RF COMPONENTS MODELING AND ANALYSIS

S. Kishore Reddy*

ABSTRACT

RF Passive components are used in the design of RF blocks as well as in system level integration. The efficient and accurate analysis of these components is required in order to meet the RF block or system level performances. I had studied the modeling of RF passive components for fast analysis approaches I had studied and developed a prototype design to simulate RF circuits in Pspice. The aim was to develop a functionality to simulate a RF block in spice simulator. Following issues were considered in this simulation

i. S-parameter file compatibility with spice simulator

*ii. Synthesis and analysis of matching circuits*

iii. Modeling of passive components

S-parameter file compatibility with spice simulator: *For the circuit design I had considered system level LNA block and made it compatible with spice format The S-parameter file for LNA was not directly compatible with the spice format. An approach was investigated and a translator was developed to make it spice compatible. Hence a generalized code for N-port S-parameter file conversion to spice format was developed.*

*Ph.D. Research Scholar Electronics, CMJ University, Meghalaya, India.

I. INTRODUCTION

Demand for low cost RF simulation is very much the need of hour. A prototype for simulating S parameters in Pspice was developed and tested for this purpose. The S-parameter simulation in Pspice is made possible using Analog Behavioral Modeling (which is a new add on in spice), together with the S-parameter sub-circuit, we are able to generate the S-parameters of the circuit The simulation however, requires the S-parameter file (touchstone format) of any device (e.g. transistor, LNA, filter etc.), which is then decomposed into frequency tables (using a programming code that was developed as a part of the prototype) that Pspice understands. So, the model or the black box whatever you may consider now has the frequency table inside it to realize that particular device

Concept of Analog Behavioral Modeling in Pspice
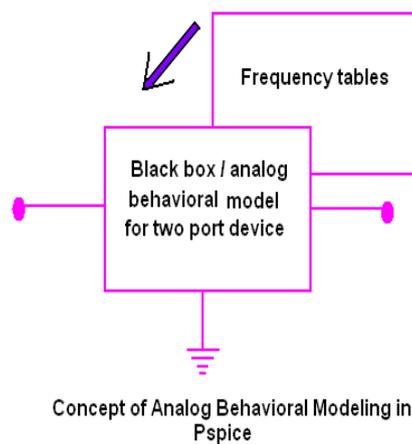
In the similar manner you can have analog behavioral models of various devices and they could when they meet a discontinuity caused by the insertion of a network into the transmission line.be connected in a meaningful way to form a circuit and the S-parameter response of the entire system could be analyzed in this way

II WHAT ARE SCATTERING PARAMETERS?

Scattering parameters or S-parameters are properties that describe the electrical behavior of linear electrical networks when undergoing various steady state stimuli by small signals.They are members of a family of similar parameters used in electronics engineering, other examples being: Y-parameters, Z-parameters, H-parameters, T-parameters or ABCD-parameters.They differ from these, in the sense that *S-parameters* do not use open or short circuit conditions to characterize a linear electrical network; instead matched and unmatched loads are used Moreover, the quantities are measured in terms of power.

Many electrical properties of networks or components may be expressed using S-parameters such as gain, return loss, voltage standing wave ratio (VSWR), reflection coefficient and amplifier stability. The term 'scattering' is more common to optical engineering than RF engineering, referring to the effect observed when a plane electromagnetic wave is incident on an obstruction or passes across dissimilar dielectric media. In the context of S-parameters, scattering refers to the way in which the traveling currents and voltages in a transmission line are affected when they meet a discontinuity caused by the insertion of a network into the transmission line This is equivalent to the wave meeting impedance differing from the line's characteristic impedance.

Although applicable at any frequency, S-parameters are mostly used for networks operating at radio frequency (RF) and microwave frequencies. S-parameters change with the measurement frequency so this must be included for any S-parameter measurements stated, in addition to the characteristic impedance or system impedance. S-parameters are readily represented in matrix form and obey the rules of matrix algebra.

Historically, an electrical *network* would have comprised a 'black box' containing various interconnected basic electrical circuit components or lumped elements such as resistors capacitors, inductors or transistors. For the S-parameter definition, it is understood that a network may contain any components provided that the entire network behaves linearly with incident small signals. It may also include many typical communication system components or 'blocks' N^2 such as amplifiers, attenuators, filters, couplers and equalizers provided they are also operating under linear and defined conditions.

An electrical network to be described by S-parameters may have any number of ports. Ports are the points at which electrical currents either enter or exit the network. Sometimes these are referred to as pairs of 'terminals so for example a 2-port network is equivalent to a 4-terminal network, though this terminology is unusual with S-parameters, because most S-parameter measurements are made at frequencies where coaxial connectors are more appropriate.

The S-parameter matrix describing an N-port network will be square of dimension 'N' and will therefore contain N^2 elements. At the test frequency each element or S-parameter is represented by a unitless complex number, thus representing magnitude and angle, or amplitude and phase The complex number may either be expressed in rectangular form or, more commonly, in polar form. The S-parameter magnitude may be expressed in linear form or logarithmic form. When expressed in logarithmic form, magnitude has the "dimensionless unit" of decibels.The S- parameter angle is most frequently expressed in

degrees but occasionally in radians.Any S- parameter may be displayed graphically on a polar diagram by a dot for one frequency or a locus for a range of frequencies. If it applies to one port only (being of the form S_{nn}), it may be displayed on an impedance or admittance Smith Chart normalised to the system impedance. The Smith Chart allows simple conversion between the parameter, S_{nn}

equivalent to the voltage reflection coefficient and the associated (normalised) impedance (or admittance) 'seen' a that port.

The following information must be defined when specifying any S-parameter:

(1) The characteristic impedance (often 50).

(2) The allocation of port numbers.

(3) Conditions which may affect the network, such as frequency, temperature, control voltage, and bias current, where applicable.

III THE S-PARAMETER SUPPORT IN PSPICE:

As of now there is no methodology present within Pspice to measure the S-parameters. Using the S-parameter subcircuit (explained in 1.8) we can calculate both the transmission (S21 and S12) and reflection (S11 and S22) parameters of a given circuit.In microwave and RF design work, a Pspice subcircuit whose S-parameters are defined as tables is also useful; the data corresponds to that provided by many manufacturers for microwave transistors and other microwave devices. The Pspice envoriment understands the S-parameter files provided if they are supplied to it in the form of the frequency tables for each scattering parameter separately i.e. if we want to measure the S-parameter of a two port network, then Pspice must be provided with S-parameter file consisting of four frequency tables for S11, S12, S21, S22 separately.

The frequency tables are in general available in many formats like citi format touchstone format etc. In our implementation in Pspice we will however restrict ourselves to touchstone format only. Again the touchstone format falls under three main types viz.

1) frequency, magnitude, phase

2) frequency, magnitude(dB), phase

3) frequency, real, imaginary

All the above types have been taken into account in the implementation.The first step in the process is to read the *.snp file provided and convert it into *.lib file which the Pspice understands.The VC++ program basically reads the *.snp file and then breaks the single large frequency table of the S-parameters into separate frequency tables each for a particular S-parameter.The output is then written into a *.lib file.

Pspice then creates a model (with the same name as the file name) out of the *.olb file .(the *.olb file is generated from the *.lib file).The model is then attached in the schematic to measure the S-parameters.However,one should bear in mind that Pspice would require n number of schematics for a n-SRUW QHWZRUN¶V 6-parameter measurement.

IV ANALOG BEHAVIORAL MODELING

In the good old days, to model a simple LOG function, you reached for a diode from the SPICE tool shed and embarked on synthesizing the log function. More complex functions required deeper math and device theory knowledge. So, we used to spend more time developing and debugging the math function than actually using it in your system or circuit. But now, SPICES's added capabilities – analog functions, have greatly simplified and extended the available modeling power.

Two Types of Modeling

Modeling devices or systems in SPICE falls into two general categories: structural and behavioral.

STRUCTURAL MODELING: For every component in the actual circuit (resistor, capacitor, diode, etc.), there's corresponding component in the SPICE model. The structure of the actual circuit is preserved in the SPICE circuit.

BEHAVIORAL MODELING: The behavior of several components (typically grouped as functional block) is modeled by various SPICE devices. The SPICE model may bear little resemblance to the actual circuit. The focus is on the input/output relationships of the block.

If accuracy is desired than structural modeling is the way to go. Almost every circuit component and its effect is accounted for. However, generating the actual circuit may be difficult, time consuming, and SPICE run time grow painfully long for larger circuits. Behavioral models, on the other hand, describe a functional block with relatively few components, making simulations zip along much faster. On the downside, some secondary circuits effects may be lost, popping up in actual circuit performance.

V REVIEW OF STUDIES

VI DISCUSSION

The one of the major limitation in Pspice envoirnment is that it requires N number of test benches/schematics for measuring the S-parameter of a N port network. However, one could

implement switches to change the source and load positions and thus we could implement this entire prototype design within a single schematic. Another notable limitation with Pspice is that it gives augmented values of the phase.It has been noticed during our course of experimentation that for the reflected coefficients of the S-parameters Pspice adds an additional phase of 360o to that particular phase part where ever there is a change of sign in comparision to the first value of phase in the table and the phase in this case remains less than 360o respectively.However,for the transmission coefficients of the S-parameters Pspice adds an additional phase of 360o to the phase part from then on in cycles where ever there is a change of sign in comparision to the first value of phase in the table and the phase in this case would exceed 360o .

The coversion program suffers from the fact that it will be able to handle only upto nine ports on account that the from ten ports on the extention *.snp would be of four characters which is not permissible in the windows envoriment.

REFERENCES

[1] Microwave Engineering, third edition, David M. Pozar, University of Massachusetts at Amherst, John Wiley & Sons, Inc., Copyright 2005

[2] Computer-Aided Design of Microwave Circuits, K.C. Gupta, Ramesh Garg, Rakesh Chadha, ARTECH house, Inc. , Copyright 1981.

[3] Create S-Parameter Subcircuits for Microwave and RF Applications, by John S. Gerig Wideband Associates.

[4] Analog Behavioral Modeling, eCircuit Center.

http://www.ecircuitcenter.com/Circuits/ABM1/ABM1.htm.

[5] Analog Behavioral Modeling, eCircuit Center.

http://www.ecircuitcenter.com/Circuits/ABM1/ABM1.htm.

[6]www.artechhouse.com/GetBLOB.asp?Name=Bahl_CH12.pdf

[7]http://www.microwaves101.com/encyclopedia/Wilkinson_splitters.cfm

[8]http://www.microwaves101.com/encyclopedia/Branchline_couplers.cfm

[9]www.artechhouse.com/GetBLOB.asp?Name=Bahl_CH12.pdf

[10]radarcafe.com/documents/Grebennikov_Impedance_Matching_Lecture_2.ppt

[10]http://www.microwaves101.com/encylopedia/Smithchart.cfm

[12] http://www.sss-mag.com/smith.html#tutor

[13] Mysteries of the Smith Chart, Stephen D. Stearns, K6OIK,Chief Technologist, TRW Firestorm Wireless Communication Products, stearns@ieee.org..