Regular paper

# COMBININING ELECTROMAGNETIC FIELD COMPUTATION WITH CIRCUIT SIMULATION TO DESIGN MICROWAVE AMPLIFIERS

B. EBERT[1]

**Abstract:** To design amplifiers operating at microwave frequencies demands accurate models for microstrip transmission lines connecting lumped elements such as resistors, capacitors and transistors mounted on a printed circuit board. Commercially available software tools provide the means to combine electromagnetic field computation with circuit simulation. Using this approach the deviations between simulation and measurement are far less in comparison with other methods.

**Keywords:** Electromagnetic field computation, circuit simulation, hybrid simulations

## INTRODUCTION

For sufficiently low frequencies, phase shift of currents and voltages along transmission lines can be neglected. This assumption will lose its validity when line lengths exceed one-twentieth of the wavelength of the transmitted signal. The wavelength can be calculated by

$$\lambda=\frac{c}{\sqrt{\epsilon_{r,eff}}\,f} \tag{1}$$

whereas $\epsilon_{r,eff}$ represents the effective permittivity of air and substrate of the printed circuit board (PCB). Phase shift and propagation delay can be encountered either by transmission line models based on empirically derived formulas or by electromagnetic field computation. The former has the advantage of consuming less computing time. The latter yields higher accuracy and supports arbitrary shapes of transmission line segments.

Power gain, stabilty, noise, bandwidth, input and output matching are important issues of common amplifier design. At microwave frequencies, the amplifier is no longer fully described by lumped elements. Based on the abovementioned explanations it is advisable to include also the transmission line elements connecting the lumped resistors, capacitors, transistors, etc.. In doing so, the behavior of the designed amplifier should be close to the simulation results.

Electromagnetic field computation and circuit simulations were for a long time seen as two stand-alone topics. Shifting operating frequencies of electronics into the GHz range necessitates a unification in the design process. Enhanced computing power and improved solvers have accelerated the convergence of both in recent years [1]. Nowadays, there are several commercial computer aided design tools capable to perform such electromagnetic/circuit co-simulation. The present work is based on simulations with ADS® from Agilent Technologies but should be considered as an overview of the design flow process for hybrid simulations. Microwave amplfiers are only one possible application among others like mixers, oscillators and so on.

## THEORY OF OPERATION

A microwave amplifier can be either monolithically integrated in a chip or mounted on a printed circuit board with discrete devices. The articel is confined to the latter.

At the beginning, the simulation program has to be fed with accurate substrate parameters. Permittivity, height and dielectric loss factor of the PCB material are of crucial importance. Conductivity of the metal layer permits to calculate resistive losses within the strip lines and ground planes.

Next the amplifier is designed in the schematic editor. Then all footprints of the utilized devices have to be drawn and connected in the layout editor (fig. 1). Additionally all terminals of a device are represented by internal ports (drawn as arrows). Every port serves as an excitation point for the electromagnetic field computation. ADS® uses the method of moments [2] to calculate scattering parameters of planar circuits. Consequently, the PCB layout represents a linear n-port device characterized by its scattering matrix in the frequency domain. Due to the field computation, electromagnetic interference between striplines are included.

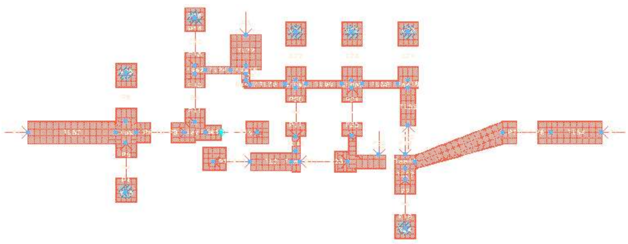

**Fig. 1** - *Top layer of the PCB for a common-emitter amplifier*

The linear n-port is placed in the schematic editor and its terminals have to be connected to the corresponding lumped device. A simulation is now performed as an electromagnetic/circuit co-simulation.

## MEASUREMENT VS. SIMULATION RESULTS

The designed broadband microwave amplifier consists of a common-emitter stage responsable for a voltage gain of 20 dB. An intermediate common-collector stage isolates the output from the amplifier stage (fig. 2). The circuit is realized with SiGe heterojunction bipolar transistors BFP 620 from Infineon Technologies mounted on a standard FR-4 as PCB material.

Three kinds of simulation were performed in order to expose their different level of accuracy and speed. The first one comprises only lumped elements (denoted by Discrete in fig. 4 and 5). Interconnections between these elements are totally neglected. Consequently simulation accuracy is limited for the sake of speed. The second one

[1] Technische Universität Ilmenau, Helmholtzplatz 2, D-98684 Ilmenau, Germany, E-mail: bjoern.ebert@TU-Ilmenau.de

uses microstrip models to simulate the interconnections (denoted by Microstrip in fig. 4 and 5) . Although speed remains sufficiently high, accuracy improves significantly. The last kind of simulation is based on full-wave formulation for stripline modeling using general Green functions (denoted by Method of Moments MoM in fig. 4 and 5). In comparison with the other simulations speed is considerably reduced but exactitude should outperform.

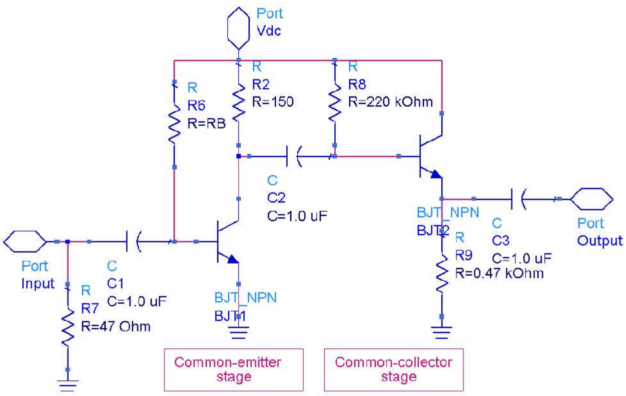


**Fig.2** - *Schematic of the designed broadband amplifier*

The designed amplifier was assembled (fig. 3) and thoroughly characterized with an HP 8720C network analyzer. Magnitude and phase of the voltage gain $G$ can be derived from S-parameters using

$$G = \frac{s_{21}}{1 + s_{11}} \qquad (2)$$

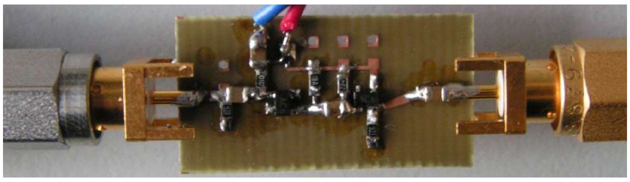

**Fig.3** - *Photography of the assembled amplifier*

A comparison between simulation and measurement results shown in figure 4 and 5 exhibits that all kinds of simulation are appropriated to predict the circuit behavior up to 300 MHz. For higher frequencies discrete simulations are no longer suitable. Using microstrip models, deviations between simulation and measurement are less but obvious. Especially the magnitude shows a sharp resonance which was not observable during measurement. The best match between measurement and simulation is reached with an electromagnetic field computation. Nonetheless, above 3 GHz deviations become also evident. Possible reasons are variations of the parameters of transistors, capacitors, resistors and the printed circuit board. The ground plane of the PCB is supposed to be infinite large and so reflections occurring at the boundaries of the board are disregarded by the electromagnetic field solver.

## CONCLUSION

In this paper a short review of combining circuit simulation with electromagnetic field computation is given. This approach predicts behavior of circuits operating at radio frequencies more accurate than other methods do. The simulations are verified by measurements so that the aptitude of different simulation methods can be clearly observed.

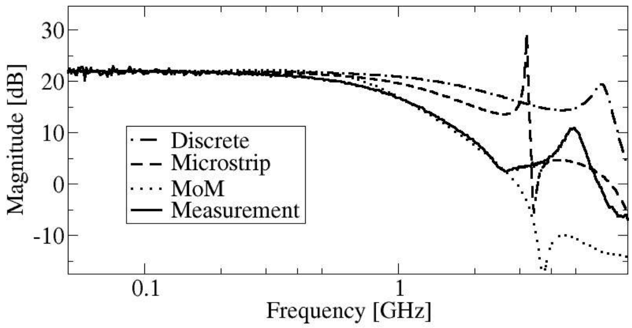


**Fig.4** - *Comparison between measurement and simulation results for the magnitude of voltage gain*

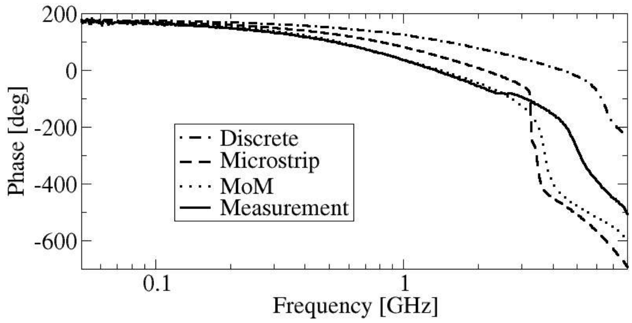


**Fig.5** - *Comparison between measurement and simulation results for the phase of voltage gain*

## REFERENCES


[1] R. Achar, M.S. Nakhla: “Efficient transient simulation of embedded subnetworks characterized by S-parameters in the presence of nonlinear elements”, *IEEE Transactions on Microwave Theory and Techniques*, Vol. 46, 1998, pp. 2356-2363.

[2] R.F. Harrington: *“Field computation by moment methods”*, IEEE Press, 1993, Piscataway.


Björn Ebert was born in Ilmenau, Germany, in 1981. He received the Diploma degree in electrical engineering and information technology from the University of Technology Ilmenau, Germany, in 2005. As one part of his studies, he stayed at University of Savoie, France, and did a traineeship at the CEA-Grenoble, France.

In 2005, he joined the RSFQ design-group of the Department Electrical Engineering and Electromagnetic Field at the University of Technology Ilmenau. His main research fields are analog and RF circuit design for applications in a cryogenic operation environment.