

## TIME-DOMAIN INTERCONNECT MODELING FOR UWB APPLICATIONS

Dr. Michael C. Heimlich  
Director, Consulting Services  
mike@mwoffice.com

Dr. Evgeny Wasserman  
Development Engineer  
evgeny@mwoffice.com

Ryan Welch  
Director, Engineering Consultants  
ryanmwoffice.com

APPLIED WAVE RESEARCH, INC.  
1960 East Grand Avenue, Suite 430  
El Segundo, CA USA 90245  
(310) 726-3000  
[www.mwoffice.com](http://www.mwoffice.com)

### **Extended Abstract**

Interest in ultra wideband (UWB) techniques has grown remarkably in recent years, especially in the area of high data rate, short distance telecommunication systems. Immunity to multipath effects and high throughput makes UWB attractive as a substitute for, or in conjunction with, other technologies, such as 802.11 or general packet radio service (GPRS). The advantages of UWB come at a price—very short pulses. UWB circuit designs by their very nature involve signals over tens of octaves into the microwave and millimeter wave frequency regimes. Frequency-domain modeling and simulation are normally used for such high frequency designs because the distributed nature of the interconnect as well as many circuit features are sized on the order of the signal wavelength. Time-domain simulation is often more desirable for pulsed applications because it is the technique of choice for many analog designers, however, the designer most often compromises in the modeling of interconnects by relying on idealized, dispersionless transmission line models or overly simplified lumped element equivalent circuits. This paper demonstrates pulsed response simulations for UWB applications using a commercially available, SPICE-based interconnect modeling technique, which is both fast and accurate in comparison to frequency-domain harmonic balancing. A brief overview of the modeling theory is also included.

### **Introduction**

Designers of circuits for analog in general, and pulsed applications in particular, have traditionally been comfortable using time-domain tools, such as SPICE. SPICE offers a generalized approach to nonlinear circuit simulation but can be time consuming where

there is a large discrepancy between the time constant(s) for circuit elements and the fundamental frequency and harmonics of the signals and stimuli. The situation has become further complicated by shrinking manufacturing dimensions and ever higher operating frequencies and clock rates that require frequency-domain modeling of interconnects<sup>1</sup> and passive components<sup>2</sup>, as well as designs with significant harmonic content<sup>3</sup>. Frequency-domain techniques have generally been used as circuit dimensions become smaller and are more typically the methods used by microwave and millimeter wave designers. While s-parameters and harmonic balance (HB) can be effectively used to model UWB designs, they may not be the “comfort zone” for many designers.

A combination of the two simulation technologies is needed—time- and frequency-domain, SPICE and HB. In this paper a previously reported application technique<sup>4</sup> is updated using exactly that: a combination of Synposys’ HSPICE<sup>®</sup> as the time-domain simulator and the HB simulator in the Microwave Office<sup>™</sup> design suite from Applied Wave Research (AWR<sup>™</sup>). AWR’s Analog Office<sup>™</sup> design suite provides the common interface and model equivalency technology between the two simulators. Two cases are discussed and both the theory and application are provided for each. In the first case, a transmission line with and without fan-out is examined. In the second case, arbitrary metal is examined.

#### Case 1 Theory: Transmission Line

Pulses in general create a tremendous number of harmonics. The harmonics can be determined by considering the mathematical definition of the pulse

$$f(t) = \begin{cases} 0, & t/2 < t < T_0/2 \\ A, & 0 \leq t \leq t/2 \end{cases}$$

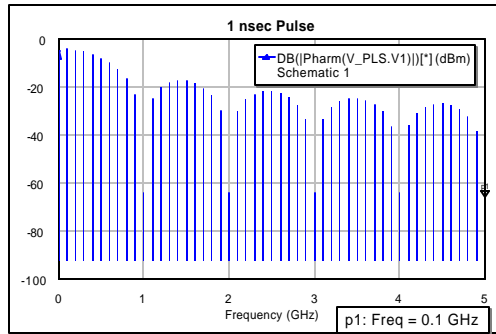
Where t is time on a periodic domain  $T_0$  such that

$$t \in [-T_0, T_0],$$

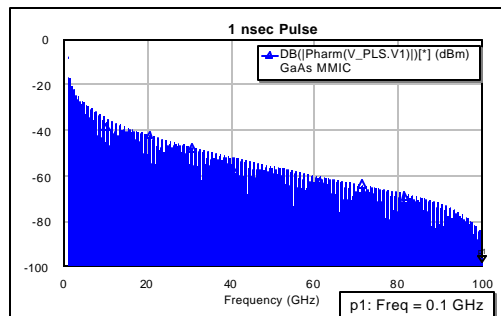
t is the pulse width and A is the pulse amplitude. Then the Fourier transform of the pulse is taken

$$F(f) = \sin(2\pi f t) / 2\pi f$$

Shown in Figure 1 is the pulse response for a 1 nsec ideal pulse with a 10 nsec or 100MHz period using HB simulation.



(a)



(b)

Figure 1 – Pulse response for  $t=1\text{nsec}$  and  $T_0=10\text{nsec}$  simulated using harmonic balance. The two graphs differ in their frequency scales: (a) from 0 GHz to 5 GHz, and b) from 0 GHz to 100 GHz

It is evident that significant energy is still found at 5.0 GHz, or 5 harmonics from the fundamental frequency of the pulse, and it can be seen in the Figure 1b that this is true into the hundreds of harmonics. What determines the need to accurately model these harmonics is two-fold: the application and the manufacturing process. Applications can be as varied as digital electronics to analog pulse generators and UWB communication systems. Digital electronics are most often concerned with the fundamental and efforts are made to minimize the need to model the energy in the higher order harmonics. The effect of the interconnect is only considered after the electrical design, as a parasitic. This leads to a design where the circuit designer only uses a fraction of the  $f_t$  of the transistors. For analog electronics, the designer is more apt to use more of the  $f_t$  of the device.

The effect of manufacturing technology can be seen in Figure 2, below, which shows the pulse responses of a microstrip line. Identical transmission lines are constructed and simulated on silicon (Si) complementary metal oxide semiconductor (CMOS) and gallium arsenide (GaAs) monolithic microwave integrated circuit (MMIC) material. The loss of the bulk silicon substrate contributes to significant attenuation of the higher frequencies. This reduces the need to model the structure at higher frequencies for digital applications, as any signal energy at these frequencies is not delivered to the load. For UWB, the

desire is to support better higher frequency characteristics, so metallization is often buffered from the Si substrate by a metal ground barrier.

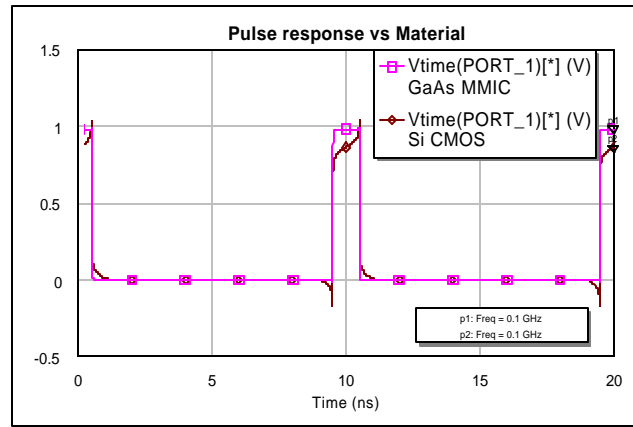


Figure 2 – Pulse response for the 1nsec pulse in GaAs MMIC vs Si CMOS IC technologies.

With so much energy at higher frequencies, the transmission line cannot be treated as a frequency-independent structure as might be modeled with simple resistors (R) and capacitors (C). Normal RC parasitic extraction is insufficient because it is not frequency-dependent and does not typically include hundreds of frequency points. Lossless and dispersionless transmission line models will not work either, since the material systems clearly have conductive and dielectric loss and finite frequency limits. Including a true transmission line model in SPICE is computationally expensive because of the need for a convolution integration to incorporate the stored energy in the line. All of these traditional alternatives are undesirable, and a new approach is needed.

To accurately model the transmission line structure, a recursive convolution method is used as incorporated in the W-element in HSPICE. This model is based on the rational approximation of two well-behaved functions, the characteristic admittance and the delayless propagation function, W. Normally, this supports simulation up to 1-3 GHz, but its application has recently been extended by an order of magnitude<sup>3</sup>.

Through the Analog Office interface, the user specifies locally or globally a frequency-domain description of how detailed a model should be extracted. Because this is done concurrently with implementing the interconnect, these nets are referred to as intelligent nets, or iNets™. iNets and subcircuits can be specified as to the complexity of the model and the detail of the frequency-dependence. In nominatively pulsed applications, like high speed digital, it is assumed that significant energy and information content is not transmitted in the higher order harmonics of the digital pulse, and is therefore only required to model a few tens of harmonics. For UWB applications it is generally necessary to consider a hundred or more harmonics to accurately model the structures.

#### Case 1 Example: Transmission Line

In this first example (Figure 3), a single transmission line connecting two resistors is considered. The interconnect between the two resistors is specified in the Analog Office software as an iNet.

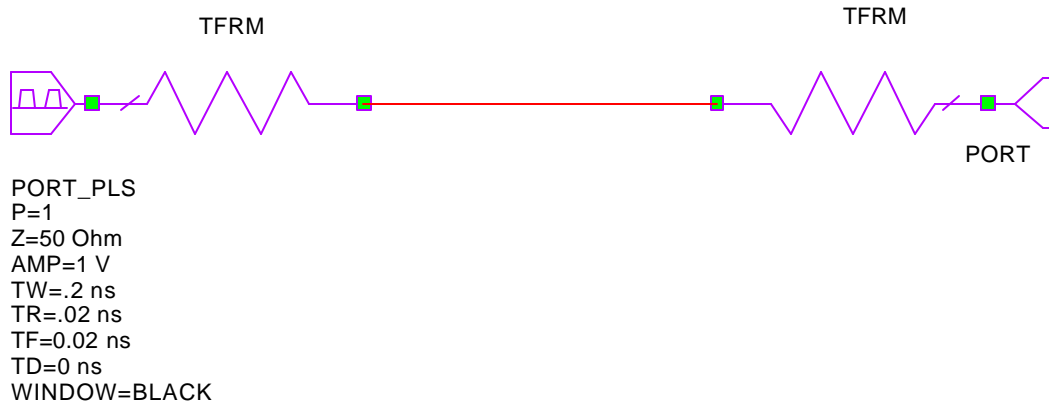


Figure 3 – Analog Office schematic for two resistors interconnected and driven by a 200 psec pulse. The connection between the two resistors in this schematic is a routed interconnect in the layout (see Figure 5a).

In the Analog Office product, we simply change the iNet connecting the two resistors from “implemented” to “unimplemented” and then route the line. Analog Office software is set up to automatically create a highly accurate model for HSPICE from the HB description of the iNet. Although the software models the iNet as something other than an ideal short-circuit, effectively, the schematic is simulated as in Figure 4,

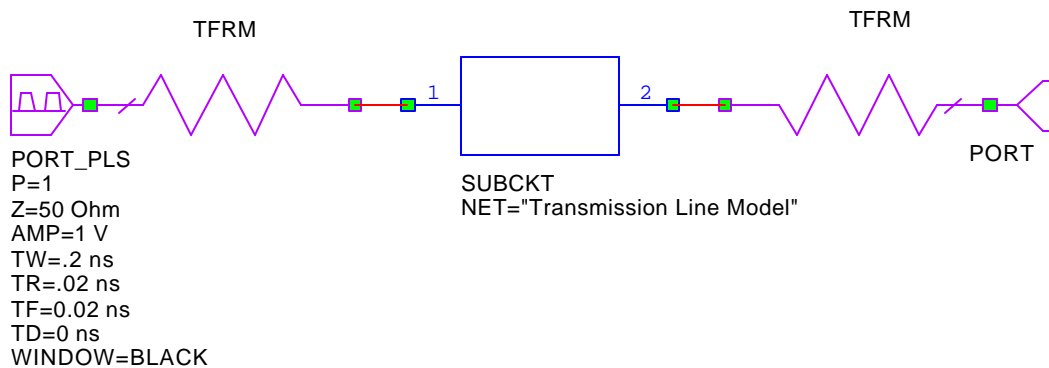


Figure 4 – Analog Office schematic of same network in Figure 3 as it would be modeled. The inclusion of the subcircuit is not necessary in Analog Office because of the iNet capability.

where the subcircuit defined by the “Transmission Line Model” network represents the Analog Office-generated model for the iNet and models the electrical properties of the

routed line. The software will automatically create this model as the line is being routed to a degree of accuracy as specified by the user.

The results are shown in the next Figure 5.

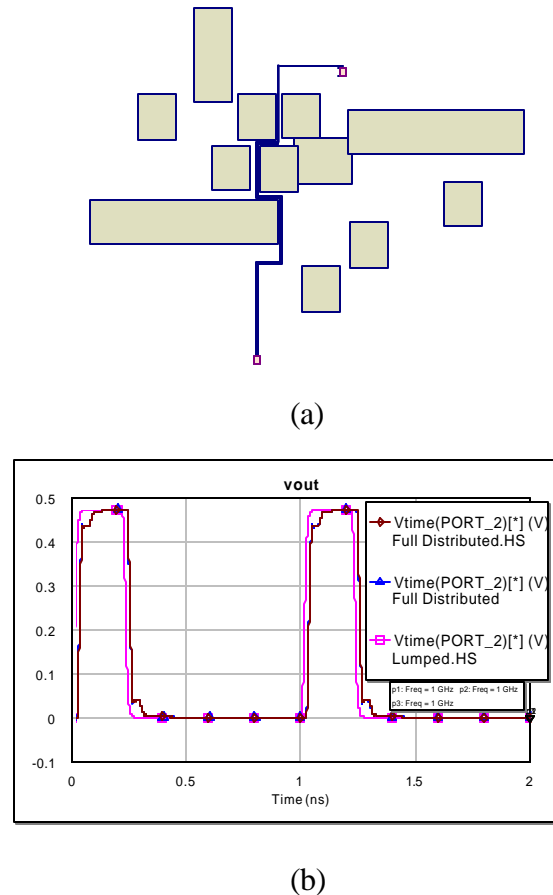


Figure 5 – Analog Office views of two-routed-resistor design: a) layout view, and b) time-domain measurements using HSPICE® and AO HB

The resistors are placed at arbitrary locations relative to some routing obstacles (Figure 5a), such as might be found in a complicated low temperature cofired ceramic (LTCC) or RFIC. The simulation results compare HSPICE and HB as well as two different types of interconnect representation (Figure 5b). The first two traces compare a 1GHz pulse for HSPICE with the accurate interconnect modeling vs. HB at a 1GHz fundamental with 1024 harmonics. The interconnect model generated for HSPICE is based on 512 harmonics of a 1GHz fundamental and includes frequency-dependence of the extracted elements. In comparison, the third trace does not use the accurate interconnect model creation, but instead the Analog Office software has been told to create a simple lumped element model representing the routed line and to simulate the model with HSPICE simulation. This simple model is clearly not complete enough in its representation of the higher frequency effects. In the case of both HSPICE simulations, the time for the HSPICE simulations is roughly the same.

The same modeling technique can be applied to coupled line structures. The results for a coupled line structure 200 $\mu\text{m}$  long, with two 10  $\mu\text{m}$  wide lines separated by 10 $\mu\text{m}$ , is shown in the figure below. The pulse is generated on one of the inputs of one line and the measurement is made at the output of the adjacent line. Excellent agreement is shown between HSPICE and HB (Figure 6).

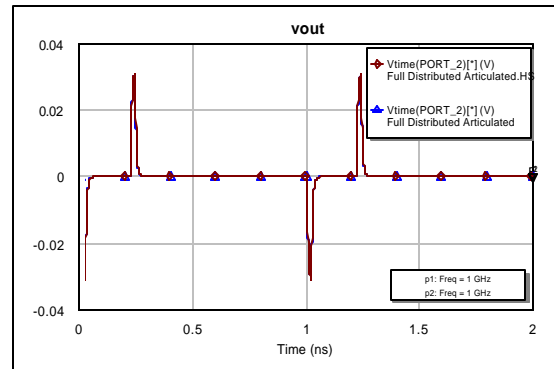


Figure 6 – Coupled line structure ( $L=200 \mu\text{m}$ ,  $W=10 \mu\text{m}$ ,  $S=10 \mu\text{m}$ ) HSPICE vs AO HB

In cases where the physical arrangement of the line or multi-coupled lines can be predetermined, this technique would allow the user to *anticipate* the effect of the interconnect by capturing it as part of the schematic. This is not possible in most cases as the design methodology captures these lines as “nets” and implements them later in the process as routed lines. Hence, design flows have tended to capture the electrical performance of nets as “parasitics,” after the fact.

### Case 2 Theory: Arbitrary Metal

The same notion of pulses being application and manufacturing dependent is true for arbitrary metal as it is for transmission lines. Whereas the transmission line modeling in Case 1, above, can be viewed as closed form models normally defined as schematic circuit elements or routed interconnect, the case with arbitrary metal arrangement here is much more general and applies to any N-port network that can be modeled with an admittance matrix.

Such networks can be decomposed into two subtypes: active and passive. Active N-ports are those that do not exhibit energy creation (or signal regeneration) as represented by their admittance matrix. These networks normally have independent signal sources, transistors, or diodes, contained within them. Passive N-ports are those which only dissipate and distribute signal energy. In some cases, a network containing only metallization, and no sources or active devices, can exhibit “active” characteristics due to measurement errors or inaccuracies of the field solution. While the actual, measured circuit may in fact be totally passive, when this same passive circuit is modeled with an electromagnetic solver (EM), for example, the admittance matrix may in fact contain

entries that imply signal creation at a specific frequency due to the accuracy of the solver or, if “fast-frequency sweeps” are used, errors in interpolation.

Here we limit our consideration to passive networks. These arbitrary N-port structures are modeled by taking their admittance matrix and approximating it by rational functions. These functions are then realized as a network of voltage controlled current sources. A second technique, more applicable to active networks, is also available but not explored here<sup>3</sup>.

Case 2 Example: Arbitrary Metal

In this example, three resistors connected by routed metal are addressed. This can be viewed as a source stage being loaded by two inputs of a second stage—a fan-out of 2. As before, the actual schematic for this simple circuit is straightforward (Figure 7),

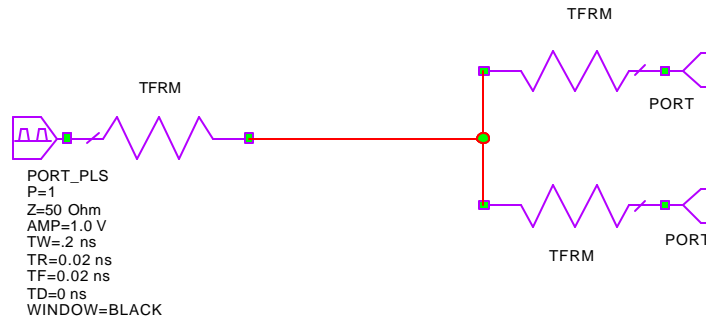


Figure 7 – Analog Office schematic for three resistors interconnected and driven by a 200 psec pulse. The connection among the three resistors in this schematic is a routed interconnect in the layout (see Figure 9a).

but when the iNet connecting the three resistors is actually implemented, the iNet is effectively modeled as a 3-port with a corresponding admittance matrix (Figure 8),

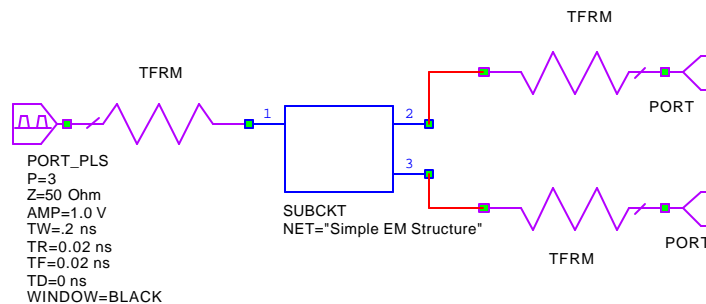
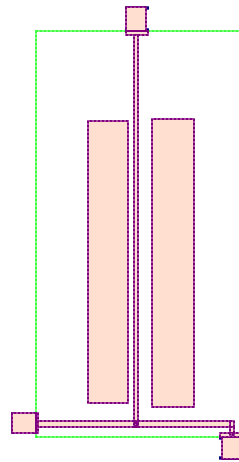
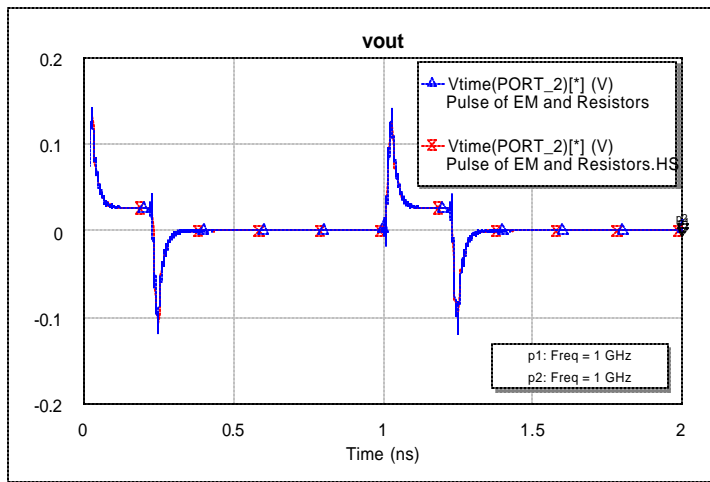


Figure 8 – Analog Office schematic of same network in Figure 7 as it would be modeled. The inclusion of the subcircuit is not necessary in Analog Office because of the iNet capability.

where the iNet 3-port is modeled by the “Simple EM Structure” admittance matrix. The iNet is routed between metallized sections as shown in the layout below (Figure 9a). In comparison, for the example in Case 1 above, the effect of metal in the cells adjacent to the routing channel was ignored, but here it is modeled to first order as ungrounded metal. Although exaggerated here, this sort of arrangement would ordinarily be encountered in densely-packed field programmable gate-array (FPGA), gate-array or analog-mixed signal (AMS) custom application-specific integrated circuit (ASIC) designs.



(a)



(b)

Figure 9 – Analog Office views of three-routed-resistor design: a) layout view, and b) time-domain measurements using HSPICE® and AO HB

The results (Figure 9b) show the waveform measurement at the load resistor located at the top of the layout, based on a drive signal applied to the resistor at the lower right. The source is 1V, 5GHz pulse. The HB and HSPICE simulations are based on the iNet model for the metallization as simulated using Sonnet’s em 3D planar simulator

integrated into the Analog Office software. After the EM results are generated directly from the Analog Office layout, the software creates the model as described above for the HSPICE time-domain simulation while the admittance matrix is used directly in the HB simulation. The agreement between HSPICE and Microwave Office software is excellent.

### Conclusion

The need for accurate time-domain modeling of interconnects and arbitrary metallization is keen in UWB applications, as much because of the design and manufacturing technologies as because of designers' comfort with time-domain simulators. Several examples have been shown that demonstrate the speed and accuracy by which time-domain simulation can take advantage of component modeling techniques normally reserved for frequency-domain designers.

The integration of all the key parts needed to design in this manner—time-domain simulation, frequency-domain modeling, electromagnetic analysis, and circuit layout—in an interactive, concurrent, and ergonomic platform such as the Analog Office design suite, has also been demonstrated. This capability now allows interconnects to be designed or engineered, rather than analyzed or extracted, thereby enabling a totally new design flow that inherently addresses the timing issues so critical to UWB design.

### References

- <sup>1</sup> Edwards, T.C., Foundations for Microstrip Circuit Design, John Wiley & Sons, 1981.
- <sup>2</sup> Choi, K.L., et al, "Characterization of Embedded Passives using Macromodels in LTCC Technology", IEEE Trans. On Components, Packaging, and Manufacturing Technology-Part B", pp. 258-268, August, 1998.
- <sup>3</sup> Wasserman, E., Neilson, D., and Quan, T., submitted to 2004 International Microwave Symposium for publication.
- <sup>4</sup> Heimlich, M.C., et al, "Effect of Substrate Thickness on the Radiation Properties of Coplanar Strip Antennas," Electronics Letters March 1992.