A Hybrid Method for Simulation of Mixed Circuit and Electromagnetic Problems in Frequency Domain

Sohrab Safavi and Jonas Ekman

Department of Computer Science, Electrical and Space Engineering, Luleå University of Technology, Luleå, Sweden
Sohrab.Safavi@LTU.SE, Jonas.Ekman@LTU.SE

Abstract: The simulation of mixed circuit and electromagnetic (EM) structures in frequency domain has application in most electromagnetic problems. Many of the developed methods work by modifying a SPICE-like solver to incorporate an EM numerical method, or to extend the EM numerical method to handle the circuit components (e.g. diodes, transistors, ..) by re-implementing their model definitions. In order to find the frequency response of the system, a DC bias point calculation is required for which a novel hybrid technique has been proposed. This technique utilizes the Partial Element Equivalent Circuit (PEEC) method as the EM solver and employs OrCAD as the circuit solver. The link between the two solvers is acquired by defining the circuits connected to the EM structure as ports. Each port's current-voltage relations are approximated by a linear system of equations and the Newton-Raphson algorithm is employed to calculate the bias point of the whole structure. Having the bias points, the frequency behavior of the system is acquired by calculating the admittance models of the ports using OrCAD. To demonstrate the capability of the developed method, a high frequency amplifier circuit connected to two transmission lines is examined. The results show good agreement and acceptable accuracy has been obtained that shows the feasibility of the developed method to solve this type of mixed problems. Since OrCAD is held responsible for the circuit simulations, the need to modify a SPICE-like solver or to re-implement the definitions of the circuit devices has been removed. On the other hand, by manipulating the system of equations and proper optimization techniques, an optimal solver can be achieved.

Keywords: Partial Element Equivalent Circuit (PEEC), SPICE, frequency-domain simulation, hybrid electromagnetic/circuit analysis.

1. Introduction

The simulation of mixed circuit and electromagnetic (EM) structures in frequency domain has attracted special attention over the last decade and simulation techniques where both the EM structures and lumped passive/active elements can be incorporated are desired. The effects of EM radiations, crosstalk, couplings, packaging, orientation of wires and components, selection of dielectric substrate, and other structural parameters in the PCB assembly all should be considered in the design of new products. A co-design process should be utilized in which the design of the EM structure, such as an antenna, and the active/passive circuit components connected, such as the antenna’s drive circuit, can be performed simultaneously.

In a frequency domain response, all the non-linear circuit components are considered to be working linearly around their bias point, which makes a DC bias point calculation an inevitable part of a frequency domain analysis of a mixed system of circuits and EM structures. After a DC analysis, when the bias-point of components is detected, all the non-linear components can be replaced with their small-signal
models at that dc operating point and also network models such as admittance models can be extracted and used to examine the frequency behavior of the system.

The PEEC method [1] is a numerical method in electromagnetics which is based on an integral form of Maxwell’s equations and proposes an equivalent circuit for a mixed conductor/dielectric structure. By defining (partial) coefficients of potential [2] and partial inductances [3], PEEC transforms the Maxwell’s system of equations into a system of circuit equations described by the Kirchhoff’s laws. Having an equivalent circuit for the EM structure provides an easy method to make a link between the EM structure and the circuit components connected to it. For example, the equivalent circuit of an antenna can be connected directly to its driver circuit, making possible to simulate the complete system at once.

The approach proposed in this paper is a hybrid method which works based on defining the circuits connected to the EM structure as multi-terminal networks or ports. The system of equations includes two sections, first the equations obtained for the EM problem by using the PEEC method, and the second one contains a linear system of equations for each port, referred to as port characteristic. In the DC bias point calculation, the port characteristic is basically a linear approximation for the I-V curves of the corresponding port obtained by applying multi-variable Taylor’s expansion to the port I-V relations. The unknown coefficients in the Taylor’s expansion [4] are acquired by using OrCAD as a SPICE solver. The Newton-Raphson algorithm [5] is then used to solve the whole system of equations. By having the bias points of the system, a frequency domain calculation is made possible and the admittance models of the ports can be obtained. Again, OrCAD is used to calculate the unknown coefficients in the admittance models of the ports. To solve the resulted system of equations, a dedicated solver is developed in which the system of equations can be manipulated and proper factorization and optimization techniques/libraries can be applied [6]. OrCAD is used to simulate the circuit parts of the system which means that no component model e.g. diode, BJT, MOSFET, etc. need to be re-implemented in the developed solver [7].

2. THE PEEC METHOD

Starting from the electric field integral equation and by discretizing the EM structures into volume cells and surface cells, and using the definitions of partial inductance and coefficients of potential, and also ignoring the retardation (quasi-static situations), the following system of equation [1] can be obtained

\[
\begin{bmatrix}
-\Pi & -(R + L_p \frac{d}{dt}) \\
-P^{-1} \frac{d}{dt} & \Pi^T
\end{bmatrix}
\begin{bmatrix}
\Phi \\
I_L
\end{bmatrix} =
\begin{bmatrix}
V_s \\
I_s
\end{bmatrix}
\]

(1)

where \( R \) is the resistance matrix, \( L_p \) is the matrix of coefficients of partial inductances [3], and \( P \) is the matrix of coefficients of potentials [2]. \( \Pi \) is the connectivity matrix and describes the connections between the volume cells and the surface cells. \( V_s \) and \( I_s \) are source vectors and \( \Phi \) and \( I_L \) are the potentials of the surface cells and the currents through the volume cells, respectively. This formulation will be referred to as MNA (Modified Nodal Analysis) formulation later on [8].

PEEC model, the values for \( R, L_p, P \) and \( \Pi \), is the same in both time-domain and frequency-domain, and the frequency-domain MNA formulation can be extracted without requiring to recalculate the PEEC model. Equation (1) can be simply reformulated for frequency domain by replacing the time derivatives \( \frac{\partial}{\partial t} \) with \( j\omega \), which results in a linear system of equations of the form \( A(j\omega)x = b(j\omega) \) as follows,

\[
\begin{bmatrix}
-\Pi & -(R + j\omega L_p) \\
j\omega P^{-1} & \Pi^T
\end{bmatrix}
\begin{bmatrix}
\Phi \\
I_L
\end{bmatrix} =
\begin{bmatrix}
V_s \\
I_s
\end{bmatrix}
\]

(2)

3. MULTI-TERMINAL CIRCUIT NETWORKS (PORTS)

The idea proposed here works based on dividing the combined EM/circuit problem into a PEEC model with multitude of multi-terminal circuit networks (ports) connected to it, as depicted in Fig. 1a. Ports are numbered from \( I \) to \( M \). Port \( p \) is connected to nodes \( \text{node}(p,1) \) to \( \text{node}(p,N_p) \) where \( N_p \) is the
number of terminals of the corresponding port. For port $p$, by defining the terminal $N_p$ as the "potential reference", voltages of the other terminals of the port related to this reference can be defined as

$$v_{p,n} = \Phi_{p,n} - \Phi_{p,N_p}.$$  

where $\Phi_{p,n}$ is the potential of the $n$’th terminal of port $p$, or equivalently the potential of $node(p,n)$. Current flowing out of the $n$’th terminal of the port is labeled as $i_{p,n}$ and is assumed to be a function of voltages of the port terminals, thus port $p$ can be characterized by

$$i_p = f_p(v_p),$$  

where $i_p$ is a $(N_p - 1) \times 1$ vector of $i_{p,n}$, $v_p$ is a $(N_p - 1) \times 1$ vector containing $v_{p,n}$, and $f_p$ is a $(N_p - 1) \times 1$ vector of non-linear functions $f_{p,n}$ where $1 \leq n < N_p$. First to note is that the currents going into the port should be equal to the currents flowing out of it, or

$$\sum_{n=1}^{N_p} i_{p,n} = 0.$$  

Since the MNA system equations represent linear relationships, non-linear equations of (4) should be modified and linearized in order to be put in the MNA matrix formulation [5], [4]. The hybrid method works based on applying the multi-variable Taylor’s expansion on terminal currents ($i_p$) around $v_p = v_{p,0}$ and ignoring the second and higher order terms, which will result in a linear approximation of the currents of the terminals as follows,

$$i_p \approx f_p(v_{p,0}) + \frac{\partial f_p}{\partial v_p} \bigg|_{v_p=v_{p,0}} (v_p - v_{p,0}),$$  

The $(N_p - 1) \times (N_p - 1)$ matrix $\frac{\partial f_p}{\partial v_p}$ is the Jacobian of $f_p$ with respect to $v_p$. Equation (6) is a linearized form of (4) and can be used in the MNA formulation. However, unlike the case of linear components, where potentials and currents at each time point are calculated in a single step, an iterative procedure known as Newton-Raphson algorithm [4], [5] should be used. The Newton-Raphson algorithm updates the unknown values $f_p(v_{p,0})$ and $\frac{\partial f_p}{\partial v_p} \bigg|_{v_p=v_{p,0}}$ at each iteration in the MNA formulation $Ax = b$ follows.
At k’th iteration,
1) make an initial guess for each potential and current \(x_k\).
2) linearize all non-linear equations based on \(x_k\): (e.g. calculate \(f_p(v_{p,0})\) and \(\frac{\partial f_p}{\partial v_p}\)|\(v_p=v_{p,0}\) in (6)).
3) build matrices \(A_k\) and \(b_k\) based on all linear and linearized equations.
4) solve the system \(A_k x_{k+1} = b_k\) to calculate \(x_{k+1}\).
5) if \(x_k\) and \(x_{k+1}\) are close enough (convergence), the solution has converged and \(x_{k+1}\) is the solution, otherwise replace the previous initial guesses with the current solution, i.e. \(x_k ← x_{k+1}\) and go to step (2).

As an example, in Fig. 1b a three-terminal port is depicted in detail. The third terminal is being considered to be the potential reference (ground) for this port. The currents flowing out of the terminals are functions of the defined voltages \(v_1\) and \(v_2\) and can be written as

\[i_n = f_n(v_1, v_2), \quad n = 1, 2\]  

(7)

Applying (6) will transform the non-linear system of equations (7) into a linear approximate system of

\[i_n = f_n(v_{1,0}, v_{2,0}) + \frac{\partial f_n}{\partial v_1}\bigg|_{v_1=v_{1,0},v_2=v_{2,0}} (v_1 - v_{1,0}) + \frac{\partial f_n}{\partial v_2}\bigg|_{v_1=v_{1,0},v_2=v_{2,0}} (v_2 - v_{2,0}), \quad n = 1, 2\]  

(8)

in which the Taylor’s expansion has been performed around \(v_1 = v_{1,0}\) and \(v_2 = v_{2,0}\).

4. The Hybrid Method

The key point in the proposed hybrid method is to calculate \(i_n|_{v_n=v_{n,0}}\) and the Jacobian matrix in (6) by employing OrCAD circuit solver instead of using analytical formulations. By this technique, the EM solver does not need to have any knowledge of the components or their I-V equations inside the ports, neither is it required to use any analytical formulation for calculation of the Jacobian.

The idea can be explained better by examining a simple case of three-terminal network. The goal is to employ OrCAD to calculate the unknown coefficients in equations (8). It can be seen that \(f_n(v_{1,0}, v_{2,0})\) is basically the current through the n’th terminal when the first and second terminal are excited with voltages \(v_1 = v_{1,0}\) and \(v_2 = v_{2,0}\) respectively. As mentioned earlier, the third terminal is being considered as the potential reference or the ground for this port. Hence, one OrCAD simulation is this one with a voltage source \(V_1 = v_{1,0}\) connected to terminals 1 and 3, and another voltage source \(V_2 = v_{2,0}\) connected between terminals 2 and 3. By reading the currents \(I_1\) and \(I_2\) through the voltage sources, \(f_1\) and \(f_2\) will be obtained.

In order to calculate \(\frac{\partial f_1}{\partial v_1}\bigg|_{v_1=v_{1,0},v_2=v_{2,0}}\), the first terminal should be excited with voltage \(V_1' = V_1 + \Delta V\), and the second terminal with a voltage \(V_2' = V_2\). The currents through the voltage sources \(V_1'\) and \(V_2'\) will be \(I_1'\) and \(I_2'\) respectively. \(\frac{\partial f_1}{\partial v_1}\) can be simply approximated as

\[\frac{\partial f_1}{\partial v_1} ≈ \frac{I_1' - I_1}{V_1' - V_1} = \frac{I_1' - I_1}{\Delta V},\]  

(9)

In the same way, another OrCAD simulation should be performed to calculate \(\frac{\partial f_2}{\partial v_2}\) and \(\frac{\partial f_2}{\partial v_1}\) in which the second terminal is excited by \(V_2'' = V_2 + \Delta V\), and the first terminal with a voltage \(V_1'' = V_1\). Thus, only by three OrCAD simulations, the whole system of equations for the port can be obtained at each iteration.

In DC analysis, all inductors are modeled as short circuits and all capacitors and the mutual electric couplings and mutual magnetic couplings have to be excluded. Therefore, the PEEC model will be pure resistive. The linear system of equations (5) and (6) can be appended to the MNA system of equations and solved using the Newton-Raphson algorithm. The modified version of the PEEC system of equations, (1), for a DC analysis can be written as
the equivalent circuit of the EM structure in OrCAD where and AC responses of the whole system. Then, the same simulations of the whole circuit are performed between the two TLs. The BJT employed is a BFR92A, a wide band 5GHz NPN transistor in SOT-23 package (Philips Semiconductors 1997). The developed hybrid method is used to find the DC between the results has been obtained.

The AC analysis is the simplest type of analysis. After a DC analysis all the non-linear components are considered to be working linearly around their bias point and network models such as admittance model can be extracted. Thus, port \( p \) can be characterized by

\[
i_p(j\omega) = Y_p(j\omega)v_p(j\omega).
\]

\( Y_p(j\omega) \) is a \((N_p - 1) \times (N_p - 1)\) matrix and can be simply obtained by \( N_p - 1 \) OrCAD simulation. In each simulation, only one terminal will be excited with an AC voltage source of amplitude 1 while other terminals are connected to AC voltage sources of amplitude 0. By reading the AC currents through the AC voltage sources, the unknown parameters \( Y_p(i, j) \) will be obtained. In order to provide the proper DC bias, the terminals should be also connected to DC voltage sources in series with the AC voltage sources. The voltages of these sources are acquired from the previous DC simulation.

Back to the three-terminal port example, two OrCAD simulations are required. In the first simulation \( v_1 \) will be \( 1\angle0^\circ \) and \( v_2 \) equal to zero, in which reading the currents through these voltage sources \( i_1 \) and \( i_2 \) will result in values for \( Y_{11} \) and \( Y_{21} \) as follows

\[
Y_{11} = \left. \frac{i_1}{v_1} \right|_{v_2=0}, \tag{12}
\]

\[
Y_{21} = \left. \frac{i_2}{v_1} \right|_{v_2=0}. \tag{13}
\]

Second simulation exciting \( v_1 = 0 \) and \( v_2 = 1\angle0^\circ \) will result in the values for \( Y_{12} \) and \( Y_{22} \).

\[
\begin{bmatrix}
-\Pi & -R & 0 \\
0 & \Pi^T & 0 \\
\Pi_p J_p & 0 & -\bar{I}_p \\
0 & 0 & \bar{K}_p
\end{bmatrix}
\begin{bmatrix}
\Phi \\
\bar{I}_p \\
I_p
\end{bmatrix}
= \begin{bmatrix}
V_s \\
0 \\
-f_p(\Pi_p \Phi_0) + \Pi_p J_p \Phi_0
\end{bmatrix}, \tag{10}
\]

in which \( I_L \) is equal to the current flowing into the resistors of the PEEC model, \( \bar{I}_p \) is an identity matrix of size \((N_p - 1) \times (N_p - 1)\), and \( \bar{K}_p \) is a vector of size \( N_p \times 1 \) with all elements equal to 1 accounting for equation (5). \( \Pi_p \) is the connectivity matrix of size \((N_p - 1) \times N\), \( N \) being the number of nodes in the PEEC model or equivalently the size of \( \Phi \), describing the connections between the EM structure (the PEEC nodes) and the port terminals. \( \Phi_0 \) is a vector containing the potentials of the PEEC nodes at previous iteration. More ports can be appended to this formulation (10) in the same way.

The AC analysis is the simplest type of analysis. After a DC analysis all the non-linear components are considered to be working linearly around their bias point and network models such as admittance model can be extracted. Thus, port \( p \) can be characterized by

\[
i_p(j\omega) = Y_p(j\omega)v_p(j\omega).
\]

\( Y_p(j\omega) \) is a \((N_p - 1) \times (N_p - 1)\) matrix and can be simply obtained by \( N_p - 1 \) OrCAD simulation. In each simulation, only one terminal will be excited with an AC voltage source of amplitude 1 while other terminals are connected to AC voltage sources of amplitude 0. By reading the AC currents through the AC voltage sources, the unknown parameters \( Y_p(i, j) \) will be obtained. In order to provide the proper DC bias, the terminals should be also connected to DC voltage sources in series with the AC voltage sources. The voltages of these sources are acquired from the previous DC simulation.

Back to the three-terminal port example, two OrCAD simulations are required. In the first simulation \( v_1 \) will be \( 1\angle0^\circ \) and \( v_2 \) equal to zero, in which reading the currents through these voltage sources \( i_1 \) and \( i_2 \) will result in values for \( Y_{11} \) and \( Y_{21} \) as follows

\[
Y_{11} = \left. \frac{i_1}{v_1} \right|_{v_2=0}, \tag{12}
\]

\[
Y_{21} = \left. \frac{i_2}{v_1} \right|_{v_2=0}. \tag{13}
\]

Second simulation exciting \( v_1 = 0 \) and \( v_2 = 1\angle0^\circ \) will result in the values for \( Y_{12} \) and \( Y_{22} \).

5. NUMERICAL EXAMPLES

To test the reliability of the developed method, a class-A power amplifier depicted in Fig. 2a is examined. The PEEC model consists of two copper parallel plate transmission lines \((l = 50\, \text{mm},\ w = 20\, \mu\text{m},\ t = 1\, \mu\text{m},\ \text{and} \ d = 20\, \mu\text{m})\) separated in x-direction by 10\, \text{mm}. Each of the conductors are uniformly discretized with \( n_x = 10\), \( n_y = 4\), and \( n_z = 0\) nodes along length, width, and thickness directions, respectively, which results in 376 volume cells and 220 surface cells. Three ports have been defined (shaded areas in Fig. 2a), a sinusoidal current source with a frequency of 800 MHz in series with a 50\, \Omega resistor at the source end, one 50\, \Omega resistor at the load end, and one 3-terminal port connected in between the two TLs. The BJT employed is a BFR92A, a wide band 5GHz NPN transistor in SOT-23 plastic package (Philips Semiconductors 1997). The developed hybrid method is used to find the DC and AC responses of the whole system. Then, the same simulations of the whole circuit are performed in OrCAD where the equivalent circuit of the EM structure is used. The results from both methods are compared. The DC simulation shows a agreement between the results and then the AC simulation for frequency range of 1 MHz to 2 GHz for 50 frequencies is performed. The AC voltages at port’s entries, acquired from both OrCAD and the hybrid method, are presented in Fig. 2b. A very good agreement between the results has been obtained.
6. Conclusion

A novel technique has been developed to simulate mixed circuit and electromagnetic problems in frequency domain. It is assumed that problem can be divided into multitude of multi-terminal circuit networks (ports) connected to EM structures. By using the multi-variable Taylor’s expansion, a linear system of equation for the current-voltage relations of each network has been obtained. OrCAD circuit solver is employed to calculate the unknown coefficients in the linear equations. The resulted equations are appended to the PEEC system of equations and solved in a dedicated solver to obtain the DC bias points. Having the bias points, the frequency behavior of the system is acquired by calculating the admittance models of the ports using OrCAD. An example including an EM structure and linear/non-linear circuits is examined. Good agreement of the results shows the capability of the developed method to solve this type of mixed problems. The next step is to extend this method to treat multiple-terminal circuit networks in time-domain.

References