ARMMS
RF & Microwave Society

Transient Simulations of RF and High Data Rate circuits

Malcolm Edwards and Evgeny Wasserman
Email: malcolm@appwave.com
Applied Wave Research UK (Europe), 2 Huntingate, Hitchin, Herts, UK. SG4 0TJ

Abstract

This paper provides an overview of the challenges facing the simulation technology used in modern EDA tools that are employed in the research and design of RF, microwave and high data rate circuits. With the operating frequency of consumer products moving well into the microwave region, attention has to be paid to the way in which dispersive elements, data defined models and non-linear models are captured and represented in both steady state solvers and transient solvers.

Steady state simulation and transient simulation data each provide an insight into circuit operation. Both are required to provide the designer with the information needed to ensure that the circuit is compliant to specification. What were once subtle effects in circuits driven at low frequency or low slew rate, with an increasing operating frequency now become more profound, and in some instances serious signal integrity issues. A design environment that combines both classes of solver becomes a necessity.

Introduction – The high frequency design problem

When analyzing the behavior of electronic circuits two major classes of simulator are available to the high frequency engineer, Harmonic Balance and Transient Time Domain. When the steady state behavior of a circuit is being investigated, the popular and natural choice is the Harmonic Balance (HB) solver. This class of solver has been popular with microwave engineers for some considerable time, and is now becoming used more frequently by RF engineers who in the past have been more familiar with transient solvers. If on the other hand, the transient nature of a circuit needs to be investigated, for example, oscillators in their start up regime, chaotic pulse generators and frequency dividers, then the Transient Time Domain (TTD) simulator is the preferred solver. Each simulator provides an insight into these two states of the circuit under investigation. Strictly speaking, the TTD solver will yield the same information as the HS solver if the end time of the TTD solution is set to be well beyond the transient period of the circuit. The HB solver of course also provides time domain data, but only waveforms (current and voltage) in the steady state operation of the circuit which are constructed from the data set of sine waves.

This paper will focus primarily on TTD simulators, and for brevity these will be referred to simply as transient simulators. A typical mix of circuit elements that need to be modeled are illustrated in Figure 1. This figure shows a section of a GaAs HBT high speed amplifier. Naturally many other designs could have equally been chosen to illustrate this mix of circuit elements. The circuit elements that will be discussed are non-linear devices, data defined elements (data supplied from a VNA or EM solver), transmission lines and their associated discontinuity models and lastly simply modeled passives, such as lumped capacitors. Several circuit simulators based on the original Berkley Spice simulator include up to date model libraries for nonlinear devices, such as field effect transistors (FET), bipolar transistors and diodes. The topic of model verification, i.e. the comparison between non-linear models found in various simulators is beyond the scope of this paper. Regardless of the quality of the nonlinear models, many of these simulators lack accurate and effective modeling solutions for any distributed components that exhibit dispersion and the associated discontinuity models. They also handle data defined elements in an in-efficient manner. Distributed components are often employed in the design of RF and Microwave circuits and with the frequency of operation increasing, their use is becoming more common. As mentioned, another simulation requirement common in the microwave field is the need for the simulator to handle data sets that are derived either from VNA measurements or EM analysis. This data has to be handled in an effective manner, by which we mean; accurate and efficient in computation resources (CPU clock and RAM). Ensuring that the translation of these frequency domain descriptions into a form that is appropriate for the transient solver, is a subject of research within the simulator community. This article describes successful approaches to distributed device modeling and transient simulation and efficient computation of data defined elements. The result is accurate transient simulations for circuits containing coupled transmission lines, discontinuities and components characterized by frequency-dependent multi-port parameters obtained from numerical EM analysis or measurement.

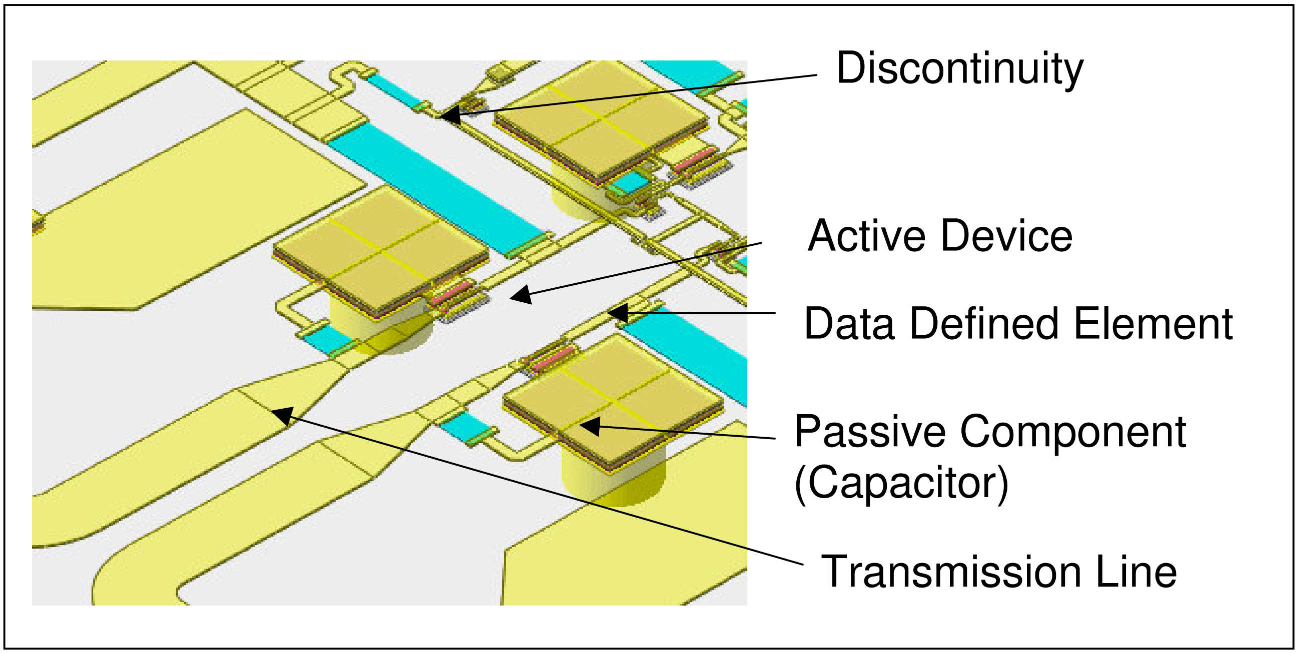

Figure 1: Typical Circuit Elements

Transient simulation of distributed elements and discontinuities

The interconnect and distributed matching circuits of complex high frequency circuits are often implemented by using one or more uniform transmission lines. The circuit representation (Figure 2: schematic with transmission line elements) of this arrangement is composed of a set of uniform transmission lines sections connected to one another with discontinuity models placed at appropriate junctions. The discontinuity models account for energy storage at the junctions of two or more transmission lines. In the AWR Design Environment, (AWR DE®) the EM derived discontinuity models are inherently causal, that is, they are constructed from circuits that only contain positive valued parameters. For example, if a discontinuity model needs an inductive element to capture the evanescent H fields, then the equivalent L parameter will be positive. This is not always the case for the published frequency domain analytical models; in many cases they have negative values for the inductance which leads to instability in the transient solver. As the

EM based circuit models are causal no special attention needs to be paid to their use in a transient solver.

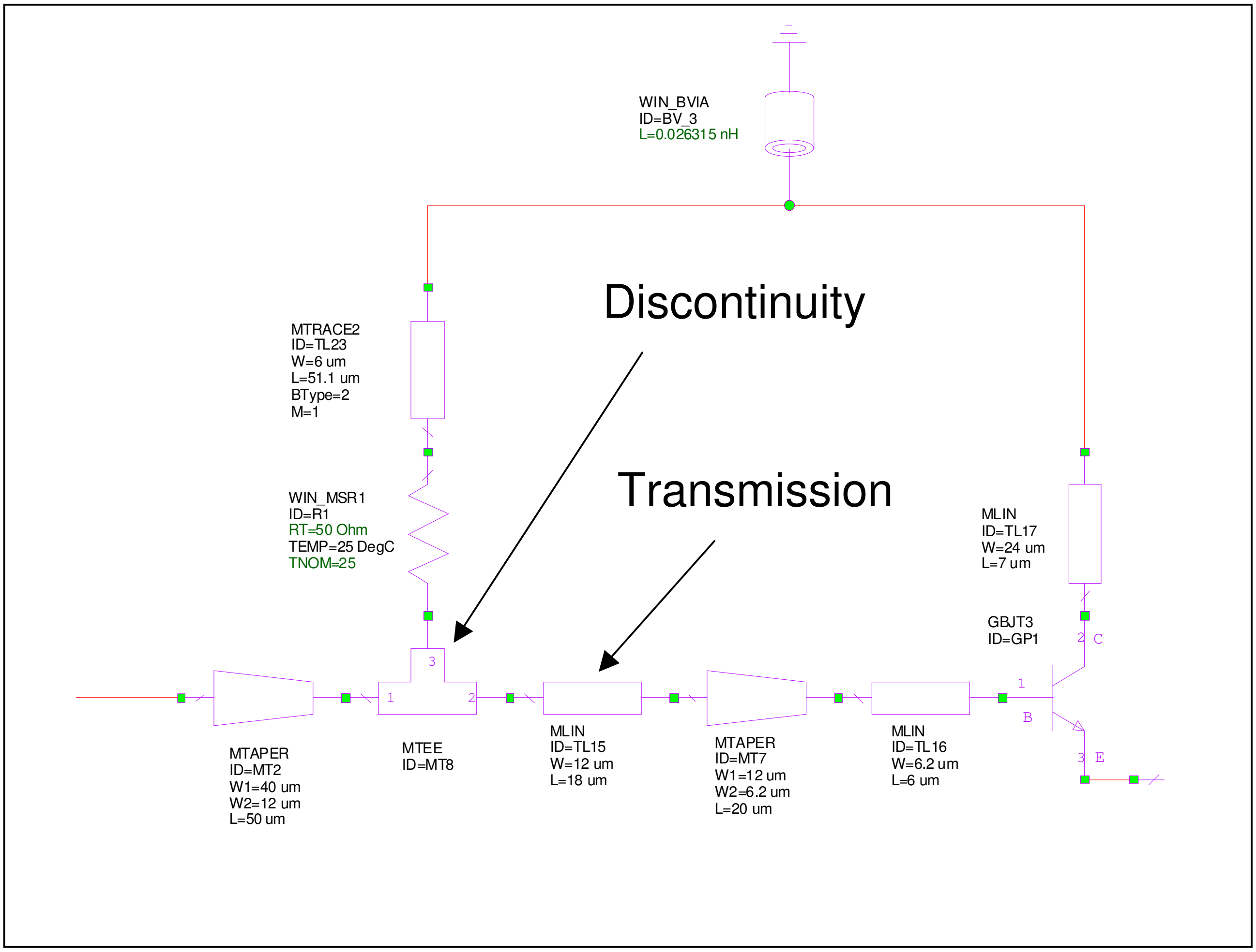

Figure 2: Schematic with Transmission Line Elements

Distributed elements realized in inhomogeneous structures inherently exhibit dispersion. Here the characteristic impedance and propagation velocity are frequency dependent. Both microstrip and CPW are good examples of such transmission structures that display significant dispersive impedance and propagation properties. Both these transmission line systems have been researched for some considerable time resulting in a body of excellent frequency domain models. When any of aforementioned transmission line systems operate over a significant bandwidth the dispersive effects become difficult to approximate using the basic transmission line models composed of RLGC elements suitable for transient simulation. The built in transmission line models in most transient simulators are described by velocity of propagation and characteristic impedance, properties which are excellent for coaxial systems but poor for the classes of transmission lines being considered here. Therefore, as the operating frequency increases, the importance of an accurate model becomes apparent.

With one modern EDA tool built for the design of RFIC and high speed analogue circuits an alternative interconnect paradigm is offered. In place of using explicit transmission lines and discontinuity models, the AWR DE® has an advanced interconnect router (iNET ®). This intelligent router builds a comprehensive interconnect model on the fly. It allows the designer to connect up the building blocks of a design as he would by using a conventional router, and at the same time minimize the number of elements seen on the schematic. The design environment offers one more degree of freedom; if the interconnects are considered to be close together in certain semi-critcal regions of the layout, the internal model can be substituted using an EM extraction tool such as NetAN® (OEA). The term semi-critical has been employed to suggest that critical interconnection between elements are best modeled using explicit transmission lines and discontinuities. This design technology is focused on dense Si designs such as RFICs for RF and microwave applications. EM extraction tools are beyond the scope of this paper; suffice to say, these solvers return an equivalent RLGCK model; here, K represents the magnetic coupling between tracks.

Transmission lines are known to satisfy the telegrapher's equations, which (in the time domain) are defined by …

$$\frac{\partial V(x,t)}{\partial x} = -\left(R + L\frac{\partial}{\partial t}\right)I(x,t)$$

$$\frac{\partial I(x,t)}{\partial x} = -\left(G + C\frac{\partial}{\partial t}\right)V(x,t)$$

Transient simulation with constant values for RLGC is a well established technique. However, with the effects of dispersion, skin effect and dielectric losses all contributing to a significant frequency dependency in the RLGC parameters (plus the K term if the model is generated by an extraction process) the transient solution becomes more complex. At present, transient simulation techniques for handling these frequency-dependent RLGC values are actively being researched. As expected, using these values for frequency-domain simulations is a relatively straightforward process, and well documented in the literature [1]. A unique advantage of the HSPICE® circuit simulator is its implementation of an advanced technique for handling frequency-dependent RLGCK matrices for coupled transmission line systems. This solver technology is known as the W-element. HSPICE® uses recursive convolution methods [2, 3] to accelerate the time-domain simulations, while still making accurate use of the frequency-domain RLGC data. The W-element is very often used for high speed digital PCB SI (Signal Integrity) applications, typically in the range of 1 to 6 GHz.

From a computation standpoint the W-element algorithm has no hard frequency cutoff point. Its frequency range of applicability is determined by the quality of the frequency-dependent RLGC parameters provided. During simulation, frequency-dependent RLGC data is computed for each transmission line structure on the fly, and the appropriate W-element models are then packaged ready for the HSPICE® transient simulation. In other transient simulation tools, transmission lines are often modeled with simplified low frequency approximations, such as an assumption of constant, frequency-independent, RLGC parameters, or by completely neglecting losses. Which ever reduced model is selected, the result is that the true nature of the element is lost. As it has been pointed out before, microwave frequency simulation demands that dispersion, skin effect and other frequency-dependent losses must be modeled accurately to obtain useful results. In the

HSPICE® simulator, the W-element approach is based on decomposing the coupled transmission line system into several well-behaved transfer functions based on the system's characteristic admittances and complex propagation factors. The decomposition captures the frequency dependencies accurately in the time domain by using a rational function approximation which in turn is analyzed using a recursive convolution approach.

Other available techniques lack the following distinct advantages:

• Computational performance is superior to that of a direct numerical convolution. A linear vs. quadratic scaling of the CPU time is seen with the length of transient simulations. After the initialization stage, a W-element compares well with a simple circuit element such as a resistor.

• The manner in which the frequency dependence of RLGC parameters can be described is more versatile; these can be user-specified (in tabular format), or extrapolated, based on standard skin effect and loss-tangent parameters.

• There is no limit to the number of signal conductors in coupled transmission line systems. The modal decomposition for multi-conductor lines is performed automatically.

As with any modeling approach, some care must be taken when using the W-element to avoid any possible undesirable effects:

• Extremely short and extremely long transmission lines should typically be avoided for transient simulations. By their nature, extremely short transmission lines are more effectively model as lumped elements. In RFICs, extremely long transmission lines don't exist!

• To ensure causality in the time domain simulation, the interdependencies of frequency-domain RLGC data should be preserved (for example, the relationship between R (f) and L (f) values due to skin-effect).

• For broadband simulations, i.e. a large number of octaves, the correct asymptotic behavior of the RLGC data at high frequencies is needed for causality and accuracy.

With refined generation of the RLGC data, and by paying attention to the cautions listed above, the W-element provides excellent accuracy. In the AWR DE, the RLGC model generation is performed automatically, resulting in transient simulations with voltages errors that never exceed 0.1 per cent when multiple W-elements are present. As mentioned before, the need for transient simulation is most often necessary for circuits containing nonlinear devices (FETs, BJTs and diodes); nevertheless the accuracy of this approach is more clearly demonstrated with an example of a distributed linear system containing coupled line sections.

The suitable test case is illustrated in Figure 3, a band-pass filter based on edge-coupled transmission lines. The filter model includes end-effects (open-circuit) and step (line-width) discontinuity effects. The AWR DE® includes sophisticated frequency-domain models for each element within the filter based on geometric and substrate parameters. These models include dispersion and frequency-dependent loss effects.

The MSUB element seen in the schematic defines the cross sectional geometry of the microstrip substrate. Note also that equations have been used to define the widths and lengths of the coupled line sections. The use of equations helps to speed up the optimization of this structure by linking the symmetric elements together; in this example the two outer coupled line sections share the same parameters.

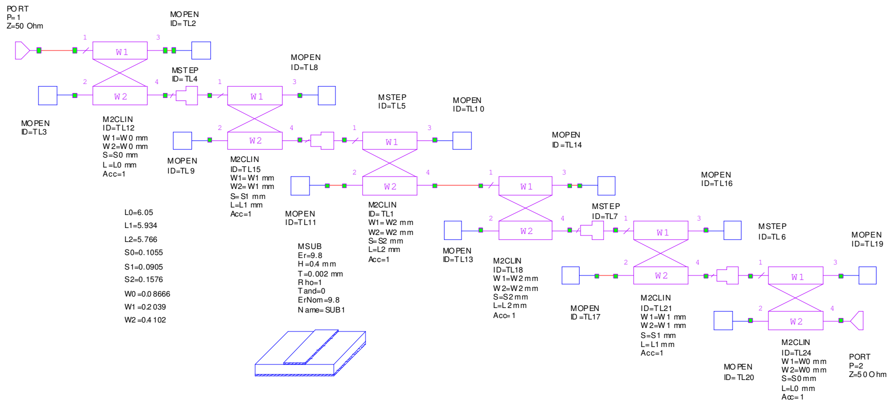

Figure 3: Band-pass Filter

Using the built in models of the coupled line sections, an equivalent HSPICE® W-element model is generated as a frequency-dependent RLGC data set. The quality of the AWR DE-to-HSPICE translation can be examined in the frequency-domain using scattering parameters. Figures 4 and 5 shows frequency domain data derived directly from the schematic along with scattering parameters generated by steady state linear simulation using HSPICE®. We see both input reflection coefficient and insertion loss simulation data for the complete filter. Both the dB plots and Smith chart loci show excellent agreement.

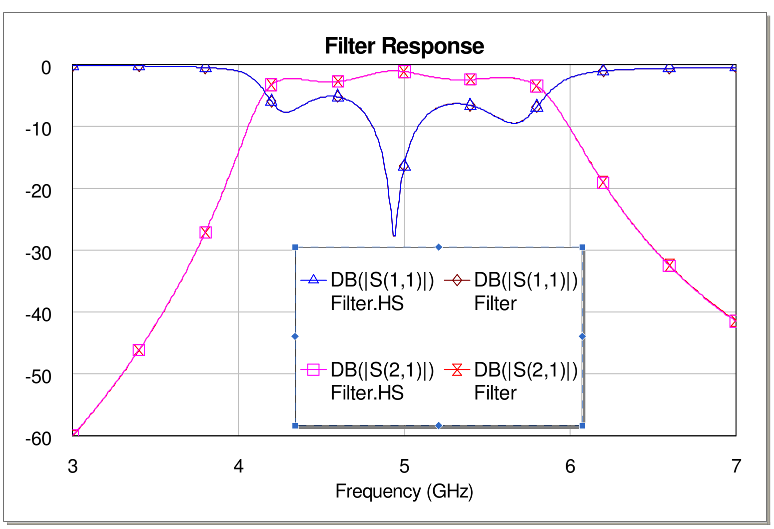

Figure 4: Band-pass S21 data from internal AWR DE® simulation and HSPICE® simulation.

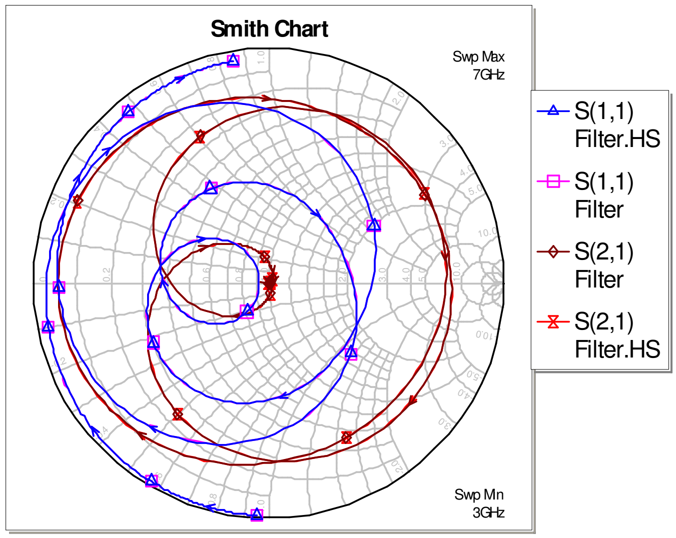

Figure 5: Band-pass filter S11 data from internal AWR DE® simulation and HSPICE® simulation

The last two graphs show excellent agreement when comparing the linear steady state results from two simulators, which suggests that there is a negligible loss of information when moving from an analytical model to a W-Model.

By using Fourier analysis methods it is possible to assess the quality of the translation when the W-Model is used in transient simulations. The AWR DE® can perform an inbuilt Fourier analysis after the HB simulator has solved the circuit equations. From a users' perspective, adding a Vtime measurement to a graph will display the voltage of any circuit node over time. This measurement request will have automatically invoked a Fourier analysis of the HB frequency domain data set. By its' very nature, the transient simulator will inherently follow the time domain waveform behavior of the circuit. In order to evaluate the accuracy of the distributed element model translation into the W-Model, these two data sets, direct time domain data and indirect time domain data gathered from an HB simulation are presented on the same graph, figure 6. A test circuit was constructed by connecting a pulse voltage source at port 1 (amplitude of 5 V), a rise/fall time of 0.1 ns and a pulse width of 2 ns. The filter is loaded at port 2 with a 100 ohm resistor. Since the test circuit is linear, Fourier analysis can be used to provide the reference solution. A large fundamental period (10 ns, i.e. a drive frequency of 100 MHz) and a large number of harmonics (4096) are used to ensure Fourier series convergence for all practical intents and purposes.

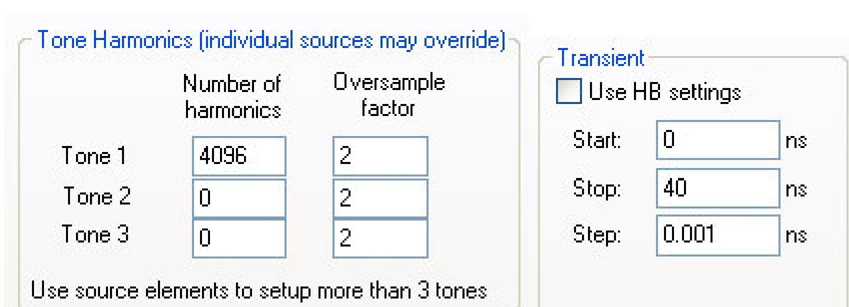

Figure 7: Harmonic Balance and Transient Time Domain Simulation Settings.

Furthermore, the transient simulations were run for 40 ns (4 periods of the fundamental frequency), and only the last period was shown for comparison. The simulator settings are show in figure 7. The pulse response comparison between the AWR DE® and HSPICE® models is shown in Figure 8.

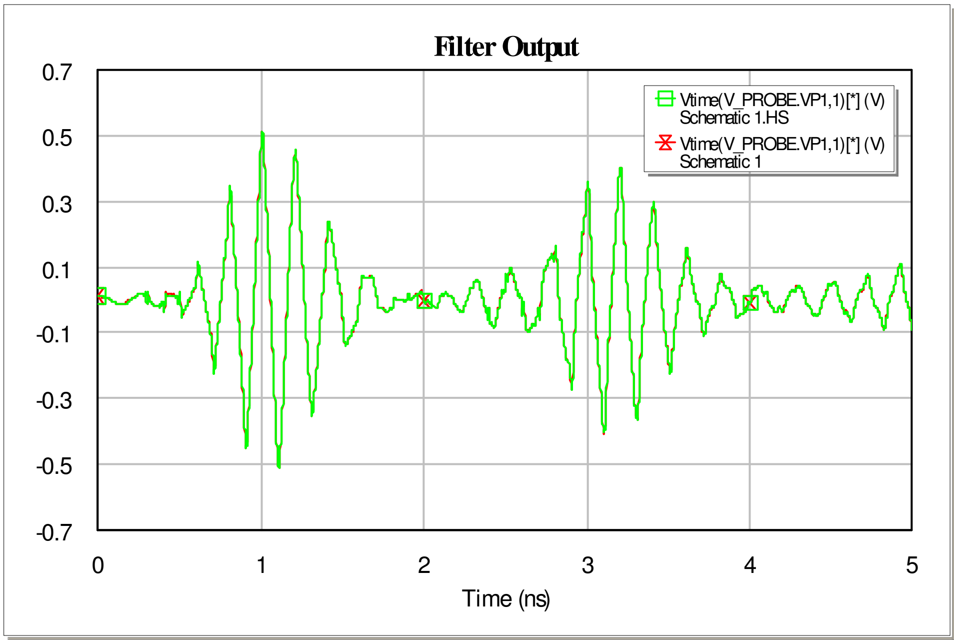

Figure 8: Comparison between the Fourier Transformed HB results and HSPICE.

As before, we see that there is excellent agreement between the two methods; this has validated the approach adopted provides a way to realize accurate transient, time-domain simulations by translating frequency-domain models into HSPICE® W-elements.

Transient simulation with models obtained from VNA measurements and EM analysis

VNA measurements and EM analysis are two popular sources of s-parameters. Many component manufacturers supply s-parameters in place of complex circuit level models to characterize their devices. During their development phase, many passive structures are designed using EM solvers. The data provided by these solvers is also S-Parameters. Regardless of the source of S-Parameters, two techniques are available to the designer to use this data in transient simulations. The first, a technique that is often used in package design and signal integrity investigations is to use the data to construct a lumped element circuit equivalent. These models do not typically achieve the required accuracy (–30 to –20 dB) over a wide frequency range. With the second technique, two methods are provided to incorporate the frequency-dependent S-Parameters into a transient simulation:

1. Rational function approximation of the N-port admittance matrices.
2. Direct numerical convolution.

Method 1 can be used only for passive devices and has been shown to be highly accurate and computationally efficient. Passivity means that the device does not generate power, for example, spiral inductors, couplers, filters, transmission line discontinuities, etc. The rational function approximation is performed automatically within the AWR DE® software. The method employed is a combination of those described in various references, including the vector fitting (VECTFIT) method by Bjorn Gustavsen and Adam Semlyen. [5, 6] The computed rational functions are translated into HSPICE voltage-controlled current sources (VCCS) with Laplace transfer functions. The MNA stamps for these sources are calculated efficiently using a state-variable formulation, and no numerical convolution is necessary. The second method is more general, and it caters for active devices. Tests suggest that the accuracy for distributed models is lower and the CPU time scale quadratically with the length of the simulated transient. Method 2 is also more tolerant of the class of data in that describes non-causal elements, while causality is strictly enforced with Method 1. Transient simulation with non-causal models is to be avoided if possible, since the results may not be physically meaningful.

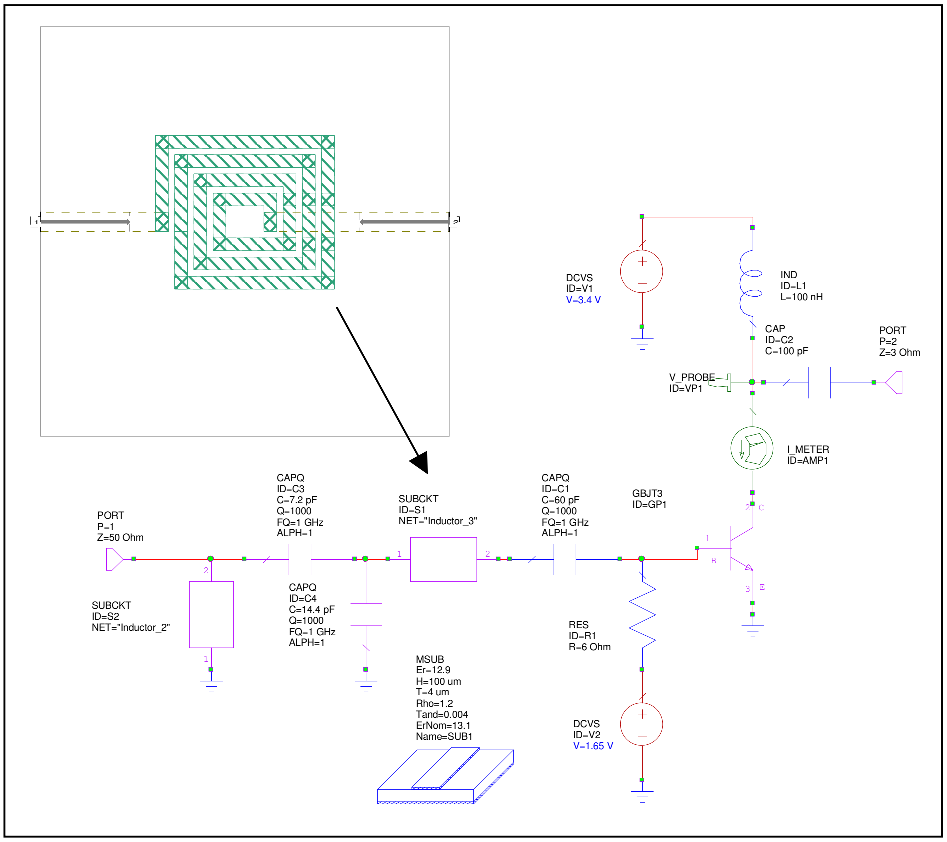

Figure 9: HBT amplifier with matching inductor modeled using EM derived S-Parameter Data

An example using EM-derived data is the heterojunction bipolar transistor (HBT) amplifier with spiral inductors shown in Figure 9. The scattering parameters for the spiral inductors were derived automatically from EM analysis within the AWR DE, and rational function approximations were used to translate the EM analysis results into HSPICE models for transient simulation. A Gummel-Poon BJT model was used for the HBT device. The transient simulation results for the HBT amplifier example are shown in Figure 10. Since large capacitors (100 pF) and inductors (100 nH) were present in the circuit, it takes on the order of 160 ns for the transient processes to finish. The rational approximation is constructed during an initialization stage that takes approximately one second of CPU time.

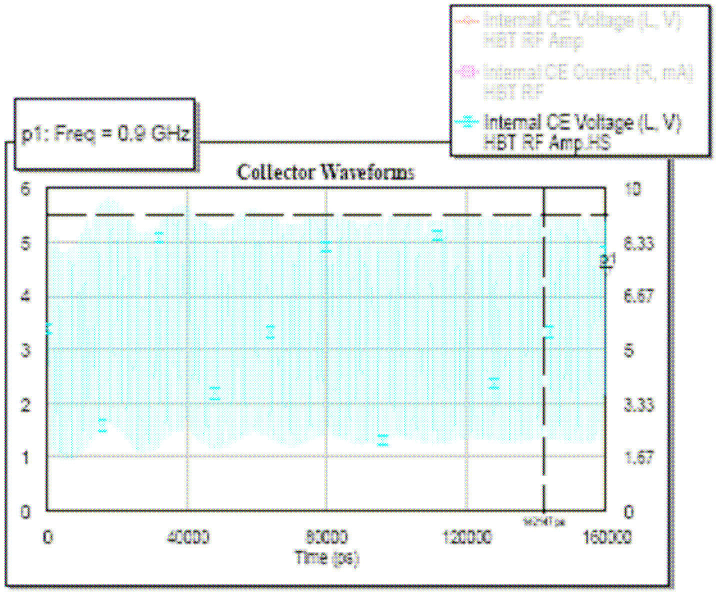

Figure 10: HBT amplifier simulation results

In order to verify that the transient simulation produces results consistent with the frequency-domain data, the transient waveforms at steady state can be compared with those computed from harmonic balance analysis. In Figure 11, the last two periods of the transient waveform are plotted along with the steady-state waveform derived from HB analysis.

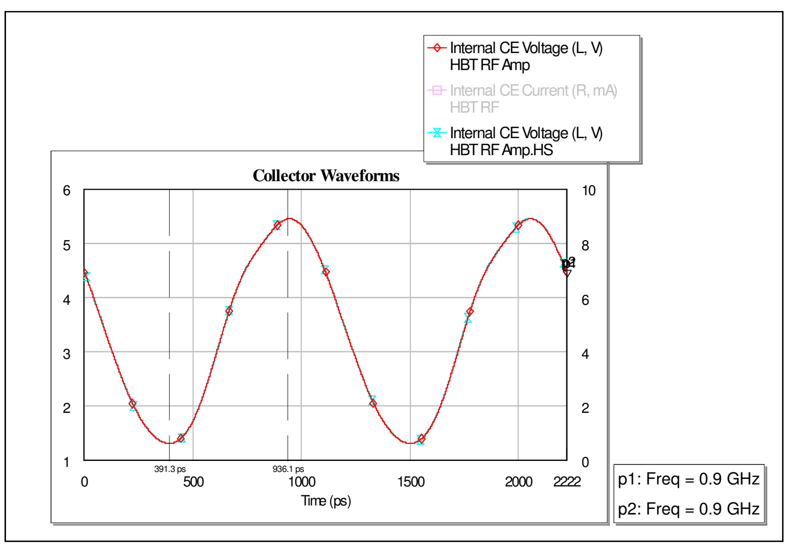

Figure 11: Transient HSPICE® simulation data in the steady state region of the circuit compared to the harmonic balance result.

The plots demonstrate excellent agreement between the HSPICE® transient and harmonic balance simulations.

Conclusion

This article presents the optimum techniques for translating model data (analytic equations, measured and computed data, extracted interconnect data) into a form that offers efficient transient time domain simulation. A brief overview of a unique approach to transient simulations of microwave and RF circuits using the AWR Design Environment® coupled with the high performance transient time domain solver HSPICE®, from Synopsys has been given. The methodology is based on decomposing circuits into components that are either uniform transmission lines, or elements of reasonably small electrical length. Accurate modeling of distributed components such as interconnects, transmission lines, transmission line discontinuities (such as T-junctions, crosses, etc.) and microwave devices (such as spiral inductors and baluns) is essential if accurate simulation results for the complete circuit are to be obtained. Accurate transient simulation of transmission lines is obtained by using the HSPICE® W-element with frequency-dependent RLGC matrices that are automatically generated from the internal frequency-domain component models in AWR DE® software. Also demonstrated was an accurate and efficient approach to transient simulation using models for passive components based on rational function approximations of frequency-dependent N-port parameters.

References

1. C.R. Paul, *Analysis of Multi-conductorTransmission Lines*, John Wiley & Sons Inc., New York, NY, 1994.
2. D.B. Kuznetsov and J.E. Schutt-Aine, “Optimal Transient Simulations of Transmission Lines,” *IEEE Transactions on Circuits and Systems*, Vol. 43, No. 2, February 1996, pp. 110–121.
3. K.S. (Dan) Oh, “Accurate Transient Simulation of Transmission Lines with the Skin Effect,” *IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems*, Vol. 19, No. 3, March 2000, pp. 389–396.
4. S.G. Grivet-Talocia and F.G. Canovero, “TOPLine: A Delay-dole-residue Method for the Simulation of a Lossy and Dispersive Interconnect,” *IEEE 11th Topical Meeting on Electrical Performance of Electronic Packaging Digest (EPEP)*, Monterey, CA, October 21-23, 2002, pp. 359–362.
5. B. Gustavsen, “Computer Code for Rational Approximation of Frequency-dependent Admittance Matrices,” *IEEE Transactions on Power Delivery*, Vol. 17, No. 3, 2002, pp. 1093–1098.
6. B. Gustavsen and A. Semlyen, “Rational Approximation of Frequency Responses by Vector Fitting,” *IEEE Transactions on Power Delivery*, Vol. 14, No. 3, 1999.