A Parallel Computational Technique for High Frequency HBT Circuit Simulation

Yiming Li[1,2,*], Kuen-Yu Huang[3], Cheng-Kai Chen[4], and C. P. Lee[3]

[1]National Nano Device Laboratories, Hsinchu 300, Taiwan, *ymli@cc.nctu.edu.tw*
[2]Microelectronics and Information Systems Research Center[**];
[3]Institue of Electronics[**]; [4]Department of Computer and Information Science[**]
[**]National Chiao Tung University, Hsinchu 300, Taiwan
[*]Corresponding author. Mailing address: P.O. Box 25-178, Hsinchu 300, Taiwan

ABSTRACT

We present a parallel-in-waveform simulation technique for high frequency heterojunction bipolar transistor (HBT) circuit characterization. On the contrary to the conventional frequency domain analysis, the HBT circuit equations are solved with waveform relaxation and monotone iterative methods in large signal time domain. This approach has been applied to calculate HBT circuit RF output power, the 3rd order inter-modulation and intercept point. The studied unusually high linearity of HBTs' circuit at ultra high frequency region is quite useful for wireless applications.

Keywords: HBT circuit, RF characteristics, and Parallel-in-waveform.

1 INTRODUCTION

Due to the unusually high linearity of HBTs at high frequencies, these active devices have been of great interests for RF and wireless applications in recent years [1-6, 11-15]. The common approach to calculate the inter-modulation distortion and two-tone characteristics for an HBT is to solve a set of equivalent circuit ordinary differential equations (ODEs) in frequency domain. The harmonic balanced method [14-15] is an approach for such RF problem. This frequency domain approach has its merits and limitations in studying the physical properties of HBT with time variation. Another approach to the analysis of electric characteristics for HBT RF circuit is to solve a set of equivalent circuit ODEs in time domain. The time domain results are then further calculated with fast Fourier transformation (FFT) for obtaining its spectrum. However, the discretized ODEs in circuit simulation are often solved with conventional Newton's iterative (NI) method. It is well known the NI method is a local method; in general, it has a quadratic convergence property in a sufficiently small neighborhood of the exact solution, and hence it encounters convergence problem for practical engineering applications.

In this paper, we propose a novel parallel simulation method for HBT physical characteristics calculations in large signal time domain. Various RF characteristics, such as RF output power, the 3rd order inter-modulation and the 3rd order intercept point are verified for a HBT circuit operating at high frequency systematically. This robust approach is mainly applying the monotone iterative method instead of the NI method to solve the discretzied ODEs. The MI method has been successfully developed and applied to semiconductor device simulation by us earlier [7-10]. Based on the waveform relaxation method, the monotone iterative method, and the parallel-in-waveform technique, the HBT circuit equations can be directly solved with larger input signal in time domain. The computed output results are then analyzed with FFT to obtain the necessary information in frequency domain. This approach strongly depends on the robustness of the nonlinear ODE solver; compared with the conventional NI method, our approach converges globally and is inherently parallel.

First of all, the coupled ODEs are decoupled with WR method. Each decoupled nonlinear ODE is then solved directly with MI as well as Runge-Kutta method. The proposed computational technique has been successfully implemented on a PC-based cluster with message passing interface (MPI) library. The primary parallel results show that a well-designed parallel algorithm can significantly reduce the execution time up to an order of magnitude.

This paper is organized as follows. Sec. 2 states the HBT circuit model and introduces the computational algorithms. Sec. 3 presents the simulation results. Sec. 4 draws the conclusion.

2 MODEL AND SIMULATION METHODS

As shown in Fig. 1, based on the node current flow conservation theory, the simulation model is firstly formulated with nodal equations. The system of node equations for time dependent HBT circuit is a set of nonlinear coupled ODEs. At nodes C, E, and B we have the equations (1)-(3), respectively. The current models I are nonlinear functions of unknown variables to be solved. At nodes BX, CX, and EX we can further formulate, respectively, other governing equations. The capacitances C in these equations (1) - (6) are nonlinear functions of unknown variables. The Gummel-Poon large signal model for HBT device, as shown in Fig. 2, is included in this circuit simulation. The unknowns to be solved in the system of ODEs are V_C, V_E, V_B, V_{BX}, V_{CX}, and V_{EX}.

The nonlinear system is strongly coupled different nonlinear ODEs, due to the exponential dependence of current and capacitance models. To solve the system efficiently in time domain, we propose here a decoupled and global convergent parallel simulation techniques to solve this system ODEs in large signal time domain directly. Firstly, under the steady state condition, we find the DC solution as the starting point to compute all transition solutions. Based on global convergence of MI method, we can solve all time step jobs independently and hence this algorithm implies a parallel-in-waveform scheme. This inherently parallel scheme is quite robust for large signal transition time analysis.

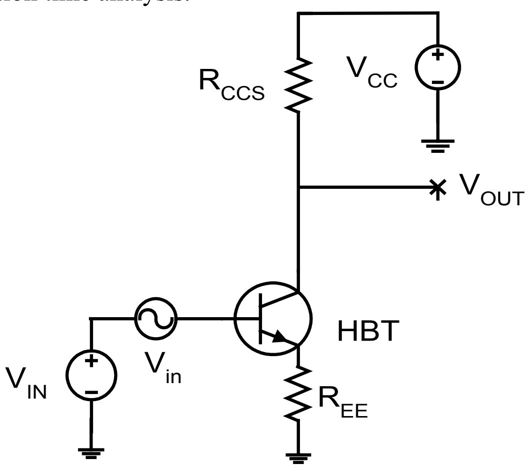

Fig. 1. A HBT circuit for large signal time domain analysis.

$$C_{JCX}(\frac{dV_{BX}}{dt}-\frac{dV_C}{dt})+C_{DR}(\frac{dV_B}{dt}-\frac{dV_C}{dt})+ C_{JCI}(\frac{dV_B}{dt}-\frac{dV_C}{dt})+I_2+I_{BL2}-\frac{I_{CT}}{q_b}+ \frac{V_{CX}-V_C}{R_C}=0 \tag{1}$$

$$C_{DF}(\frac{dV_B}{dt}-\frac{dV_E}{dt})+C_{JE}(\frac{dV_B}{dt}-\frac{dV_E}{dt})+ I_1+I_{BL1}+\frac{I_{CT}}{q_b}+\frac{V_{EX}-V_E}{R_C}=0 \tag{2}$$

$$C_{DR}(\frac{dV_B}{dt}-\frac{dV_C}{dt})+C_{JCI}(\frac{dV_B}{dt}-\frac{dV_C}{dt})+ C_{DF}(\frac{dV_B}{dt}-\frac{dV_E}{dt})+C_{JE}(\frac{dV_B}{dt}-\frac{dV_E}{dt}) +I_1+I_{BL1}+I_2+I_{BL2}+\frac{V_B-V_{BX}}{R_B}=0 \tag{3}$$

$$C_{JCX}(\frac{dV_C}{dt}-\frac{dV_{BX}}{dt})+\frac{V_B-V_{BX}}{R_B}+\frac{V_{IN}-V_{BX}}{R_{B2}}=0 \tag{4}$$

$$\frac{V_C-V_{CX}}{R_C}+\frac{V_{CC}-V_{CX}}{R_{CCS}}=0 \tag{5}$$

$$\frac{V_E-V_{EX}}{R_E}-\frac{V_{EX}}{R_{EE}}=0 \tag{6}$$

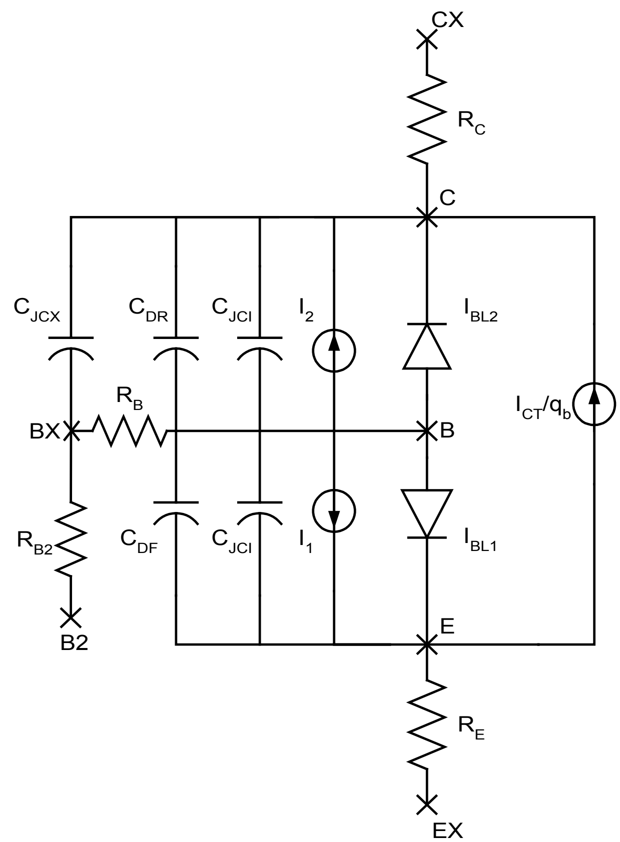

Fig. 2. A Gummel-Poon large signal model for HBT circuit simulation.

Fig. 3 demonstrates the proposed simulation procedure for large signal time domain simulation. To solve these nonlinear ODEs in time domain, let a time step $t = t_n$ is given, our approach is: (i) Use WR method to decouple all equations. (ii) Each decoupled ODE is solved with Runge-Kutta and MI methods (iii) Update the newer results and back (i). The iteration loops will be terminated when the specified stopping criterion is reached. If the results are computed at this time step t_n, then $t_n = t_n + \Delta t$ and repeat the solution procedure (i) - (iii) until the time step meets the specified time period T. The proposed parallel-in-waveform computing method (see Fig. 4) has been successfully implemented on our constructed PC-based cluster with MPI library. This parallel methodology is motivated by the principle of pipeline. Our parallel experiences show that a well-designed parallel algorithm can reduce the execution time up to an order of magnitude. For a HBT circuit simulation at 1GHz with more than 40 waveform periods, the achieved parallel speedup factor is about 6.5 on a 8-PCs based Linux-cluster with MPI library.

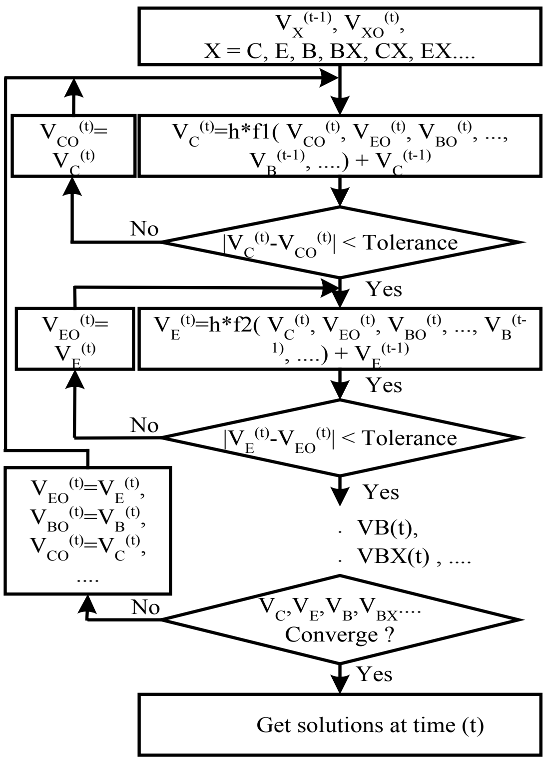

Fig. 3. Flowchart of the proposed computing algorithm for the HBT circuit simulation.

The notation subscript *o* is the previous iteration results and the superscript *t*-1 and *t* represent the previous and present time steps. In our calculation experience, the convergence criterion for all quantities (maximum norm error < 10^{-10} and 10^{-8} for MI and WR loops) can be reached by only 8 - 12 MI loops and 25 - 30 WR loops, respectively.

3 RESULTS AND DISCUSSIONS

In this section, we present DC and various nonlinear characteristics for HBT circuit operated at a two-tone high frequency input signal. The designed simulation shows a good consistency with those experimental data. For the DC I-V curves of HBT device, our approach compared with the well-known SPICE tool shows the same accuracy results. As shown in Fig. 5, the DC family curves of simulated HBT device are presented to demonstrate the accuracy of the method with a very fast calculation time. The symbols-marked curves are with SPICE simulator results and the lines are our calculations.

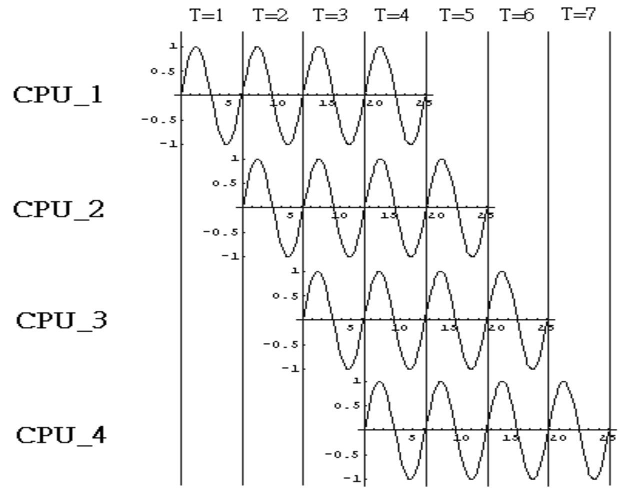

Fig. 4. An illustration of the proposed parallel-in-waveform technique for HBT circuit time domain simulation.

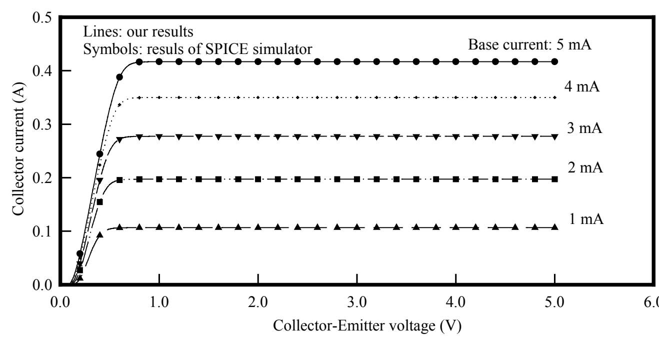

Fig. 5. The DC family curves of simulated HBT circuit. The lines are the results with our approach and the symbols present the output of SPICE.

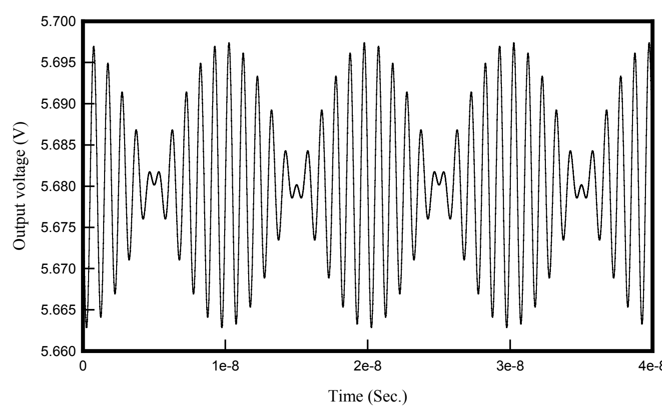

Fig. 6. The output characteristics of the HBT circuit in time domain, where the HBT circuit is operated at a two-tone input signal. More than 50 cycles are computed.

Fig. 6 shows the output voltage (V_{OUT}) with two-tone input signal in time domain. More than 50 cycles are computed with our parallel circuit simulator. Furthermore, the excellent characteristics of simulated HBT circuit are further analyzed. As shown in Fig. 7, the spectrum of V_{OUT} as the amplitude of input voltage (V_{IN}) equals 0.008V are plotted. They are calculated from FFT with computed time domain data. It confirms the experimental measurement; however the same result is impossible to calculate with SPICE simulator. In addition, compared with the results obtained with harmonic balanced technique in frequency domain, our results are rather stable and more accurate.

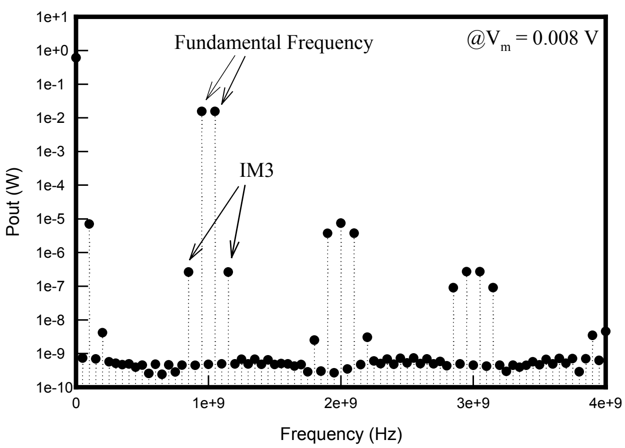

Fig. 7. The computed spectrum for the output signal. It is directly calculated from time domain data with FFT. Very good IM3 is obtained with our approach.

The OIP3 result is illustrated in Fig. 8. Output Power of the fundamental frequency and the third order inter-modulation (IM3) as a function of input power are calculated. As shown in Fig. 8, the cross point of those two extrapolated lines is the third-order intercept point (OIP3). Fig. 9 shows the OIP3 of the HBT device as a function of the collector current. It indicates the nonlinearity of the HBT device with different operation current.

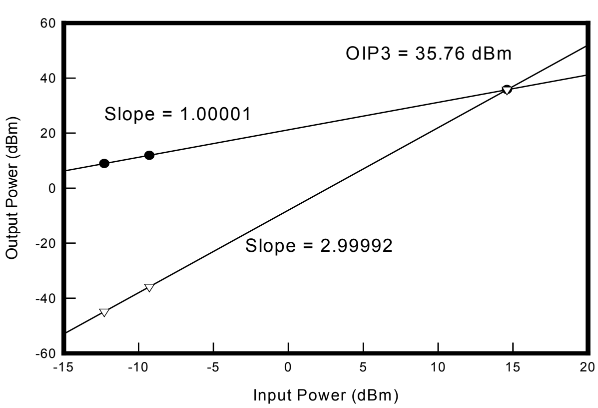

Fig. 8. The computed output power and IM3. The cross point indicates the OIP3.

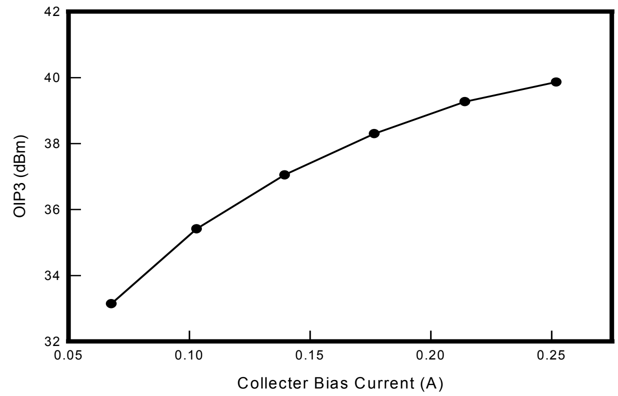

Fig. 9. The plot of OIP3 versus collector current of HBT device.

4 COLCLUSIONS

In conclusion, based on the WR and MI methods we have presented here a parallel-in-waveform circuit simulation technique. With this approach, high frequency characteristics of HBT circuit have been directly analyzed with large signal time domain results. Simulation results on a realistic InGaP HBT with 20μm^2 emitter area showed the accuracy and efficiency of the method. This approach has been applied to calculate HBT circuit RF output power, the 3rd order inter-modulation and intercept point. Our studies presented in this work provide an alternative for RF characterization and wireless applications.

REFERENCES

[1] N. Guofu, et al., Proc. Bipolar/BiCMOS Circuits & Tech. Meeting, 50, 1999.
[2] N. Guofu, et al., IEDM Tech. Dig., 573, 1999.
[3] S. Sheu, et al., IEEE Trans. ED, 45, 326, 1998.
[4] S. Sheu, etal., Proc. IRPS, 248, 1997.
[5] M. F. Chang, et al., Proc. IEEE MILCOM, 704, 1998.
[6] D. Tsoukalas, "Simulation of Semiconductor Processes and Devices 2001," Springer-Verlag, 226-229, 2001.
[7] Y. Li, et al., Proc. IEEE VLSI-TSA, 27-30, 1999.
[8] Y. Li, et al., Tech. Proc. Modeling and Simulation of Microsystems, 538-541, 2001.
[9] Y. Li, et al., Abst. Book Conf. Comp. Phys., O38, 2001.
[10] Y. Li, et al., Comp. Phys. Comm., 142, 285-289, 2001.
[11] L.E. Getreu, Modeling the Bipolar Transistor, Elsevier, New York, 1984.
[12] W. Liu, Handbook of III-V Heterojunction Bipolar Transistors, John Wiley & Sons, 1998.
[13] Ron M. Kielkowski, SPICE: practical device modeling, McGraw-Hill, New York, 1995.
[14] B. Trpyanovsky, et al., Comp. Meth. Appl. Mech. Eng., 181, 467-482, 2000.
[15] K. S. Kundert, J. K. White, and A. S.-Vinventelli, "Steady-State Methods for Simulating Analog and Microwave Circuits," Kluwer Academic, 1990.