APPLICATION OF NEURAL NETWORKS TO EFFICIENT DESIGN OF WIRELESS AND RF CIRCUITS AND SYSTEMS

Ravender Goyal and Vladimir Veremey

Xpedion Design Systems, Inc.
4677 Old Ironsides Drive, Santa Clara, CA, 95035
Phone: (408) 987- 0603, Fax: (408) 987- 0615
www.xpedion.com
e-mail: ravenderg@xpedion.com
vladimirv@xpedion.com

ABSTRACT

Complexity of RF designs in a wireless system continues to increase significantly, to support multiple standards, multiple frequency bands, need for higher bandwidth and stringent adjacent channel specifications. Using existing design tools, time required to carry out a virtual prototyping of such complex circuits and their trade- off analysis with the baseband circuitry, is unacceptably long, because both the circuit simulation and optimization procedures can be very time-consuming. In this paper we show how Xpedion Design Systems, Inc. can considerably speed up, the modeling of RF circuits by using the neural network approach. Also, these neural net based C models are in industry standard format, and hence compatible with circuit level and system level simulators for accurate trade-off analysis of a wireless system, early in the design process.

INTRODUCTION

Present and emerging wireless systems such as GSM, CDMA, TDMA, PCS, 2.5G, and 3G systems use complex digital modulation/demodulation techniques to increase the channel capacity and the ability to transmit and receive messages/data faster, and with greater accuracy in the presence of noise and distortion [1,2]. Accurate modeling of these systems early in the design cycle, require accurate and detailed models for the circuit elements used in these systems, which include the baseband elements and the RF- transmitter and receiver elements. At the same time, for any practical use of these models at the system level design, these models should be extremely efficient to perform meaningful simulations with millions of bit stream of input signal. Behavior models are commonly used at this level of simulation, which are insufficient to accurately represent the RF elements, or require in-depth knowledge of the behavior of these RF elements and very time consuming to develop.

Neural Net technology is applied to solve this complex problem at several research labs. Xpedion Design Systems, Inc. has adopted this technology and customized it specifically for RF design problems, resulting into neural network based modeling tool to substantially enhance the numerical efficiency, while maintaining the simulation accuracy of RF circuits. In this paper, we illustrate the application of this unique modeling tool and present simulation results by analyzing a wireless communication sub-system which employs modern digital modulation techniques (N-PSK).

These numerically efficient and accurate RF circuit models, built with Xpedion's GoldenGate/NeuralNet Model Compiler (GG/NN-MC), are in industry standard C language and can be interfaced through a standard C interface, with an RF circuit level simulator, including Xpedion's **GoldenGate™** RF simulator, and system level simulators, such as COSSAP from Synopsys, SPW from Cadence, and SystemView from Elanix.

NEURAL NETWORK MODELING APPROACH

The circuit elements of RF section of a wireless communication system, such as amplifiers, mixers, multipliers or VCOs, comprise the non-linear elements that must also be simulated during the process of RF design. Let us consider an amplifier and assume that the designed circuit has an input and output impedance of Z_0. At the system level, the amplifier can be described by a set of non-linear input-output relationships. The telecommunication signal is presented in the form $s(t) = \hat{S}(t)e^{j\omega_0 t}$, where $\hat{S}(t)$ is a complex modulation function and ω_0 is the carrier frequency. The general block-diagram of a circuit with N parameters $(p_1, p_2,..., p_N)$ is presented in Fig.1. The list of the circuit parameters depends on the circuit design specifications; for example, the carrier frequency ω_0 can be considered as one of the parameters of the design space of an amplifier $(\omega_0 = p_1)$.

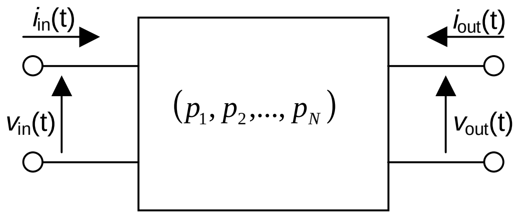

Fig. 1. General block diagram of a circuit with N parameters

Let equations (1) and (2) describe the input-output relationships between the various voltages and currents:

$$\mathrm{Re}\left(\hat{I}_{out}(t)\right) = f_1\left(p_1, p_2,..., p_N, \mathrm{Re}(V_{in}(t)), \mathrm{Im}(V_{in}(t))\right), \quad (1)$$
$$\mathrm{Im}\left(\hat{I}_{out}(t)\right) = f_2\left(p_1, p_2,..., p_N, \mathrm{Re}(V_{in}(t)), \mathrm{Im}(V_{in}(t))\right), \quad (2)$$

Using the measurement results from an already designed and manufactured amplifier circuit or the simulation results obtained from the harmonic balance simulator, GoldenGate/NN-MC software builds and trains a neural network model that can be subsequently used as very accurate model for the amplifier. This network, designed to efficiently model the non-linear circuit modeling problem at hand, contains one hidden layer of radial "Inverse Mexican Hat" functions, N+2 elements in the input layer and one element in the output layer. For the present case, we need to create two neural networks for the real and imaginary parts of the output current as functions of the circuit parameters and the input voltage. Next we design an efficient training algorithm based on the Powell's optimization scheme, with the initial point determined by the Genetic Algorithm (GA). An alternate training procedure described in [5] can also be used. This procedure is based on the wavelet theory and quasi-Newton Broyden-Fletcher-Goldfarb-Shanno algorithm.

MODELING EXAMPLE

To illustrate the application of the technique described above, we consider an amplifier designed and manufactured using state-of-the-art BiCMOS, sub- micron process. This amplifier is a complex design and includes fairly large number of BJTs and diodes (several hundred) non-linear devices and large number of lumped elements, for biasing circuitry, modeling of parasitics, and the core amplifier circuitry, and operates at cellular band frequency of 800 MHz. As a first step, we simulated the amplifier by using the harmonic balance technique available in Xpedion's **GoldenGate**^{TM} simulator. **GoldenGate**^{TM} is a suite of RF and microwave simulators including linear RF, non-linear RF using Harmonic Balance and most importantly for wireless communication design Envelope transient simulator, all integrated in a unified design environment. Next, we created the neural networks and trained them with the obtained data. The results of the circuit simulation are presented in Fig. 2, along with those derived by using the NN.

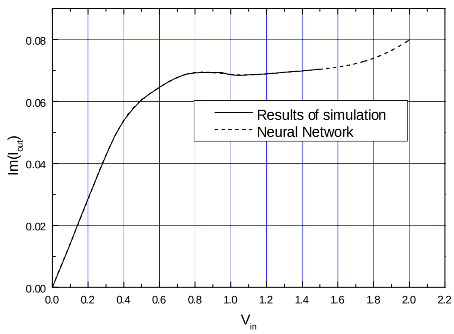

Fig.2. (a) Imaginary part of output current versus the magnitude of input voltage

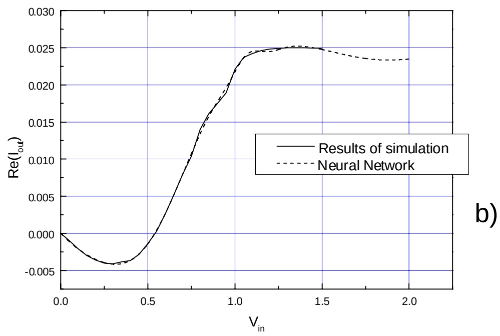

Fig.2. (b) Real part of output current versus the magnitude of input voltage

It is important to note from these figures that the NN technique can be also rather effectively applied for extrapolation, the region where the simulations are not performed, or the measurements are not made. To verify the model efficiency, we performed envelope transient analysis [6] to the amplifier circuit as well as to its NN model. We have used an input QPSK signal and have compared the spectra of the amplified output signals. The simulations performed by using Envelope Transient simulation in Xpedion's **GoldenGate**^{TM} software are presented in Fig. 3.

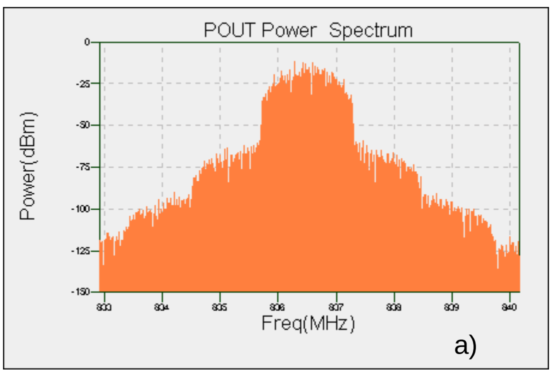

Fig. 3. Result of Power Spectrum Simulation using the Envelope Transient simulator of Xpedion's **GoldenGate**^{TM} software.
(a) Spectrum of the output signal obtained with envelope transient analysis of the amplifier

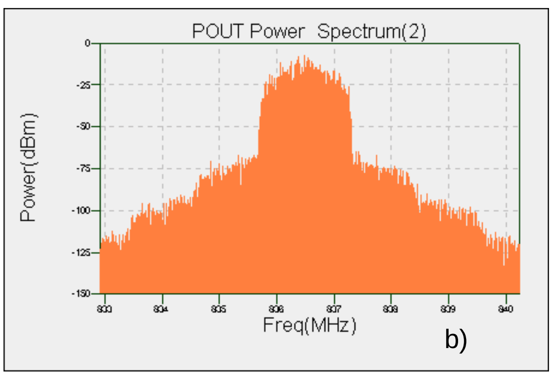

Fig. 3. Result of Power Spectrum Simulation using the Envelope Transient simulator of Xpedion's **GoldenGate**^{TM} software.
(b) Spectrum of the output signal obtained with the NN model

The use of NN models, typically improve the simulation performance (CPU time) by a factor of 50x to more than a factor of 100x, while maintaining the accuracy of simulation results. Spectral re-growth, adjacent channel interference and other characteristics of the amplifier were also evaluated by using the amplifier model within 3% accuracy, compared to transistor level circuit simulation.

REFERENCES

1. Lathi, B.P., Modern Digital and Analogous Communication Systems, Holt, Rinehart and Winston, New York, NY 1983.
2. Feher, K., Advanced Digital Communications; System and Signal Processing Techniques, Prentice-Hall, Englewood Clifs, NJ, 1987
3. Gupta, K.C., Garg, R., Chadha, R., Computer-Aided Design of Microwave Circuits. Artech House, 1981.
4. Veremey, V., Mittra, R., “Efficient computation of interconnect capacitance using domain decomposition approach,” IEEE Trans. Advanced Packaging, vol.22, no.3, August 1999, pp. 348-355.
5. Harcouss, Y., et al., “The use of artificial neural networks in nonlinear microwave devices and circuit modeling: an application to telecommunication system design,” International Journal on RF and Microwave CAE, John Wiley & Sons, Inc., vol. 9, 1999.
6. Ngoya, E., Larcheveque, R., “Envelop transient analysis: a new method for the transient and steady state analysis of microwave communication circuits and systems,” IEEE MTT Symposium Digest, 1996, pp. 1365-1368.