analysis.

I. INTRODUCTION

It is very helpful to use the enhanced possibilities of the general-purpose analysis programs such as *OrCAD PSpic*e for investigation of microwave circuit in the frequency domain, as well as for noise, stability and sensitivity analyses. Some of the factors contributing to the ever growing possibilities of the general-purpose analysis programs of the *PSpic*e type for description and investigation of microwave circuits are:

❑ improvement and greater possibilities of the input language for description of microwave circuit computer models on the basis of behavioural modelling by introducing of elements with frequency-dependent parameters;

❑ possibilities for the user to easily define library models for basic building blocks intended for different types of investigation (analysis in the frequency domain, noise analysis, sensitivity analysis, etc.), as well as the inclusion of these blocks in the corresponding model and symbol libraries;

❑ enhanced possibilities of the *Probe* graphical analyzer for visualization of the analysis results and for automatic display of the results using the correponding macro and command files, etc.;

❑ diverse possibilities of the *Probe* graphical analyzer for the post-processing

Supported by the National Foundation "Scientific Investigations" under Grant VRP-I-2/1999

of the analysis results allowing the definition of additional characteristics (e.g. total sensitivity, possibilities to compare different structures with respect to noise, sensitivity and others, etc.)

The rich possibilities of the *OrCAD PSpice* input language for description of frequency-dependent functions and for subcircuit definition allow to describe blocks realizing the matrix description of circuits consisting of *arbitrarily connected multiport subcircuits*, characterized by *scattering* (*S*) parameters. As a result, the *S*-parameters of the whole circuit are obtained related to the external ports.

The scattering (*S*) parameter system is the most general method for description of microwave circuits [1,2]. Some of the factors which contribute to the application of the *S*-parameter description for the computer simulation are:

- The possibility for the easy determination of the *S*-parameters at RF from the corresponding measured reflection and transmission quantities;
- The lumped parameter equivalent circuits represent only an approximate description of the distributed parameter components. The multiport description based on the scattering parameters allows greater accuracy;
- The circuits containing *both types* of components (with distributed parameter description and with lumped parameter description) can be analyzed accurately using the *S*-parameter description by means of multiport *S*-parameter representation of lumped parameter components.

II. DESCRIPTION OF THE MODELS

Consider the microwave circuit N consisting of m arbitrarily connected multiport components. The circuit N is considered as multiport with p external and q internal (connected between themselves) ports [1] (Fig. 1).

Each component with n_i ports is described by the equation:

$$b_j^{(i)} = \sum_{k=1}^{n_i} S_{jk}^{(i)} . a_k^{(i)} \ , \ i = 1,2,\dots m; \ \ j,k = 1,2,\dots,n_i \qquad (1)$$

where $a_j^{(i)}$ and $b_j^{(i)}$ are the corresponding falling and reflected waves for the j-th port of the i-th multiport;

$S_{jk}^{(i)}$ - transmission S-parameter (for $j \neq k$) or reflection S-parameter (for $j = k$) of the i-th multiport.

The interconnections of the components are described by the relationships between the corresponding falling of reflected waves. If the port i of the multiport A is connected with the port j of the multiport B as shown in Fig. 2, the following equations are valid:

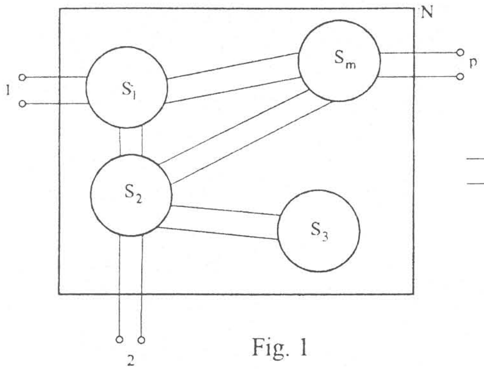

Fig. 1

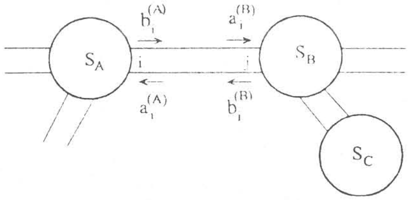

Fig. 2

$$a_i^{(A)} = b_j^{(B)}\ ;\ b_i^{(A)} = a_j^{(B)} \tag{2}$$

The independent sources, connected to the external ports, are described in the form:

$$b_g = S_g.a_g + c_g \tag{3}$$

where c_g is the wave description of the source and $S_g = 0$ for the matched sourse.

The equations (1-3) represent a full description of the whole circuit based on the S-parameter definition of the subcircuits and the independent sources, taking into account the interconnections of the elements.

The matrix representation of equations (1-3) has the form [1]:

$$\mathbf{W.a} = \mathbf{c} \tag{4}$$

where

$$\mathbf{W} = \mathbf{\Gamma} + \mathbf{\Gamma_g} - \mathbf{S} \tag{5}$$

The matrices $\mathbf{W}, \mathbf{\Gamma}, \mathbf{\Gamma_g}$ and $\mathbf{S}$ are of the order $k = p + \sum_{i=1}^{m} n_i$ and

$$\mathbf{S} = \text{diag}\ [\mathbf{S}_1, \mathbf{S}_2, \ldots, \mathbf{S}_m, \mathbf{0}_p] \tag{6}$$

is a diagonal matrix and the submatrices $\mathbf{S_1}, \mathbf{S_2}, \ldots, \mathbf{S_m}$ are the scattering S-matrices of the corresponding multiports;

The matrix $\mathbf{\Gamma}$ is an interconnection matrix, describing the connections between the internal ports where $\Gamma_{ij}=1$ if port i is connected with port j and $\Gamma_{ij}=0$ if port i is not connected with port j. The matrix $\mathbf{\Gamma_g}$ is an interconnection matrix, describing the connections between the external ports of the multiports and the independent sourses connected with these ports: $\Gamma_{gij} = 1$ if the external port i is connected with the port of the independent source j; otherwise $\Gamma_{gij} = 0$.

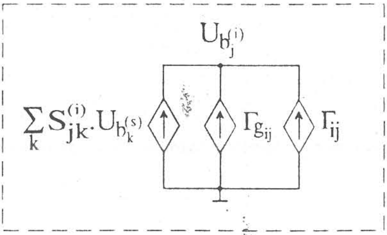

Fig. 3

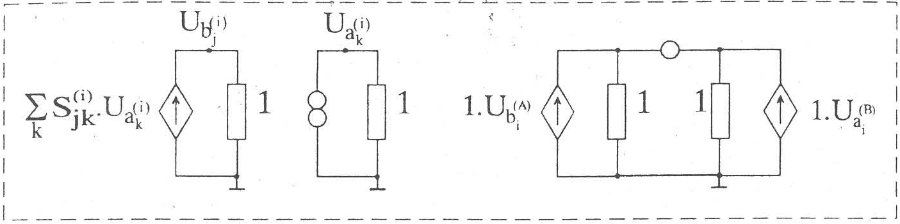

Fig. 4

III. COMPUTER MODELS FOR THE ANALYSIS IN THE FREQUENCY DOMAIN

A behavioral equivalent circuit can be constructed according to the equations (1-3). Two models are proposed for the frequency analysis of microwave circuits.

❑ The correspondence between *reflected wave* and *node voltage variable* is introduced in the first model:

$$U_{b_j}^{(i)} \Leftrightarrow b_j^{(i)} \ ; \ j = 1,2,\ldots,n_i \ ; \ i = 1,2,\ldots,m \qquad (7)$$

This model is a schematical representation of the matrix equation (4) using a set of dependent voltage controlled current sources (Fig. 3).

❑ *Both the falling and reflected waves* are represented by means of a corresponding *voltage variables* in the second model.

$$U_{b_j}^{(i)} \Leftrightarrow b_j^{(i)} \ ; \ U_{a_j}^{(k)} \Leftrightarrow a_j^{(k)} \ ; j = 1,2,\ldots,n_i \ ; \ i,k = 1,2,\ldots,m \qquad (8)$$

The relationships between the falling and reflected waves (2) are involved using the corresponding *nullator-norator equivalent circuits* in this model as shown in Fig. 4. This model allows operations with the falling and reflected waves.

The frequency (*AC Sweep*) analysis of the developed *PSpice* computer models makes it possible to determine the frequency characteristics of resulting circuit *S*-parameters.

The visualization of the *S*-parameters according to the external ports of the circuit is accomplished using the *Probe* graphical analyzer.

The frequency characteristics of the multiports using *S*-parameters can be described in two ways – using **FTABLE** or **LAPLACE** functions of the input language of the *PSpice* simulator.

❑ *using frequency look-up tables*

The frequency look-up tables **FTABLE** describe the phasors of controlling corfficients of the VCCS sources in the form:

```
Grefl 0 node_b FREQ {V(node_a)}  (<freq1>,<magn1>, <phase1>)
+ (<freq2>,<magn2>, <phase2>)...
```

The data can be described in magnitude/phase or in real part/imaginary part format.

❑ *using parameter definition and* **LAPLACE** *operator*

In this case, the scattering parameters are described as parameters in the form:

```
Grefl1 0 b LAPLACE{V(A)}{S11m*EXP(S*(S11p)/TWO_PI)}
```

where `S11m` and `S11p` are the magnitude and phase of S_{11}.

As an example *S*-parameters of the equivalent two-port are obtained for the parallel connection of two three-ports described by their *S*-parameters using the proposed nullor behavioral models.

A section of the library text file describing the reflected waves according to (1) for calculation of b_{11} has the form:

```
*b-waves description
* b11
RV1 0 B1_1 1
G11 0 B1_1 LAPLACE{V(A1_1)}{S11m*EXP(S*(S11p)/TWO_PI)}
G12 0 B1_1 LAPLACE{V(A2_1)}{S12m*EXP(S*(S12p)/TWO_PI)}
G13 0 B1_1 LAPLACE{V(A3_1)}{S13m*EXP(S*(S13p)/TWO_PI)}
```

The description of the nullor subcircuit which modells relationships (3), has the form:

```
* a-waves description
V1 A1_1 0 AC 1
R2 A2_1 0 1
E2 A2_1 0 A2_1 B2_2 1E11
R3 A3_1 0 1
E3 A3_1 0 A3_1 B3_2 1E11
```

As a result, *S*-parameters of the equivalent two-port are calculated and saved in the output file using the reflected waves at the external ports. S_{11} is obtained in the form:

```
AC ANALYSIS
  FREQ        VM(B1_1)     VP(B1_1)
1.000E+10   6.696E-01   -7.344E+01
```

Based on the proposed approach, *n*-port blocks are constructed as library elements with user-defined *S*-parameters. These universal blocks are included in corresponding *PSpice* symbol and model libraries.

CONCLUSIONS

A macromodel approach has been developed to the frequency analysis of microwave circuits consisting of arbitrarily connected multiport subcircuits, characterized by scattering (*S*) parameters. The *S*-parameters of the whole circuit are obtained related to the external ports. The possibilities of the general-purpose analysis program OrCAD *PSpice* for the description of elements with frequency-dependent parameters are used.

The computer behavioral models have been developed corresponding to the matrix equations with *S*-parameters of the single multiports and to the relationships between falling and reflected waves in the interconnection nodes. The correspondence between falling wave and node voltage variable is introduced in the model. The frequency (*AC Sweep*) analysis of the developed *PSpice* computer models makes it possible to determine the frequency characteristics of resulting circuit *S*-parameters. The phasors of the *S*-parameters are described using the **LAPLACE** or **FTABLE** function of the input language of the *PSpice* simulator.

The developed unified approach to the analysis of microwave circuits provides the ground for investigation of additional characteristics such as sensitivity and stability characteristics, and enhances the *PSpice* possibilities in the field of microwave circuit analysis.

REFERENCES

[1] G. Vendelin, A. Pavio, U. Rohde, Microwave Circuit Desing Using Linear and Nonlinear Technique, Jon Wiley & Sons, 1990.

[2] K. Gupta, R. Garg, Computer-Aided Design of Microwave Circuits, Artec House Inc., 1981.

[3] PSpice, Circuit Analysis User's Guide, *Microsim Co.*, Irvine, 1991.

[4] Application Notes, *Microsim Corp.*, USA, 1996.