A Piecewise Approach to the Transient Simulation of Hybrid Systems Involving Multiple Timescales

***Anand Ramachandran*[1], *Andreas C. Cangellaris*[1]**

[1]University of Illinois at Urbana-Champaign, Department of Electrical and Computer Engineering
357 Everitt Lab, MC-702, 1406 West Green St., Urbana, IL 61801 USA
Phone: 217-333-6037, Email: cangella@uiuc.edu

Abstract

A methodology is described for the piecewise transient simulation of complex electronic systems involving distinct subsystems that evolve at different rates. The method uses dependent sources to account for the interactions between the different subsystems. In this manner, each subsystem can be simulated using a numerical integration scheme and timestep that is most appropriate to capture the dynamics involved in the physical operation of that subsystem. The example of transmission lines feeding and loading two coupled short dipole antennas is used to demonstrate the applicability of the technique.

1. Introduction

Modern electronic systems are becoming increasingly complex, involving many different subsystems each performing a different physical function in conjunction with the other subsystems. Meanwhile, frequencies of operation continue to get higher and device and package sizes are simultaneously shrinking. Contemporary electronic designs may utilize frequencies up to tens or hundreds of gigahertz. In addition, many different technologies may be used within a single package to serve the need for highly integrated functionality within a single system. This means that no longer can the different components of a package or system be designed independently without consideration given to the limitations and requirements posed by the other components. An abstraction of a system consisting of several different physical domains is shown in Figure 1 below.

Consequently, it is imperative to consider the distributed, electromagnetic contribution in the simulation of such global, hybrid systems in order to be able to accurately account for the myriad physical effects in play. However, it is computationally prohibitive to simulate an entire system with electromagnetic accuracy. This poses the problem of combining a full-wave, electromagnetic simulation and the simulation of the associated nonlinear, lumped circuitry that drives and loads a distributed element. Also, the need to consider other physical effects at play is becoming increasingly important. For example, nonlinearities due to thermal or magnetic effects can appear in the signal path, resulting in passive intermodulation noise sources which can be severely debilitating to the integrity of modern communication systems. These kinds of effects must be accounted for in a numerical simulation in order for the simulation to be useful to a designer.

This work introduces a hybrid solver which concurrently performs time integrations using both a standard circuit solver based on an MNA formulation as well as a FDTD solver. The different subdomains of the system are linked by appropriately defined dependent sources. One key advantage of this method is that it allows asynchronous integration between the different subdomains of a system. In addition, it allows different numerical integration schemes to be applied to the different subdomains of the system, where the choice of integration scheme and timestep would be guided by the physical attributes of each subdomain.

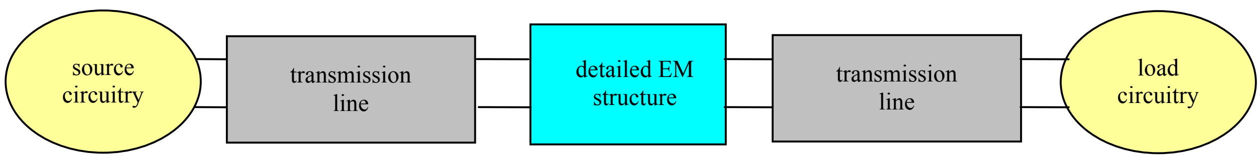


Figure 1: A typical system showing the interconnection of five physical domains. In the modeling methodology presented here, each of these domains is integrated individually, with the interconnections to the other domains becoming valid at the appropriate time points using voltage and current source boundary conditions.

2. Formulation of the Decomposition Scheme

The decomposition scheme presented in this work is based on a similar decomposition scheme introduced for the purpose of a harmonic-balance simulation in [1]. Consider an arbitrary network S consisting of two subnetworks S_1 and S_2 that are connected in a port-based model. Denote the transfer voltage and current between the two subnetworks as v(t) and i(t), respectively. If the two subnetworks S_1 and S_2 do not contain dependent sources which refer to the value of the state of the other network, then the network S can be equivalently represented by replacing the port-based interconnection to each system with equivalent voltage and current sources v(t) and i(t), as shown in Figure 2. Since the voltage and current characteristic seen at the ports by each of the subnetworks S_1 and S_2 remains unchanged, it follows from the well-known substitution theorem of network theory that all of the voltages in both S_1 and S_2 remain unchanged.

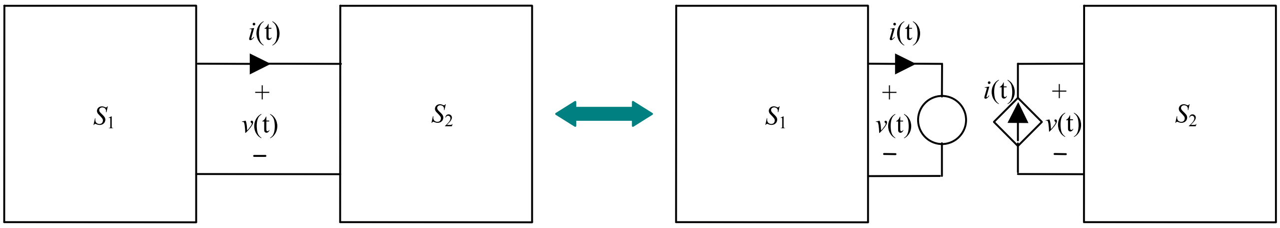


Figure 2: Arbitrary network S consisting of two subnetworks S_1 and S_2 interconnected in a port-based manner. The port-based interconnection can equivalently be represented using voltage and current sources of the same characteristic.

The simulation scheme presented in this paper is based on a modification to the equivalence described above. If a piecewise constant voltage source v_1 is augmented to subnetwork S_1, then denote the current flowing out of the terminals of S_1 by i_1. Now, if a piecewise constant current source $i_2 = i_1$ is augmented to subnetwork S_2, then define the voltage appearing at the terminals of subnetwork S_2 by v_2. At each subsequent update in time, update the value of the appropriate source v_1 or i_2 based on the last known value of the equivalent voltage or current of the other system. In this manner, each subsystem can be integrated independently. Clearly, this sort of decomposition scheme can be generalized to an arbitrary number of subnetworks. Care must be taken in assigning the direction of the current sources that are appended so as to ensure that positive feedback loops are not created which will result in instabilities.

Following this approach, each subsystem can be numerically integrated using an integration scheme and timestep that is most appropriate given the physical characteristics of that subsystem. Because sparse linear solvers scale superlinearly with the size of the matrix, it is expected that this decomposition method should be considerably more efficient than if the entire system were to be solved at once. In addition, because each subsystem is solved asynchronously, each update in time (which evolves based on the smallest timestep among all of the subsystems) only requires the solution of those subsystems which are required to be updated according to their timescale. The state of the remaining systems is unchanged, resulting in a piecewise constant voltage or current boundary source applied to the updated systems.

One important issue that must be addressed in such a hybrid simulation scheme is the timesteps that are admitted by each scheme. For instance, SPICE-like circuit simulators are often based on unconditionally stable numerical integration schemes such as backward Euler or the trapezoidal rule in the calculation of derivatives. This means that the timestep need only be chosen small enough to be able to resolve the highest frequency component of a signal to a required level of accuracy. On the other hand, standard FDTD simulation is not unconditionally stable. Rather, the timestep is limited so that the wave does not travel beyond one cell in the spatial discretization over the course of one timestep, known as the Courant limit. In addition, the spatial grid must be sampled finely enough to be able to accurately represent the frequencies present in the signals propagating through the structure. These considerations must be taken into account when choosing the timestep for each subsystem.

3. Simulation Study

In order to demonstrate the capabilities of the methodology introduced in the previous section, consider the example of two coupled short dipole antennas fed and loaded by matched transmission lines, as shown in Figure 3. Each antenna is 0.5 m in length, the antennas are 0.25 m apart and the antenna wire diameter is taken to be 50 mm. It is known that the antennas resonate at a frequency of approximately 0.25 GHz. The admittance and impedance parameters were extracted for this pair of dipoles using the commercial software tool WIPL-D by treating the two dipoles as a two-port system [2]. Using the impedance parameters, an equivalent circuit representation for the coupled antenna system was generated using the PRePFit algorithm [3-5], valid over the frequency range 0.1 – 1.0 GHz. PRePFit applies passive rational fitting of the real part of a network transfer function to capture the resonances of the system, and then casts the resulting rational approximation in the form of a SPICE equivalent circuit consisting of linear, passive elements.

Assume the transmission lines feeding and loading the dipoles are ideal and lossless, each with length 4 m, characteristic impedance of 50 Ω and propagation velocity of 3 x 10^8 m/s. Given these parameters, the expected delay in the line would be 13.33 ns. Also, assume the transmission lines are terminated with matched source and load impedances of 50 Ω.

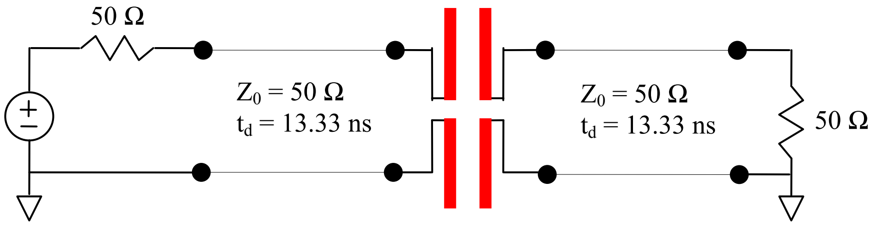


Figure 3: Two coupled dipole antennas (shown in red) fed and loaded by matched transmission lines.

Following the decomposition scheme outlined in the previous section, the above system can be decomposed into separate lumped and distributed subsystems in order to facilitate the hybrid simulation, as shown in Figure 4. It is important to note that this decomposition is not based on the usual Norton equivalent representation of the distributed system based on a grid capacitance, which is often used to perform hybrid FDTD-circuit simulation [6]. Rather, the decomposition is based on the substitution theorem from network theory, whereby a subsystem with a given port characteristic is replaced by dependent sources having the same characteristic.

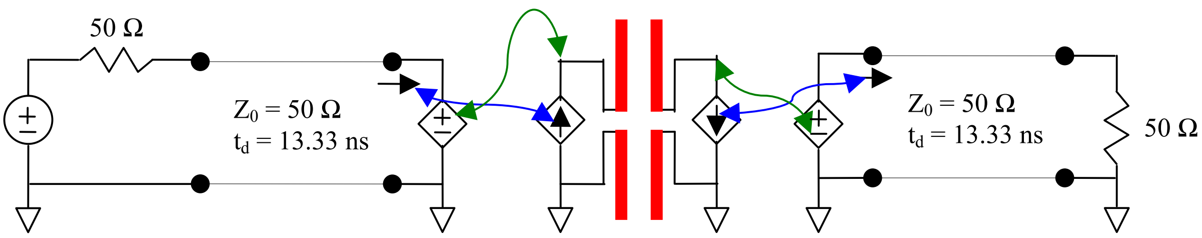
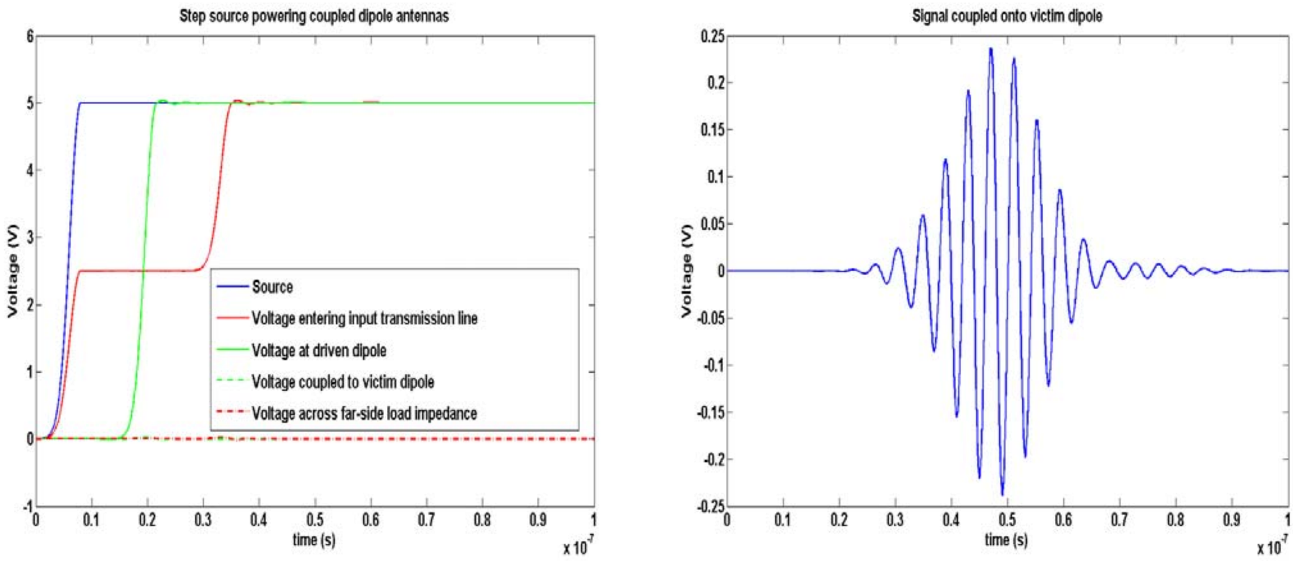


Figure 4: Decomposed system. The arrows show the associations of the dependent sources with their respective voltage nodes or current branches.

First, consider a 5V step voltage source with a rise time of 8 ns powering the system. Using a rule-of-thumb approximation that the bandwidth BW of a step function with rise time T_r is given by $BW = 2/T_r$, the bandwidth of this signal will then be approximately 0.25 GHz. This means that the dipoles are expected to display capacitive behavior in this regime of operation. The problem is simulated by discretizing the transmission lines into N_z = 200 uniform segments and using a uniform timestep of 33.33 ps, equivalent to a rate of half of the Courant limit. A uniform timestep of twice the Courant limit was used to simulate the equivalent circuit representing the two dipoles. In other words, there was a factor of four between the two timescales used in this simulation. The problem was simulated for a total time of 100 ns. The results are shown in Figure 5a below, where the voltages on the far-end transmission line show behavior proportional to the time-derivative of the near-end voltages, as would be expected by the capacitive coupling between the two dipoles.

Next, consider an input waveform consisting of a fixed frequency sinusoid operating at 0.25 GHz with amplitude 5V modulated by a unit strength Gaussian-shaped pulse with a width of 32 ns. This waveform was used to simulate the same setup as in the previous example. Figure 5b shows the resulting signal at the load of the secondary dipole.



(a) (b)

Figure 5: (a) Results of simulation showing the coupled dipoles displaying capacitive behavior. (b) Gaussian-modulated waveform recorded at the load resistance of the transmission line connected to the victim dipole.

4. Conclusion

A methodology for the hybrid simulation of complex systems utilizing multiple timescales has been developed and applied to the problem of measuring the step response of coupled dipole antennas. While the linear equivalent circuit representing the coupled dipoles was not complex enough to achieve a speed-up worthy of mention, it is apparent that the technique is promising and could produce a considerable improvement in performance for larger, more complex, and possibly nonlinear subsystems.

5. Acknowledgment

This work is supported in part by the U.S. Army Research Office as a Multi-disciplinary University Research Initiative on Standoff Inverse Analysis and Manipulation of Electronic Systems under grant number W911NF-05-1-0337.

6. References

1. M. Nakhla and J. Vlach, "A Piecewise Harmonic Balance Technique for Determination of Periodic Response of Nonlinear Systems," *IEEE Transactions on Circuits and Systems*, Vol. CAS-23, no. 2, Feb. 1976, pp. 85-91.

2. M. C. Taylor and T. K. Sarkar, "WIPL-D model and simulation results for a 46cm diameter impulse radiating antenna (IRA)," *Antenna and Propagation Society International Symposium Proceedings*, July 2005, pp. 553-556.

3. A. Y. Woo and A. C. Cangellaris, "PRePFit: Passive rational approximation of a passive network transfer function from its real part," *Intl. Journal of RF and Microwave Computer-Aided Engineering*, to be published.

4. A. Woo and A. C. Cangellaris, "Passive rational fitting of a driving-point impedance from its real part," *Proceedings of 10th IEEE Workshop on Signal Propagation in Interconnects*, Berlin, Germany, May 2006.

5. A. Y. Woo and A. C. Cangellaris, "Real-part sufficiency and its application to the rational function fitting of passive electromagnetic responses," *IEEE MTT-S International Microwave Symposium*, Honolulu, Hawaii, May 2007.

6. A. Taflove and S. G. Hagness, *Computational Electrodynamics: The Time-Domain Finite-Difference Method*, 2nd ed., Artech House, Norwood, MA, 2000.