Overview of Computer-Aided Analysis Tools for RFIC Simulation: Algorithms, Features, and Limitations

K. Mayaram* D.C. Lee† S. Moinian‡ D. Rich§ J. Roychowdhury¶

Abstract

Design of the RF section in a communication IC is often a challenging problem. Although several computer-aided analysis tools are available they are not effectively used because there is a lack of understanding about their features and limitations. This paper attempts to explain the simulator-specific terminology without resorting to mathematical details. The shortcomings of conventional SPICE-like simulators and the analyses required for RF applications are described. Various analysis methods that are currently available for RF simulation are presented and commercial simulators are compared in terms of their functionalities and limitations.

1 Introduction

The wireless/personal communication electronics market is growing at a rapid pace and complete systems on a chip are emerging as a reality. Although only a small section of the system operates at RF frequencies, this section is the biggest challenge in the design process [1]. One of the factors contributing to this problem is the lack of computer-aided analysis tools that support RF design [1]. Several commercial tools are available, but a lack of understanding of their features and limitations also contributes to this problem.

For the design of a typical RF section, one would like to simulate basic blocks such as amplifiers, mixers, oscillators, voltage controlled oscillators (VCOs) and phase-locked loops (PLLs). Because of the signal frequencies and circuit time-constants involved, and the types of analyses required, simulators such as SPICE are neither adequate nor appropriate. Although SPICE is used to simulate some of these basic blocks by running long transient analyses, the accuracy of the results may be poor because of accumulated errors. Furthermore, these poor results are obtained at the expense of significant computational resources. It is for this reason that general as well as special purpose simulation tools have been developed that address the needs of RF design. The specific features include:

- rapid simulation of the periodic or quasi-periodic steady state of a circuit
- accurate simulation of harmonic and intermodulation distortion in order to determine gain compression and intercept points
- simulation of up/down conversion of noise caused by circuit nonlinearities and phase noise or jitter for oscillators
- simulation of the turn-on transients of oscillators and the capture process of PLLs
- simulation of distributed elements

In this paper we examine these issues from a designer's perspective. In Section 2, we first explain the limitations of a SPICE-like simulator. This is followed, in Section 3, by a brief discussion of various techniques that are currently used for simulating RF circuits. An attempt is made to explain the terminology and salient concepts. Currently available commercial simulators are then described in terms of their features and limitations in Section 4. Finally, a brief discussion and conclusions are presented in Section 5.

2 SPICE and what it can simulate

The circuit simulator SPICE supports various analyses which can be classified as dc (.op, .dc), small-signal ac (.ac, .noise, .disto), and transient (.tran, .four). Since noise and distortion place a limit on the performance of communication systems, we will focus on these analyses in the context of SPICE. The shortcomings of SPICE for RF applications are also identified.

Noise calculations in SPICE are based on a small-signal linearized analysis of the nonlinear circuit at its dc operating point. Because of the linearization, noise mixing effects cannot be simulated as they arise due to circuit nonlinearities. Hence noise up/down conversion and phase noise or jitter for oscillators cannot be determined in SPICE. For this reason an enhanced noise analysis is required (see Section 3.6).

Distortion is commonly calculated using the .four command in SPICE. A transient analysis is first performed so that the circuit operates in its periodic steady state. The determination of the steady state is done by the user and is subject to errors. Then a Discrete Fourier Transform (DFT) is applied to the simulated time-domain waveform. A fundamental frequency is identified by the user and SPICE reports the magnitude and phase of various harmonics. This method can also be used to calculate intermodulation distortion (IMD) by a proper choice of the fundamental frequency. For Fourier analysis, the signals must be periodic, implying that the tones must be commensurate[1].

For example, IMD is determined by simulating the circuit with two closely spaced tones, f_1 and f_2. The third-order IM terms (IM3) are at $2f_1 - f_2$ and $2f_2 - f_1$. Assuming $f_2 > f_1$, the slowest varying component is at $f_2 - f_1$, while another component of interest is much faster, at frequency $2f_2 - f_1$. Using a conventional transient analysis, at least one period of the slow $f_2 - f_1$ component must

*School of EECS, Washington State University, Pullman, WA.
†Bell Laboratories, Allentown, PA.
‡Bell Laboratories, Reading, PA.
§Bell Laboratories, Allentown, PA.
¶Bell Laboratories, Murray Hill, NJ.

[1] i.e., all frequencies must be exactly divisible by a single common frequency.

0-7803-3669-0 $5.00 © 1997 IEEE

be simulated. To resolve the fastest signal, the maximum timestep must be smaller than half the period corresponding to $2f_2 - f_1$. The ratio of the fastest to slowest frequencies is 100:1 or more, resulting in an extremely slow transient analysis, since many time points have to be simulated. Furthermore, simulation of high-Q oscillators also requires very long transient simulations, since, for example, a Q of 10,000 suggests that the turn-on transient time will be of the order of 10,000 cycles of the oscillation period.

Determination of the distortion using transient analysis can result in several errors. An excellent discussion of these can be found in [2]. First, the circuit must be in periodic state for some fundamental frequency before the Fourier analysis is performed. This can be computationally expensive for circuits that exhibit widely-separated time constants, or for which the fundamental frequency required for commensurate tones is very small. Second, the time-domain simulation must be performed with tight tolerances to maintain a low numerical noise floor. Otherwise it is extremely difficult to resolve low power harmonics. Third, the DFT algorithm introduces interpolation and aliasing errors which can be minimized by use of the Fourier integral method [3, 2]. The last error is important in a circuit simulator that uses variable time-steps in the transient analysis. Since the DFT can be applied only to waveforms that are uniformly sampled, the non-uniformly spaced data produced by the simulator must be interpolated onto a uniform grid first. This can result in significant errors when the interpolation order is low.

Numerical errors introduced by the simulator makes resolving small intermodulation products difficult. When distortion levels are low, the Volterra series method can be used to determine IM3 accurately, using three small-signal AC analyses [4]. Thus accurate estimation of IM3 is possible at relatively small computational cost. The .disto command in SPICE allows accurate calculation of IM3 using the Volterra series method. Although SPICE2G6 supports .disto the implementation has bugs and a correct implementation is available in SPICE3 [4].

The accuracy of the calculated distortion depends directly on the quality of the semiconductor device models. Models with discontinuities introduce spurious spectral components, hence smooth models (e.g., [5]) and careful characterization are essential for RF applications.

Another problem associated with time-domain approaches is the simulation of distributed models. Although such elements typically have simple frequency-domain representations, time-domain simulation requires convolution, which, when performed directly, is computationally expensive. However, efficient algorithms are available for time-domain convolution of distributed elements [6, 7, 8]. The only distributed element available in SPICE is the transmission line.

3 Algorithms for RF Simulation

As described in the previous section, conventional SPICE analysis is not adequate for many RF needs, for which several special techniques have been developed. Most RF-related simulations can be performed using either frequency- or time-domain methods. Since the two classes of methods are in many aspects complementary, combining the two can bring together their advantages, resulting in hybrid algorithms. Several methods in these categories are described in this section.

3.1 Time-domain methods

Time-domain methods can be used to determine the periodic steady state of a circuit in the following manner. The underlying differential equations for the circuit are solved by forcing the constraint that the solution is periodic in the steady state. This condition is expressed as $v(0) = v(T)$, where v is the vector of node voltages and T is the period. $V(0)$ is the vector of node voltages at time 0 (i.e., the initial conditions), and $v(T)$ the node voltages at time T. The algorithm then determines the initial conditions that lead to a periodic solution.

For circuits that are driven by periodic signals, i.e., forced or non-oscillatory circuits, the period T is a known quantity. However, for oscillators the period is also an unknown and must be determined by the simulator in addition to the determination of $v(0)$.

One popular method that is available for solving the above system of equations is the Newton shooting method. This method has been shown to be reliable in SSpice [9] and a variant of this method is available in the SpectreRF simulator from Cadence. This method works well even in the presence of strong nonlinearities. A description of the method follows. For simplicity consider a forced circuit where the period T is known. A conventional transient analysis is performed over one period with some starting initial conditions. This solution is then used to determine the change in initial conditions required to obtain $v(0) = v(T)$ by Newton's method. The initial conditions are updated and a new transient analysis is performed to obtain a new $v(T)$. The process is repeated until a $v(0)$ is found such that $v(0) = v(T)$. Since this is an iterative process, convergence is ensured to within a prescribed error tolerance. A tighter error tolerance would require more iterations and hence more computation. It is important to note that this error tolerance affects the distortion results obtained from a subsequent Fourier analysis. A mismatch in the periodicity constraint will corrupt the results of the Fourier analysis [2].

In the above process, dense matrix manipulations are required for determining the solution. This places a restriction on the maximum size of the circuit that can be simulated as approximately 300 nodes. A recent method [10] overcomes these limitations allowing circuits with about 1000 nodes to be simulated.

As with any time-domain simulation method, accurate distortion calculation requires care in selecting simulation tolerances and the choice of algorithm used for computing the harmonics. The Fourier integral method coupled with the time-domain shooting method with appropriate tolerances provides accurate distortion calculation.

Time-domain methods are however not well suited for multi-tone problems. They can be used only when the signals are commensurate. This can be a problem when simulating mixers, in which the IF, RF, and LO frequencies are incommensurate or almost so, i.e. for a very large value of T. When the RF signal is small then the LTV method described in Section 3.5 can be used.

3.2 Harmonic Balance Method

Harmonic balance (HB) [11, 12] is a well established frequency-domain method for periodic and quasi-periodic steady-state analysis of nonlinear circuits. It is used to analyze distortion and transfer characteristics in amplifiers, mixers, and oscillators.

In harmonic balance, the circuit is assumed to be in steady state and each signal is represented in a Fourier series:

$$v(t) = a_0 + \sum_m [a_m \cos(\omega_m t) + b_m \sin(\omega_m t)]$$

with frequency components ω_m at integer combinations of the excitation frequencies $\hat{\omega}$. The Fourier coefficients a_m and b_m characterize harmonic and intermodulation distortion and are of interest to designers.

HB is formulated by expressing the circuit differential equations in terms of the Fourier coefficients above, and by replacing differentiation in the time-domain by algebraic multiplication in the frequency-domain. This results in a large system of nonlinear algebraic equations. Each circuit variable requires many Fourier coefficients, hence the size of this system is much larger than that of the circuit differential equations. The system is typically solved using a Newton's method.

Most implementations of the harmonic balance method require the manipulation of relatively dense matrices, consuming excessive CPU times and large amount of memory for even medium-sized circuits. Recent work [13, 14] has incorporated efficient matrix algorithms that extend harmonic balance to circuits containing tens to hundreds of transistors. The run times and memory requirements of careful software implementations grow almost linearly with the size of the circuit and the number of Fourier coefficients. Today, multi-tone harmonic balance problems with one million unknowns can be solved in several hours on an engineering workstation with 200 megabytes of memory.

Harmonic balance has several salient features.

- The simulation run time is relatively insensitive to the numerical values of the excitation frequencies. This is because the minimum number of "time-domain" samples in harmonic balance is dictated by the number of Fourier coefficients, and not by the Nyquist rate in conventional transient analysis. This makes harmonic balance particularly attractive for circuits driven by multi-tone excitations.
- When a circuit is designed to exhibit good linearity (low distortion), or when a weak stimulus is applied to a nonlinear circuit, the steady state solution can be approximated using a small number of Fourier coefficients. Naturally, when the circuit response is strongly nonlinear, many Fourier coefficients and time-domain samples would be needed in order to minimize the aliasing effects, and convergence of the method can become less reliable.
- Frequency-dependent distributed elements are handled with no difficulty.
- Parameter sweep (distortion and/or gain compression vs. signal amplitude and/or frequency) can be performed very efficiently, because the solution from a previous point is often an excellent initial guess for the next point.

Oscillators are handled by treating $\hat{\omega}$ as an unknown in the system of nonlinear equations, and setting one Fourier coefficient of one of the signals to zero.

3.3 Mixed frequency-time methods

For a special class of multi-tone circuits, *mixed frequency-time* approaches are superior to both time and frequency domain techniques. Two methods, the original [15] and the more recent [16] are available, differing in equation formulation and in numerical properties. [15] is capable of handling strong nonlinearities provided they are excited by only one of the tones, typically the fastest tone. All other tones must excite the circuit only in a mildly nonlinear fashion. [16] can treat any number of strongly and weakly nonlinear tones. One application of the methods is the distortion analysis of switched-capacitor circuits, in which the clock causes switching, but the path of the signal tones is almost linear. Some switching mixers and power converters are also appropriate for the methods.

The method of [15] works by integrating the circuit's differential equations in the time domain for a *few* periods of the fast, strongly nonlinear one. This establishes a relation between the (unknown) start and end points of these periods. Another relation between these points is obtained by using a Fourier series expansion in the weakly nonlinear tones. Equating the two relations results in a system of equations whose solution captures the mildly nonlinear tones of the quasi-periodic solution directly, without having to integrate every cycle of the fast tone. Hence the procedure is considerably more efficient than using time-domain shooting for the multi-tone problem. Also, since it does not use a frequency-domain expansion for the strongly nonlinear tone, which can require many Fourier coefficients, it is more efficient than purely frequency-domain methods. The recent algorithm of [16] is based on formulating the multi-tone problem as a partial differential equation (PDE). The PDE is solved in the time domain in the dimensions corresponding to the strongly nonlinear tones, and in the Fourier domain in the dimensions corresponding to the weakly nonlinear ones. The solution of the PDE provides information about the weakly nonlinear tones directly in the frequency domain.

3.4 Envelope-following methods

In many RF and other circuits, a high-frequency signal is used as carrier for the information signal, or envelope, that varies much more slowly. Often, the information signal is not periodic/quasi-periodic and it is of interest to predict its transient behavior. Applications include analyzing the stability of AGC circuits and capturing the transient behavior of phase-locked loops and DC-DC converters. A straightforward approach to this problem is to perform a conventional transient analysis of the circuit for the duration of the slow information signal. Unfortunately, the simulation is forced to follow the fast-varying carrier or clock signal, thereby requiring a prohibitively large number of time-steps and accumulating significant errors.

Recently, algorithms that avoid this inefficiency and provide good accuracy have been announced, first by [17] and independently by [18]. The algorithms operate at the time-scales of the slow envelope, and are relatively insensitive to the rate of the fast signal. A differential equation in the envelope is solved, using an outer-loop transient whose time-steps are not limited by the fast signal; at each time-point of the transient, a ***steady-state problem*** in the fast signal is solved in the inner loop. Both algorithms use harmonic balance to solve the steady-state problem, hence are of limited applicability for strongly nonlinear circuits. [18], however, uses fast harmonic balance methods [14, 19] and can analyze large circuits. The limitation to moderate nonlinearities has recently been overcome by techniques based on partial differential equations [16], in which the inner-loop steady-state problem is also addressed in the time domain. This has made envelope simulation possible for a broader class of circuits, including chopping power converters, digital PLLs and pulse-width-modulated communication circuitry.

3.5 Linear Time-Varying (LTV) Analysis

This analysis provides the large signal equivalent to the small signal AC analysis of SPICE-like simulators. Thus large signal frequency response, impedance analysis, stability analysis and noise can be performed with circuits that exhibit frequency translation such as mixers.

As an example consider a mixer represented by an ideal multiplier with two input ports and one output port. The gain from one input port to the output can be calculated in terms of the value of the other input. If the other port is at 0 the gain is 0. If the other port input is a constant 10 the gain is 10. If the other port is a sinusoid, $A\sin(\omega t)$, then this is the gain of the mixer. The gain is now a time varying quantity and accounts for frequency translation effects.

Assume a circuit is driven by two tones, one of which is a small-signal. The large-signal response is determined by applying the large-signal excitation, and computing the steady-state response using harmonic balance or the time-domain shooting method described earlier. Linearizing the circuit about its periodic operating point results in a linear time-varying circuit. Then the small-signal excitation is applied to the linear time-varying circuit and the complete response is obtained. In the harmonic balance literature, this method is known by a number of names including conversion matrix analysis, large-signal/small-signal analysis, or simply as the mixer analysis in LIBRA. An efficient implementation of this method in conjunction with the time-domain shooting method for large circuits is described in [20].

The main benefits of this method are numerical efficiency and the fact that the small-signal excitation can be at an arbitrary frequency, a feature that the large-signal time-domain shooting method cannot handle. Two fundamental assumptions should be noted:

- the small-signal excitation does not perturb the time-varying operating point of the circuit, and
- the transfer function from the small-signal excitation to every node in the circuit is linear.

A major advantage of the linear time-varying analysis is that it provides results not only for the fundamental analysis frequency, but also for all the harmonics. Thus the conversion gain for mixers can be calculated.

3.6 RF Noise

Estimation of electrical noise is important in the design of RF circuits, since noise determines critical specifications like SNR and BER. Noise within RF components like mixers and oscillators, which rely on nonlinearities and large signal swings for their operation, propagates differently from that in linear blocks like amplifiers. One difference is that the former can frequency-shift the power spectrum of device noises, whereas the latter do not. Hence conventional noise analysis in SPICE, which is based on stationary stochastic process ideas, does not suffice for RF noise; nonstationary/cyclostationary processes, or frequency-correlated noise concepts, need to be used.

Noise in nonlinear circuits manifests itself in two qualitatively different forms: mixing noise and phase noise. The former occurs in non-oscillatory circuits, e.g., in nonlinear amplifiers and mixers. The key feature of mixing noise is that it appears as a (usually small) additive component to the signal. In contrast, phase noise, which appears in all oscillatory circuits, has the effect of spreading the pure spectral components that would result if the oscillator were noiseless. Through this mechanism, small device noises can generate substantial power content at frequencies close to the oscillator's fundamental and harmonics. Phase-noise-induced transfer of carrier power to nearby frequencies can result in signficant adjacent channel interference in RF circuits. This spectral spreading can also be characterized equivalently as random jitter in the zero-crossings of the oscillatory signal.

3.6.1 Mixing noise

The Monte-Carlo approach for noise simulation is common amongst designers. In this approach, a long transient simulation is carried out, with noise sources in the circuit represented by randomly generated waveforms of the appropriate power spectra. The resulting waveform is time-averaged to obtain the average noise power. This method can be prohibitively expensive since its rate of convergence to the correct average value improves only as the square root of the length of the simulation. In addition, since noise values are typically extremely small compared to the signals they ride on, severe degradation of the result can occur because of artificial numerical noise generated by the simulator, and because of the finite precision of digital computers. Nevertheless, Monte-Carlo is applicable to any circuit and needs no special algorithm beyond a transient simulation capability and some straightforward postprocessing.

In contrast to Monte-Carlo which is brute-force, several algorithms are based on using deterministic or stochastic models for noise. The algorithms differ in their computation and memory requirements and in the mathematical model used for noise. A common representation of noise is as a superposition of deterministic sinusoids, with powers computed at each frequency and added to produce the total noise power. This is equivalent to the assumption that the noise is stationary or that its frequency components are uncorrelated, which does not hold in general in nonlinear circuits. Hence naïve analyses using this approach can be incorrect, especially when phase relationships between signals in the circuit are important. However, careful analyses based on this model of noise can incorporate correlated frequency components either implicitly or explicitly. Such methods, effective for relatively small-sized circuits, have been developed in the microwave community (e.g., [21, 22]). Recently, these methods has been extended in [20] to compute average output noise power efficiently for large circuits with a single-tone large-signal drive. Another approach employs a stochastic noise model based on harmonic power spectral densities (HPSDs), instead of correlated frequency-domain components. While the two methods are fundamentally equivalent, the HPSD approach is more convenient theoretically and also provides a relatively intuitive way of visualizing nonlinear noise propagation correctly. The technique of [23], which is based on HPSD computations in the frequency domain, can handle large circuits excited by several strong tones and can also compute higher-order HPSDs, useful for system-level noise analysis. Yet another approach is based on computing time-varying autocorrelation functions and noise power directly in the time domain, by solving stochastic differential equations (SDEs) [24]. This method is more general than the previous ones in the sense that it can calculate noise that is truly non-stationary, i.e., in circuits with any, not necessarily periodic, large-signal excitation. It can also be applied without modification to calculating jitter in oscillators. However, it has been limited so far to relatively small circuits.

An issue peculiar to nonlinear (RF) noise analysis, as opposed to analysis about quiescent operating points (e.g., SPICE .noise), is that of modelling of device noises under time-varying biases. Thermal and shot noises under time-varying biases are generally modelled as stationary white noise sources modulated by deterministic bias-dependent factors. This mechanism is generally accepted as correct, since white noise has extremely short-term correlations. In contrast, the generation of bias-dependent flicker noise is not well understood. Since flicker noise, unlike white noise, has long-term correlations, it is widely believed that treating it as a stationary process multiplied by a relatively fast-varying deterministic bias-dependent term is not realistic. In spite of this, the model is currently used (e.g., [24, 23]) because of the lack of an alternative. At the moment, bias-dependent flicker noise modelling remains an open theoretical and experimental problem.

3.6.2 Phase noise

Analyzing phase noise in oscillators is more complex than calculating mixing noise. Many analyses are based on a system-level model of the feedback loop enabling oscillation, from which an expression for the frequency or phase is obtained. The variation in this expression caused by device noises provides an estimate of the phase noise. This approach is limited in its generality, since it requires higher-level knowledge about the oscillator's structure to be incorporated into the analysis.

If small-signal perturbational analysis, standardly employed for mixing noise, is applied to oscillatory circuits, the resulting perturbations grow large with time, i.e., the noise variance increases with time. This result is intuitive, since small changes in an oscillator's frequency lead, in time, to a waveform that is offset 180 degrees in phase to the noiseless signal, hence to large differences. The SDE-based method above is essentially a small-signal analysis that calculates the growing non-stationary perturbation directly as a function of time. The phase jitter can be extracted easily from this information.

The spreading of the oscillator's carrier tone can also be shown to result from the growth of the small-signal perturbation (e.g., [25]). Approaches for mixing noise [22, 26, 23] can be used to compute the asymptotic noise spectrum away from the carrier. However, these algorithms become increasingly inaccurate as they approach the carrier frequency. Recent extensions to the HPSD-based technique of [23] has enabled it to compute accurately the shape of phase-noise-induced carrier spreading, as well as the jitter, in large circuits with oscillatory components.

4 Commercial Tools, Features, and Limitations

In this section we cover some of the simulation capabilities that are offered today by a number of EDA companies. The survey will not be comprehensive. The emphasis will be placed on some of the tools that are more commonly used for transistor level RFIC simulations. LIBRA from HP-EEsof, spectreRF from Cadence Design Systems, Spectre/XL-RF from Avista Design Systems and Jitter from Design Aid Inc are among the simulators that we will present. The tools from Compact Software and HP-MDS have also been considered, but there are many similarities to the LIBRA software so they will not be discussed here. All of these simulators support the standard SPICE analyses: dc-operating point analysis, small-signal ac analysis and nonlinear time-domain analysis. Here we will focus only on the RF specific capabilities that these simulators provide.

4.1 LIBRA

The LIBRA simulation environment which is supported under HP- EEsof SERIES-IV domain, provides access to a number of simulation engines and algorithms through what is referred to as test-benches. LIBRA supports the following RF specific analysis types:

4.1.1 Nonlinear Steady-state Analysis

This is the analysis that distinguishes LIBRA from most SPICE-like simulators. The harmonic balance method is used to calculate the large signal steady-state response of non-linear circuits. Circuits containing both linear and non-linear elements can be analyzed, with up to three tones of excitation. In addition, LIBRA provides capabilities for large signal noise analysis, large signal S-parameter analysis, and oscillator analysis. In the latter two analyses, only one independent tone is allowed.

The major shortcoming of this analysis in LIBRA is the limitation on the size of the active part of the circuit. In the conventional harmonic balance implementation, the memory requirement grows rapidly as a function of the number of non-linear branches, number of nodes, number of tones and number of harmonics of the tones. In LIBRA it is very easy to use up tens of megabytes of memory with a circuit containing only a few transistors. For example a 4-transistor single-balanced mixer can take up to 40MB of memory with two input tones and 10 harmonics. Thus, it would be difficult to use it for designing some of the major sub-blocks in a RFIC system such as mixers, VCOs etc. Another disadvantage of this analysis is that for highly non-linear circuits like those with signal limiting (clamping), a large number of harmonics are needed in order to achieve a given level of accuracy, further restricting its use.

4.1.2 4.1.2 Nonlinear time-domain Analysis

LIBRA can perform SPICE-like transient analysis. The nonlinear devices are analyzed in the time domain. Distributed elements like transmission lines are handled using a convolution analysis [27]. Since many distributed and passive elements can best be modeled in frequency domain, the time domain analysis requires the frequency domain information to be converted into time-domain. As a consequence the impulse response will be calculated. More precisely, the time-domain waveforms at the input terminals are convolved with the impulse response of a frequency-dependent element to obtain the resulting waveforms at the output terminals. This is another distinguishing feature of LIBRA.

4.1.3 Electromagnetic Analysis

This unique algorithm allows for non-hierarchical (flat) transmission line analysis for components such as microstrips found on the printed circuit boards. A spatial-domain method is used to solve the quasi-TEM wave equations [28]. It produces multi-port S-parameter characteristics of a subnetwork, that can subsequently be used along with lumped device models and S-parameter data files in circuit simulation.

4.1.4 Circuit Envelope Analysis

HP-EEsof offers the first commercial implementation of the Envelope method. This analysis complements harmonic balance. While harmonic balance is used to analyze a circuit in its steady state, Circuit Envelope can be used to study transient effects, such as PLL phase noise before and after lock, oscillator turn-on amplitude and frequency versus time, mixer spectral regrowth due to either digital modulation on the RF carrier or signal noise on the carrier, amplifier harmonics versus time.

Because Circuit Envelope uses the conventional harmonic balance engine to perform its frequency-domain analysis, there is still a limit on the problem size it can handle. Nevertheless, since one of the tones is analyzed in time domain, the simulation run times are much better than using the pure harmonic balance approach. Specifically the size of the virtual memory requirement does not grow for intermodulation distortion analysis.

4.2 SpectreRF

SpectreRF is an add-on component of Spectre, a SPICE-like simulator from Cadence Design Systems. While the main focus in Libra is on frequency-domain algorithms (harmonic balance), SpectreRF extends the traditional time-domain algorithms to handle RFIC simulation.

In addition to the common SPICE-like analyses, SpectreRF supports the following analyses:

4.2.1 Periodic Steady-State Analysis (PSS)

SpectreRF uses the Newton shooting method to calculate the periodic steady-state response of circuits. It can handle circuits with strong nonlinearities efficiently.

Although SpectreRF can handle larger circuits than LIBRA, its run time and memory requirement go up rapidly with circuit size. It cannot handle distributed components such as transmission lines. This precludes the use of the PSS analysis with N-port models represented by measured S-parameters. Also, mixer-type circuits with multiple independent large signal tones cannot be handled efficiently by SpectreRF, due to the nature of the shooting method.

4.2.2 Small-signal Periodic Analysis

The periodic AC (PAC) analysis in SpectreRF is the linear time varying analysis described in Section 3.5 which was referred to as LTV. SpectreRF linearizes the circuit at the periodically varying operating point computed using the PSS analysis for the large-signal input. The small input or noise signals are ignored and only the large-signal response is calculated. Once the PSS is known, any number of periodic small-signal AC analyses which include the small input and noise can be performed. Note however that since the AC analysis is purely linear, it is up to the user to ensure that the input amplitudes are small enough to make the linearity assumption work.

The major advantage of the periodic AC analysis is the fact that it provides results that show the mixing of the small-signal with the harmonics of the large signal. Thus the conversion gain for mixers can be calculated.

The LTV or PAC analysis in SpectreRF can also be used to calculate IM3 as described in [29]. One (two for mixers) large tone is applied and used for PSS analysis and the assumption here is that this tone drives the circuit non-linearly and generates distortion products. A second small tone is used for PAC analysis. This latter tone is used as a test signal to measure the intermodulation distortion. So if V_l(dB) is the fundamental singal due to the large input, and V_{s1}(dB) is the fundamental due to the small tone and is the upper sideband of V_l, and V_{s3}(dB) is the lower sideband of V_l, then the third order intercept point can be defined as:

$$\mathrm{IP3} = V_l - \frac{V_{s3} + V_{s1}}{2}$$

Note that this number may be slightly different from that obtained by applying two large-signal tones since the two large signals drive the non-linearities of the devices differently. This approach can be considered a modified definition for IP3 (third order intercept point) calculations.

4.3 Spectre/XL-RF

Spectre/XL-RF from Avista Design Systems is a simulator for RF microwave circuits that embeds the Spectre simulation engine [30] within the Excel spreadsheet for doing what-if analyses. It provides accelerated steady-state calculation using the harmonic balance method. A large-signal/small-signal analysis (LTV as in spectreRF) is available for frequency translation whereby signal up and down conversion can be accounted for. This provides the basis for the noise analysis of mixers and phase noise for oscillators. In addition, S-parameters can be determined for multi-port networks. Spectre/XL-RF also calculates stability and gain circles for use with the Smith chart. Both forced and oscillatory circuits with distributed elements can be simulated.

4.4 Jitter

The Jitter program from Design Aid Inc. simulates noise at the transistor level directly in the time domain. Jitter is a non-Monte Carlo simulator that is fast and accurately incorporates time-varying noise effects. It can be used for a wide variety of circuits including mixers, oscillators, and switched-capacitor networks. The output from the simulator are noise variances, jitter or phase noise and various data display options are available. In the default mode Jitter is run in conjunction with SPICE3 but it can be interfaced to any circuit simulator. This is possible since the noise simulation is done as a post processing step after a conventional transient analysis has been completed.

5 Discussion and Conclusions

In this paper, we provided an overview of the circuit simulation needs for RFIC design and techniques currently available for addressing some of these needs. System simulation and behavioral modeling are useful early in the design phase but circuit simulation is indispensable. The design of an amplifier or mixer with stringent noise and IP3 specifications or a PLL with low phase jitter specification requires detailed circuit-level simulation.

Exciting advances in circuit simulation, as outlined in this paper, are coming at an increasingly faster rate. Periodic steady state analysis, multi-tone harmonic balance, mixed frequency-time methods and envelope methods promise faster simulation. Techniques for simulating noise in a nonlinear circuit are also appearing. We expect these advances will fuel the research into other difficult problems such as the simulation of delta-sigma data converters. Commercial tools incorporating some of these new techniques are available and are useful, provided the limitations of the tools are well understood by the designer.

While advances in simulation are occurring, RF and high-speed mixed-signal design continues to be a black art practiced by a few highly skilled artisans. Analog and RF circuit engineers work with incomplete simulations or simulations that take days or weeks to complete. Exceptional design engineers can create first pass silicon in this environment because of excellent analytical skills, experience with previous circuits, clever design and good intuition. Less experienced designers are forced to run revision after revision of silicon often having to abandon their projects. Researchers and tool developers have years of challenging work ahead of them to improve this environment so that it is at par with what digital designers have today.

Much work remains to be done. Complete simulation of an RF front end including the LNA, mixer, IF strip and frequency synthesizer is still far away, especially if effects of the layout, substrate coupling and the package are to be considered. Designers of disk drive read channels, high-end audio CODECs, wide and local area network front ends would also prefer an environment in which full chip circuit simulation is available. In addition, the loop closure between requirements determined by the system designer using block diagram based CAD tools and the circuit designer remains virtually unexplored.

Acknowledgements

The authors thank members of the analog technical program subcommittee of CICC-97 for their inputs. D.C. Lee thanks David Long for many discussions on matters large and small. J. Roychowdhury would like to thank Peter Feldmann, David Long, Alper Demir and Laszlo Toth for useful discussions on envelope following and noise.

References

[1] B. Razavi. Challenges in portable RF transceiver design. *IEEE Circuits and Devices*, (5):12–25, September 1996.

[2] K.S. Kundert. *The Designer's Guide to SPICE & SPECTRE.* Kluwer Academic Publishers, 1995.

[3] K. Kundert. Accurate Fourier analysis for circuit simulators. In *Proc. IEEE CICC*, pages 25–28, May 1994.

[4] J.S. Roychowdhury. SPICE3 Distortion Analysis. Master's thesis, EECS Dept., Univ. Calif. Berkeley, Elec. Res. Lab., April 1989. Memorandum no. UCB/ERL M89/48.

[5] C. McAndrew, B. Bhattacharyya, and O. Wing. A Single-Piece C_∞-Continuous MOSFET Model Including Subthreshold Conduction. *IEEE Elect. Device Letters*, 12(10):565–567, October 1991.

[6] S. Kapur, D. Long, and J. Roychowdhury. Efficient time-domain simulation of frequency-dependent elements. In *Proc. ICCAD*, November 1996.

[7] S. Lin and E.S. Kuh. Transient Simulation of Lossy Interconnects Based on the Recursive Convolution Formulation. *IEEE Trans. Ckts. Syst. – I: Fund. Th. Appl.*, 39(11), November 1992.

[8] J.S. Roychowdhury, A.R. Newton, and D.O. Pederson. Algorithms for the Transient Simulation of Lossy Interconnect. *IEEE Trans. CAD*, 13(1):96–104, January 1994.

[9] P.N. Ashar. Implementation of algorithms for the periodic-steady-state analysis of nonlinear circuits. Master's thesis, EECS Dept., Univ. Calif. Berkeley, Elec. Res. Lab., March 1989. Memorandum no. UCB/ERL M89/31.

[10] R. Telichevesky, K. Kundert, and J. White. Efficient Steady-State Analysis based on Matrix-Free Krylov Subspace Methods. In *Proc. IEEE DAC*, pages 480–484, 1995.

[11] K.S. Kundert, J.K. White, and A. Sangiovanni-Vincentelli. *Steady-state methods for simulating analog and microwave circuits.* Kluwer Academic Publishers, 1990.

[12] R.J. Gilmore and M.B. Steer. Nonlinear circuit analysis using the method of harmonic balance – a review of the art. Part I. Introductory concepts. *Int. J. on Microwave and Millimeter Wave CAE*, 1(1), 1991.

[13] H.G. Brachtendorf, G. Welsch, and R. Laur. A simulation tool for the analysis and verification of the steady state of circuit designs. *Int. J. of Circuit Theory and Applications*, 23:311–323, 1995.

[14] R.C. Melville, P. Feldmann, and J. Roychowdhury. Efficient multi-tone distortion analysis of analog integrated circuits. In *Proc. IEEE CICC*, May 1995.

[15] K. Kundert, J. White, and A. Sangiovanni-Vincentelli. A Mixed Frequency-Time Approach for Distortion Analysis of Switching Filter Circuits. *IEEE J. Solid-State Ckts.*, 24(2):443–451, April 1989.

[16] J. Roychowdhury. Time-domain and mixed frequency-time algorithms for strongly nonlinear circuits with multi-tone excitations. Submitted for publication.

[17] D. Sharrit. New Method of Analysis of Communication Systems. *MTTS WMFA: Nonlinear CAD Workshop*, June 1996.

[18] P. Feldmann and J. Roychowdhury. Computation of circuit waveform envelopes using an efficient, matrix-decomposed harmonic balance algorithm. In *Proc. ICCAD*, November 1996.

[19] P. Feldmann, R.C. Melville, and D. Long. Efficient Frequency Domain Analysis of Large Nonlinear Analog Circuits. In *Proc. IEEE CICC*, May 1996.

[20] R. Telichevesky, K. Kundert, and J. White. Efficient AC and Noise Analysis of Two-Tone RF Circuits. In *Proc. IEEE DAC*, pages 292–297, 1996.

[21] A.R. Kerr. Noise and loss in balanced and subharmonically pumped mixers: Part 1 – Theory. *IEEE Trans. MTT*, MTT-27:938–943, December 1979.

[22] V. Rizzoli, F. Mastri, and D. Masotti. General Noise Analysis of Nonlinear Microwave Circuits by the Piecewise Harmonic-Balance Technique. *IEEE Trans. MTT*, 42(5):807–819, May 1994.

[23] J. Roychowdhury, D. Long, and P. Feldmann. Cyclostationary noise computation for large RF circuits with multitone excitations. Submitted for publication.

[24] A. Demir, E. Liu, and A. Sangiovanni-Vincentelli. Time-Domain Non Monte-Carlo Noise Simulation for Nonlinear Dynamic Circuits with Arbitrary Excitations . *IEEE Transactions on Computer-Aided Design of Integrated Circuits*, 15(5):493–505, May 1996.

[25] F.X. Kärtner. Analysis of white and $f^{-\alpha}$ noise in oscillators. *Int. J. Ckt. Th. Appl*, 18:485–519, 1990.

[26] W. Anzill and P. Russer. A General Method to Simulate Noise in Oscillators Based on Frequency Domain Techniques. *IEEE Trans. MTT*, 41(12):2256–2263, December 1993.

[27] H.M. Srivastava. *Theory and applications of convolution integral equations*. Kluwer Academic Publishers, 1992.

[28] R.F. Harrington. *Field Computation by Moment Methods*. MacMillan, 1968.

[29] R. Telichevesky, K. Kundert, and J. White. Receiver Characterization using Periodic Small-Signal Analysis. In *Proc. IEEE CICC*, pages 449–452, 1996.

[30] K.S. Kundert. *Spectre User's Guide: A Frequency Domain Simulator for Nonline ar Circuits*. University of California, Berkeley, EECS Industrial Liaison Program, University of California, Berkeley California, 94720, April 1987.