Applied Mathematics and Computation 294 (2017) 238–252

Contents lists available at ScienceDirect

Applied Mathematics and Computation

journal homepage: www.elsevier.com/locate/amc

Radio frequency and microwave numerical simulation techniques based on multivariate formulations

Jorge F. Oliveira[a,b,*]

[a] School of Technology and Management, Polytechnic Institute of Leiria, Morro do Lena, Alto Vieiro, Apartado 4163, 2411-901 Leiria, Portugal

[b] Institute of Telecommunications, University of Aveiro, Campus Universitário de Santiago, 3810-193 Aveiro, Portugal

ARTICLE INFO

Keywords:
Partial differential equations
Numerical simulation
Radio frequency circuits

ABSTRACT

Simulating modern wireless communication systems is today a hot topic. Serious difficulties arise when these systems are highly nonlinear heterogeneous circuits, combining radio frequency (RF) and baseband analog circuitry and digital components, operating in widely distinct time scales. This paper briefly reviews some recently proposed innovative techniques based on multivariate (multidimensional) formulations, which were especially conceived to efficiently simulate modern wireless systems, as a whole, at the circuit level. In order to evidence the achievements brought by the multivariate strategies into the field of RF and microwave circuit simulation, as well as their most relevant limitations, several powerful numerical techniques, operating in time, frequency, or hybrid (combination) domains, will be addressed.

1. Introduction

In general, success in nonlinear electronic design, especially in integrated circuit (IC) design where manufacturing is expensive and probing internal nodes is difficult or prohibitive, relies on the extensive use of computer-aided analysis tools. Device modeling and numerical simulation avoid breadboarding and physical prototyping, helping engineers to verify correctness and debug circuits throughout their design, and so decreasing product development costs.

In the last two decades RF and microwave system design has been found as a significant part of the electronic semiconductor industry's portfolio. Over the years, the necessity of continuously providing new wireless systems' functionalities and higher transmission rates, as also the need to improve transmitters' efficiency, has been gradually reshaping wireless architectures. Heterogeneous circuits combining baseband blocks, digital blocks and RF blocks, in the same substrate, are commonly found today. Because of that, RF and microwave circuit simulation has been conducted to an increasing challenging scenario of heterogeneous broadband and strongly nonlinear wireless communication circuits, presenting a wide variety of slowly varying and fast changing state variables (node voltages and branch currents). Thus, RF and microwave design has been an important booster for numerical simulation and device modeling development.

In order to efficiently simulate modern wireless communication circuits several innovative numerical techniques have been discussed in the scientific literature in the last few years. This paper is focused on the presentation of powerful numerical techniques based on multivariate formulations, which describe the multirate behavior of the circuits using multiple

* Correspondence address: School of Technology and Management, Polytechnic Institute of Leiria, Morro do Lena, Alto Vieiro, Apartado 4163, 2411-901 Leiria, Portugal.

E-mail address: jorge.oliveira@ipleiria.pt

http://dx.doi.org/10.1016/j.amc.2016.09.015
0096-3003/

time variables (artificial time scales), and so the solutions of the problems are computed within multidimensional frameworks. In order to provide a general overview on these simulation techniques, putting in evidence their strength and their weakness, this paper is organized as follows.

After this brief introduction, Section 2 provides some general background material on standard circuit simulation techniques commonly used for computing the numerical solution of ordinary (univariate) problems. Section 3 addresses the fundamentals of the multivariate formulation. Then, Section 4 is essentially devoted to the presentation of the numerical techniques based on the multivariate formulation. Several powerful numerical techniques, operating in time, frequency, or hybrid (combination) domains, will be reviewed. Finally, Section 5 concludes this paper.

2. Circuit simulation fundamentals

2.1. Mathematical model of an electronic circuit

The behavior of an electronic circuit can be described with a *differential algebraic equations'* (DAE) system, involving electrical voltages, electrical currents, electrical charges and magnetic fluxes. The DAE system can be constructed from a circuit description using, for example, nodal analysis, which involves applying the Kirchhoff current law to each node in the circuit, and applying the constitutive or branch equations to each circuit element. Under the quasi-static assumption [1,2], the DAE system has, in general, the following form,

$$\boldsymbol{p}(\boldsymbol{y}(t)) + \frac{d\boldsymbol{q}(\boldsymbol{y}(t))}{dt} = \boldsymbol{x}(t), \tag{2.1}$$

where $\boldsymbol{x}(t) \in \mathbb{R}^n$ stands for the excitation (independent voltage and current sources) vector, and $\boldsymbol{y}(t) \in \mathbb{R}^n$stands for the state variable (node voltages and branch currents) vector. $\boldsymbol{p}(\boldsymbol{y}(t))$ stands for all memoryless linear or nonlinear elements, as resistors, nonlinear voltage-controlled current sources, etc., while $\boldsymbol{q}(\boldsymbol{y}(t))$ models dynamic linear or nonlinear elements, as capacitors (represented as linear or nonlinear voltage-dependent electric charges), or inductors (represented as linear or nonlinear current-dependent magnetic fluxes).

The system of (2.1) represents the general mathematical formulation of lumped problems. However, distributed devices can be also included in this DAE formulation if these are substituted by their lumped-element equivalent circuit models, or are replaced, as whole sub-circuits, by reduced order models derived from their frequency-domain representation. The substitution of distributed devices by lumped-equivalent models is particularly satisfactory when the size of the circuit elements is small in comparison to the wavelengths. Typical examples of these are emerging RF technologies, as is the case of radio frequency integrated circuits (RFICs), which integrate digital high-speed complementary metal-oxide-semiconductor CMOS baseband processing and RFCMOS hardware into a single chip.

2.2. Transient simulation

Obtaining the solution of (2.1) over a specified time interval $[t_0, t_{Final}]$ with a specific initial condition $\boldsymbol{y}(t_0)=\boldsymbol{y}_0$, is what is usually known as an *initial value problem*, and evaluating such solution is frequently referred to as *transient analysis*. The most natural way to compute $\boldsymbol{y}(t)$ is to numerically time-step integrate (2.1) directly in time domain. So, it should be of no surprise that this straightforward technique was used in the first digital computer programs of circuit analysis and is still nowadays widely used. It is present in all SPICE (which means Simulation Program with Integrated Circuit Emphasis) or SPICE-like computer programs [3]. In order to numerically time-step integrate the DAE system of (2.1) commercial tools use initial value solvers, such as linear multistep methods [4,5], or Runge–Kutta methods [4,5], (the popular one-step integrators). Both of these classes of methods can provide a wide variety of explicit and implicit numerical schemes, with very distinct properties in terms of order (accuracy) and numerical stability.

2.3. Steady-state simulation

Sometimes electronics designers, as is the case of RF and microwave designers, are not interested in the circuits' transient response, but, instead, in their steady-state regimes. This is so because certain aspects of circuits' performance are better characterized, or simply only defined, in steady-state (e.g., harmonic or intermodulation distortion, noise, power, gain, impedance, etc.). Initial value solvers, as linear multistep methods, or Runge–Kutta methods, which were tailored for finding the circuit's transient response, are not adequate for computing the steady-state because they have to pass through the lengthy process of integrating all transients, and expecting them to vanish.

Computing the periodic steady-state response of an electronic circuit involves finding the initial condition, $\boldsymbol{y}(t_0)$, for the DAE system that describes the circuit's operation, such that the solution at the end of one period matches the initial condition, i.e., $\boldsymbol{y}(t_0)=\boldsymbol{y}(t_0+T)$, where T is the period. Problems of this form, those of finding the solution to a system of differential equations that satisfies constraints at two or more distinct points in time, are referred to as *boundary value problems*. In this particular case, we have a *periodic boundary value problem* that can be formulated as

$$\boldsymbol{p}(\boldsymbol{y}(t)) + \frac{d\boldsymbol{q}(\boldsymbol{y}(t))}{dt} = \boldsymbol{x}(t), \quad \boldsymbol{y}(t_0) = \boldsymbol{y}(t_0+T), \quad t_0 \le t \le t_0+T, \quad \boldsymbol{y}(t) \in \mathbb{R}^n, \tag{2.2}$$

where the condition $\boldsymbol{y}(t_0)=\boldsymbol{y}(t_0+T)$ is known as the *periodic boundary condition*.