

Generation of rational model based SPICE circuits for transient simulations

Nobby Stevens* and Tom Dhaene**

*Agilent Technologies Belgium, EEs of EDA, Kortrijksesteenweg 1093b, 9051 Gent, Belgium

**Ghent University – IBBT, Gaston Crommenlaan 8 bus 201, 9050 Gent, Belgium

nobby_stevens@agilent.com

Tom.Dhaene@UGent.be

Abstract

In this paper, we present an efficient and compact circuit netlist formulation for rational models. It is shown how to implement the method for several commercially available time domain solvers. Finally, an example is given describing the flow and procedures to follow in order to obtain a reliable and stable transient simulation.

Introduction

Designing a circuit with analog, digital and/or active components requires a rigorous analysis in the time domain. Especially with regard to signal integrity issues such as crosstalk and signal degradation, it is crucial to have a good quantitative knowledge of these effects during the design phase. As of today, a number of SPICE simulators are commonly used for transient simulations. Note however that most electromagnetic simulators and measurement tools perform in the frequency domain.

The conversion from frequency domain to time domain is, from a numerical perspective, a delicate procedure. Here, rational macromodeling comes into play. Once a passive mathematical rational model of the frequency data is constructed, a last and important step to take is the realization of a compact SPICE-like circuit using lumped components, that can be efficiently processed by a time-domain circuit simulator (such as e.g., Hspice and Spectre).

Equivalent SPICE realization

Consider the scattering matrix S of a Linear Time-Invariant system (LTI), with b and a respectively the column vectors of the reflected and incident waves:

$$b = Sa \quad (1)$$

Suppose that for this system, a broadband passive rational model has been constructed by use of the Vector Fitting technique [1-3] or the direct interpolation scheme [4]. For each element S_{kl} of the scattering matrix, one can write:

$$S_{kl} = d_{kl} + \sum_{m=1}^{M_{kl}} \frac{c_m^{kl}}{s - p_m^{kl}} \quad (2)$$

where, the c_m^{kl} are the residues and p_m^{kl} are the corresponding poles. They both are real, or occur in complex conjugate pairs. Note that the order M_{kl} may depend on the dynamic behavior of each individual element. Also a set of common poles can be used, or each element can have its own set.

We obtain the following relationship between the reflected and incident wave of an N -port system:

$$b_k = \sum_{n=1}^N \left(d_{kn} + \sum_{m=1}^{M_{kn}} \frac{c_m^{kn}}{s - p_m^{kn}} \right) a_n \quad (3)$$

Realization of vi/ab transformation

SPICE circuit simulators work with port voltages and currents. The relationship between the reflected wave b_k and incident wave a_k and the corresponding port voltage v_k and port current i_k is given by (Z_0 is the reference impedance):

$$a_k = \frac{v_k + Z_0 i_k}{2\sqrt{Z_0}} \quad (4)$$

$$b_k = \frac{v_k - Z_0 i_k}{2\sqrt{Z_0}} \quad (5)$$

Using these relations, we can see the incident wave as a (scaled) virtual voltage and the reflected wave as a scaled virtual current:

$$v_k = Z_0 i_k + 2\sqrt{Z_0} b_k \quad (6)$$

$$a_k = \frac{v_k}{2\sqrt{Z_0}} + \frac{\sqrt{Z_0}}{2} i_k \quad (7)$$

From a network point of view, this can be represented as in Figure 1. (CCVS stands for current controlled voltage source and VCVS for voltage controlled voltage source.) This way, no negative resistors are introduced as in the work by Lamecki [5] and Neumayer [6]. Some SPICE simulators may have difficulties with negative resistor values.

Realization of rational transfer function

At this point, equation (3), which describes the relation between incident and reflected waves, must be reformulated as a circuit equivalent network. The form of the rational representation (2) is particularly interesting for transient simulations since the inverse Laplace transform of each term is known. Recent versions of commercially available SPICE-like circuit simulators make use of this fact and actually support the rational presentation as defined in equation (2).

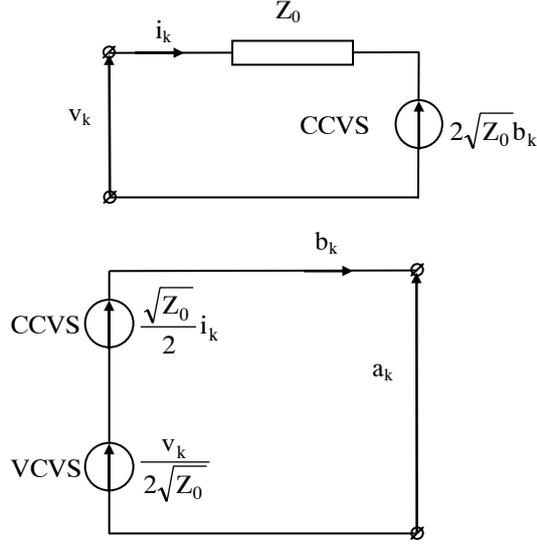


Fig. 1: From port voltage v_k and current i_k to incident wave a_k and reflected wave b_k .

For ease of representation, let us focus now on a simple two port system with only one common real pole p :

$$\begin{aligned}
 S_{11} &= d_{11} + \frac{c^{11}}{s-p} = \frac{b_1}{a_1} \Big|_{a_2=0} \\
 S_{12} = S_{21} &= d_{12} + \frac{c^{12}}{s-p} = \frac{b_2}{a_1} \Big|_{a_2=0} = \frac{b_1}{a_2} \Big|_{a_1=0} \\
 S_{22} &= d_{22} + \frac{c^{22}}{s-p} = \frac{b_2}{a_2} \Big|_{a_1=0}
 \end{aligned} \quad (8)$$

With regard to the reflected waves (or the virtual currents in the circuits as defined above), we have:

$$\begin{aligned}
 b_1 &= \left(d_{11} + \frac{c^{11}}{s-p} \right) a_1 + \left(d_{12} + \frac{c^{12}}{s-p} \right) a_2 \\
 b_2 &= \left(d_{12} + \frac{c^{12}}{s-p} \right) a_1 + \left(d_{22} + \frac{c^{22}}{s-p} \right) a_2
 \end{aligned} \quad (9)$$

The circuit can easily be realized by using eight voltage controlled current sources (VCCS) with a transconductance G . Defining VCCS elements requires the specification of four nodes: two controlling nodes for the input voltage and two nodes of the controlled element itself. For the subcircuit containing b_1 , it is clear that for the two first sources on the left hand, the controlling and controlled nodes are both a_1^+ and a_1^- . For the two other sources, the controlling nodes are a_2^+ and a_2^- , as can be seen in equations (9). The same reasoning holds for the subcircuit defining b_2 . It is obvious that this approach is not restricted to 2-port systems or to real poles. It can easily be generalized for multiport systems and for complex conjugate pole pairs.

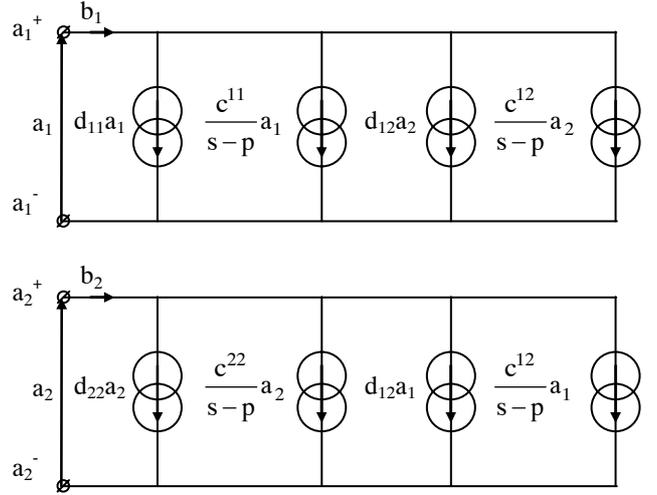


Figure 2: Circuit implementation of equation (8).

Recent versions of most commercial SPICE simulators support the rational representation as defined in (2). In HSPICE, the form is defined by using the keyword "FOSTER". In SPECTRE, the keyword to use is svccs, the "s" stands for s-controlled VCCS. In GEMINI (netlist language within Agilent's Advanced Design System (ADS)), the keyword to use is VCCS_POLE. More details can be found in the corresponding manuals.

Apart from the faster time domain simulation, another important advantage is embedded within the procedure as described above. The form is compact compared to the netlist definition as described by Lamecki [5] or Neumayer [6]. By direct implementation of the rational form, we avoid the introduction of the internal state variable x of the state-space model (for more details, see [5,6]). This leads to more compact netlists, which also takes less process time for the netlist parser.

Example

Let us consider two coupled PCB lines with a width of 252 μm and separated over the same distance. The total length of the lines is 16.85 cm. They are located on a bottom closed FR4 substrate with a thickness of 1.4986 mm (= 59 mil) with a dielectric constant of 4.5 and a loss tangent of 0.02. For these kind of long lines, crosstalk between the lines becomes an important issue. We have attached ports to each end of both lines, which results in a 4-port system. The frequency data was obtained using an adaptive frequency sweep with ADS Momentum. The next step is the creation of a broadband rational model (as in equation (2)). This is done using the Broadband SPICE Model generation tool of ADS. A fit with 25 global poles was executed with a worst case RMS error of -59 dB.

In order to obtain accurate and physical results during a transient simulation, it is also very important that the generated model is passive. Several techniques exist to enforce passivity ([5,7-9]). The necessary and sufficient condition for a rational macromodel to be passive is that the Hamiltonian of the system contains no imaginary eigenvalues ([7]). Locations

between these imaginary eigenvalues may correspond to non-passive frequency ranges (i.e. maximum singular value of the scattering matrix larger than 1.0). On Figures 3 and 4, you can respectively see the location of the eigenvalues of the Hamiltonian and the maximum singular values over the frequency domain for the example under study. Investigation of these eigenvalues indicates that the original rational fitting model is non-passive between 2.069 GHz and 2.157 GHz. This is confirmed by the plot of the maximum singular values. The largest violation is $3.05 \cdot 10^{-2}$ at 2.12 GHz.

Within the Agilent Broadband SPICE Generator tool, we make use of an advanced and patent pending technology to enforce passivity. On Figures 5 and 6, you can see respectively the perturbation of the critical complex eigenvalues of the Hamiltonian and the maximum singular value of the model after the passivity enforcement step. The worst case RMS error after passifying the model was -57 dB. Considering the magnitude of the passivity violation ($>10^{-2}$), this is an excellent result.

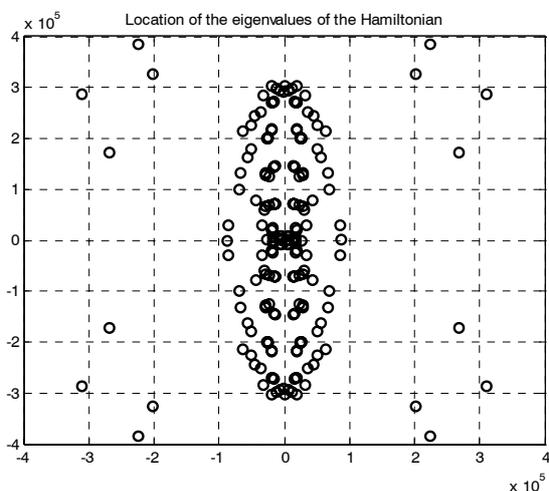


Figure 3: Eigenvalues of the Hamiltonian.

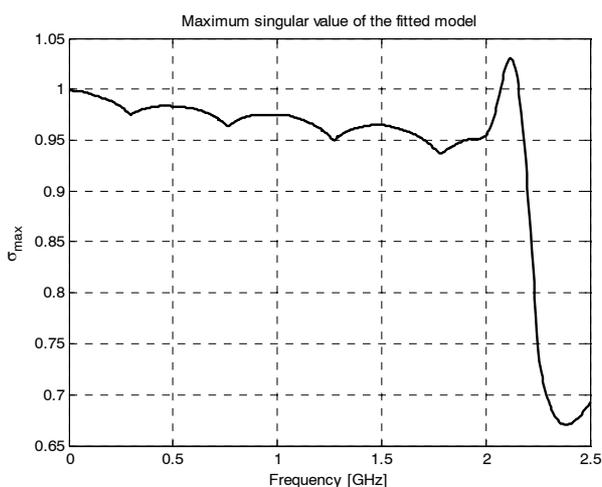


Figure 4: Maximum singular value of the fitted model.

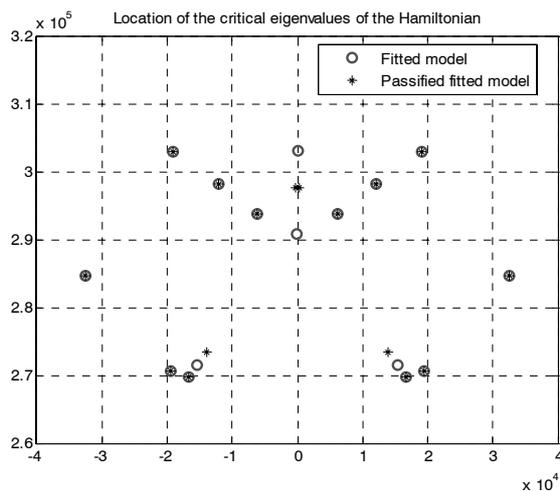


Figure 5: Perturbation of the critical Hamiltonian eigenvalues.

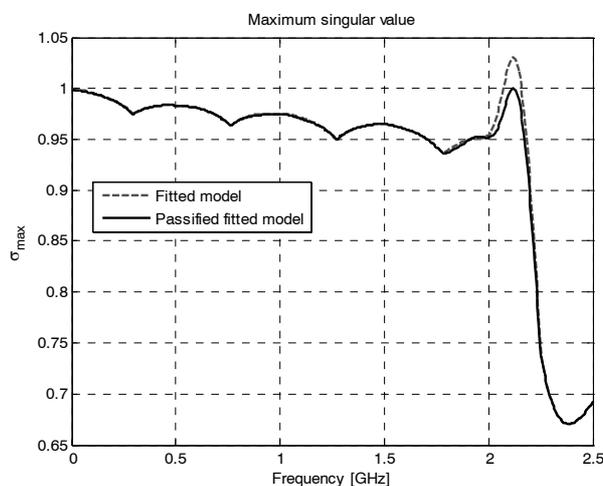


Figure 6: Maximum singular values of the fitted and passified model.

Finally, we are ready to construct the circuit for transient simulations, based on the procedure as described in the beginning of this paper. In order to quantify the near end (NEXT) and far end (FEXT) crosstalk, consider the following setup:

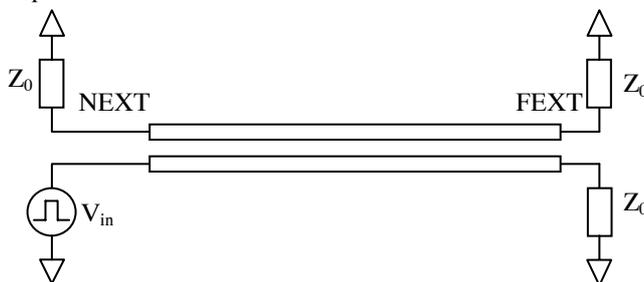


Figure 7: Setup for crosstalk characterization.

V_{in} is a voltage pulse generator with a pulse width of 1.2 ns and a linear rise and fall edge of 0.4 ns. The period of the signal is 4 ns. Based on the calculated passive rational model, we have constructed two netlists under ADS, once with the inclusion of the state as defined by Lamecki ([5]) and once as defined in this paper. The execution time of the transient

simulation with the new format took only 52 % compared to the format including the implementation of the state circuit. This kind of simulation time improvements (about half) has been observed with numerous other examples. On Figure 8, you can see the simulated results.

The FEXT signal appears at about 0.96 ns. This corresponds well with the estimated time delay based on the propagation constants calculated using the 2D solver of ADS Momentum. The sign of the crosstalk generated signals is also in agreement with the physics of the structure.

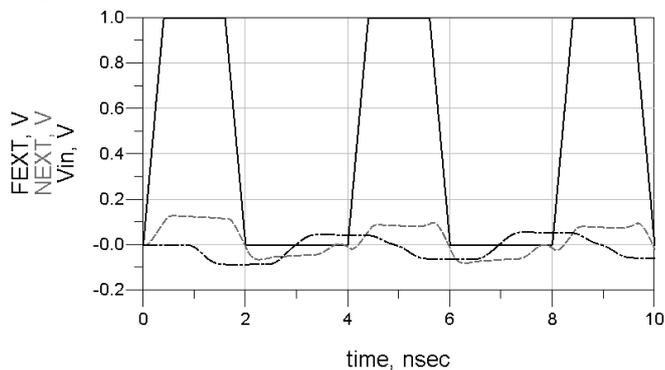


Figure 8: Simulated NEXT and FEXT signals.

Conclusions

In this paper, we have presented a fast and efficient way to obtain reliable, model based, time domain simulations. Starting from simulated or measured frequency data, it is explained how to translate the model data to an equivalent netlist that can efficiently be processed in both time and frequency domain by modern circuit simulators. They can make explicitly use of the specific pole-residue form of the rational model representation and this property is fully exploited in the netlist realisation. This results in a significant speed-up of transient simulations.

References

- [1] B. Gustavsen and A. Semlyen, "Rational Approximation of Frequency Domain Responses by Vector Fitting," *IEEE Transactions on Power Delivery*, Vol. 14 (July 1999), pp. 1052-1061.
- [2] W. Hendrickx and T. Dhaene, "A discussion of "Rational approximation of frequency domain responses by vector fitting"," *IEEE transactions on Power Systems*, Vol 21 (Feb 2006), p. 441-443.
- [3] D. Deschrijver, B. Haegeman and T. Dhaene, "Orthonormal Vector Fitting: A Robust Macromodeling Tool for Rational Approximation of Frequency Domain Responses," *IEEE Transactions on Advanced Packaging*, Vol. 30 (May 2007), pp. 216-225.
- [4] T.K. Sarkar, A.G. Lampérez and M.S. Palma, "Generation of Accurate Rational Models of Lossy Systems using the Cauchy Method," *Microwave and Wireless Components Letters*, Vol. 14 (Oct 2004), pp. 490-492.
- [5] A. Lamecki and M. Mrozowski, "Equivalent SPICE Circuits With Guaranteed Passivity From Non-Passive Models," *IEEE Transactions on Microwave Theory and Techniques*, Vol. 55 (March 2007), pp. 526-532.

- [6] R. Neumayer, F. Haslinger, A. Stelzer and R. Weigel, "Synthesis of SPICE-compatible broadband electrical models from n-port scattering parameter data," *Electromagnetic Compatibility, 2002. EMC 2002. IEEE International Symposium on*, Vol. 1 (19-23 Aug 2002), pp. 469-474.
- [7] S. Grivet-Talocia, "Passivity Enforcement via Perturbation of Hamiltonian Matrices," *IEEE Transactions on Circuits and Systems*, Vol. 51 (Sept 2004), pp. 1755-1769.
- [8] B. Gustavsen, "Computer Code for Passivity Enforcement of Rational Macromodels by Residue Perturbation," *IEEE Transactions on Advanced Packaging*, Vol. 30 (May 2007), pp. 209-215.
- [9] R. Achar, D. Saraswat and M. Nakhla, "Fast Passivity Verification and Enforcement via Reciprocal Systems for Interconnects With Large Order Macromodels," *IEEE Transactions on Very Large Scale Integration*, Vol. 15 (Jan 2007), pp. 48-59.