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

Kartikeya Mayaram, *Senior Member, IEEE*, David C. Lee, *Member, IEEE*, Shahriar Moinian, David A. Rich, and Jaijeet Roychowdhury

***Abstract*—The design of the radio frequency (RF) section in a communication integrated circuit (IC) is a challenging problem. Although several computer-aided analysis tools are available for RFIC design, they are not effectively used, because there is a lack of understanding about their features and limitations. These tools provide fast simulation of RFIC's. However, no single tool delivers a complete solution for RFIC design. This paper describes the shortcomings of conventional SPICE-like simulators and the analyses required for RF applications with an emphasis on accurate and efficient simulation of distortion and noise. Various analysis methods, such as harmonic balance, shooting method, mixed frequency-time methods, and envelope methods, that are currently available for RFIC simulation are presented. Commercial simulators are compared in terms of their functionalities and limitations. The key algorithmic features and the simulator-specific terminology are described.**

***Index Terms*—Circuit simulation, cyclostationary noise, distortion, envelope method, frequency-domain methods, harmonic distortion, intermodulation, linear time-varying analysis, mixed frequency-time methods, mixer noise, noise, periodic steady-state, phase noise, quasiperiodic steady-state, RFIC simulation, SPICE harmonic balance, shooting method, time-domain methods.**

I. Introduction

THE wireless/personal communication electronics market is growing at a rapid pace with a drive toward complete systems on a chip. Although only a small section of the system operates at radio 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 fully support radio freqency integrated circuit (RFIC) design [1]. Although several commercial tools are available, a lack of understanding of their features and limitations also contributes to this problem.

For the design of a typical radio frequency section of a circuit, one would like to simulate basic blocks such as amplifiers, mixers, oscillators, voltage-controlled oscillators (VCOs), and phase-locked loops (PLL's). Because of the signal frequencies and the circuit time-constants, and the types of analyses required, simulators such as SPICE are neither adequate nor appropriate. SPICE is used to simulate some of these basic blocks by running long transient analyses and 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 designers. The features required for RFIC simulations are:

1) rapid simulation of the periodic or quasiperiodic steady state of a circuit (Fig. 1);
2) accurate simulation of harmonic and intermodulation distortion in order to determine gain compression and intercept points;
3) simulation of turn-on transients of oscillators (Fig. 1) and the capture process of PLL's;
4) simulation of up and down conversion of noise caused by circuit nonlinearities;
5) simulation of phase noise/jitter for oscillators and PLL's;
6) simulation of distributed elements.

In this paper, we examine these issues from a designer's perspective. We present the basic concepts, whereas mathematical details can be found in [2] or the other cited references. In Section II, we first describe the analyses that are available in SPICE and then explain the limitations of a SPICE-like simulator for RFIC simulation. This is followed in, Section III, 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 IV. Finally, a brief discussion and conclusions are presented in Section V.

II. SPICE Features and Limitations

The circuit simulator SPICE [3], [4] supports various analyses which can be classified as dc (.op, .dc), small-signal ac (.ac, .noise, .disto), and transient (.tran, .four). A summary of these analyses and the types of circuits that they can be used for is provided in Table I. 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, frequency translation of noise due to the circuit nonlinearities cannot be directly determined in SPICE. Thus, traditional SPICE analyses cannot be used for estimating noise in mixers or phase noise/jitter in

Manuscript received December 1998; revised April 1999. The work of K. Mayaram was supported in part by the National Science Foundation under Grant CCR-9702292. This paper was recommended by Associate Editor A. Ushida.

K. Mayaram is with the Department of Electrical and Computer Engineering, Oregon State University, Corvallis, OR 97331 USA.

D. C. Lee was with Lucent Bell Laboratories, Allentown, PA 18103. He is now with Mentor Graphics Corporation, Allentown, PA 18103 USA.

S. Moinian is with Lucent Bell Laboratories, Reading, PA 19162 USA.

D. A. Rich is with Lucent Bell Laboratories, Allentown, PA 18103 USA.

J. Roychowdhury is with Lucent Bell Laboratories, Murray Hill, NJ 07974 USA.

Publisher Item Identifier S 1057-7130(00)03133-5.

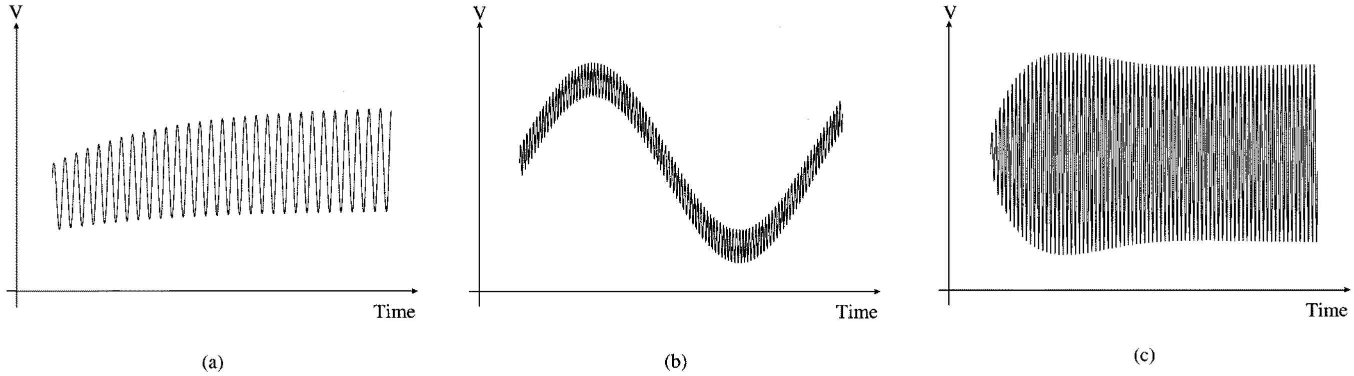


Fig. 1. Simulation of typical RF circuits. (a) Amplifier response showing the initial transient and the periodic steady state. (b) Mixer quasi-periodic steady-state. (c) Oscillator response showing buildup, overshoot, and the final periodic steady-state.

TABLE I
SUMMARY OF ANALYSIS METHODS AVAILABLE IN SPICE-LIKE SIMULATORS AND THE TYPES OF CIRCUITS THAT CAN BE SIMULATED

Analysis	Command	Description	Circuits
DC	.op	Operating point	All circuits
	.dc	Transfer curves	All circuits
AC	.ac	Small-signal AC analysis	Linear circuits, Amplifiers
	.noise	Small-signal noise analysis	Amplifiers
	.disto	Intermodulation distortion	Mildly nonlinear circuits
Transient	.tran	Large-signal time-domain analysis	Amplifiers, Mixers, Oscillators, PLLs, Switched-capacitor circuits, A/D, D/A converters
	.four	Frequency spectrum	Circuits with periodic response

oscillators. Special techniques have been developed for mixer noise [5] and oscillator phase noise [6] analyses. Although these are based on SPICE transient analyses, the designer has to go through several simulation and postprocessing steps that make these methods cumbersome to use. Furthermore, these techniques are applicable only to low-Q circuits where the periodic steady-state solution can be easily simulated in SPICE. For this reason, an enhanced noise analysis is required as described in Section III-G.

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.[1]

Determination of distortion using transient analysis can result in several errors. An excellent discussion of these can be found in [7]. First, the circuit must be in a periodic steady-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 that must be used 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 [7], [8]. The last error is important in a circuit simulator that uses variable timesteps in the transient analysis. Since the DFT can be applied only to waveforms that are uniformly sampled, the nonuniformly spaced data produced by the simulator must be first interpolated onto a uniform grid. This can result in significant errors when the interpolation order is low.

The .four command 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.[2] IMD is determined by simulating the circuit with two closely spaced tones at frequencies f_1 and f_2. For example, consider $f_1 = 1$ GHz and $f_2 = 1.0001$ GHz. The third-order IM terms (IM3) are at $2f_1 - f_2 = 999.9$ MHz and $2f_2 - f_1 = 1.0002$ GHz. The slowest varying component is at $f_2 - f_1 = 100$ kHz, while another component of interest is at a much higher frequency of $2f_2 - f_1 = 1.0002$ GHz. Using a conventional transient analysis, at least one period of the slow 100-kHz component must be simulated. To resolve the fastest signal, the maximum timestep must be smaller than half the period corresponding to 1.0002 GHz. In this case, the ratio of the fastest to slowest frequencies is more than 10 000 : 1. This results in an extremely slow transient analysis, since a large number of time points have to be simulated. The problem is much worse when the periodic steady-state is reached after many cycles of the slowly varying signal as would be the case in a high-Q circuit. As another example, consider the simulation of high-Q oscillators. These also require very long transient simulations; a Q of 10 000 suggests that the turn-on transient time starting from a zero initial state will be of the order of 10 000 cycles of the oscillation period.

Numerical errors introduced by the simulator makes resolving small intermodulation products difficult. When distortion levels are low, the Volterra series method [9] can be used to determine IM3 accurately using three small-signal ac analyses [10], [11]. Thus, accurate estimation of IM3 is possible for a relatively small computational cost. The *.disto* command in SPICE allows accurate calculation of IM3 using the Volterra series method. Although SPICE2G6 supports

[1]Only nine harmonics are reported in SPICE2G6. However, there is no such restriction in SPICE3.

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

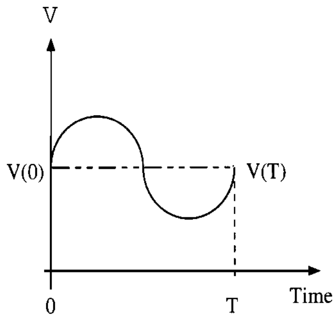


Fig. 2. The periodicity constraint applied in the time-domain method for determining the periodic steady-state solution.

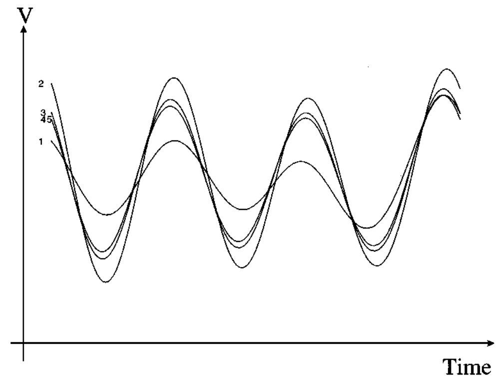


Fig. 3. Time-domain waveforms for five iterations of the Newton shooting method for determining the periodic steady-state.

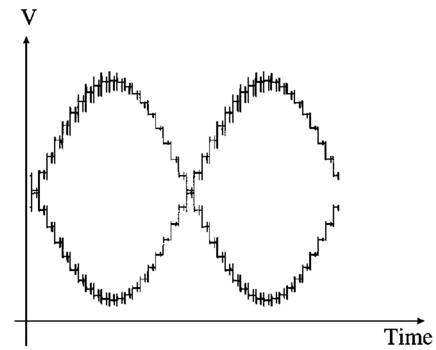


Fig. 4. Periodic steady-state time-domain output voltage waveforms for a fifth-order elliptic CMOS switched-capacitor filter.

.disto, the implementation may give erroneous results and has been corrected in SPICE3 [11].

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, accurate models (e.g., [12] and [13]) and careful ac/dc characterization are essential for RF applications.

Another problem associated with the 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 [14]–[16]. The only distributed element available in SPICE is the transmission line.

III. Algorithms for RF Simulation

As described in Section II, conventional SPICE analyses are not adequate for many RF needs. Special techniques have been developed that specifically address the simulation of RF circuits. 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. Most of the currently available methods are described in this section.

A. 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 (Fig. 2). $V(0)$ is the vector of initial node voltages at time zero (i.e., the initial conditions), and $v(T)$ is the vector of final 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 nonoscillatory 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 was initially proposed by [17] for forced circuits and later extended to oscillators [18]. Furthermore, this method has been shown to be reliable in the public domain simulator SSpice [19] 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. 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 correction 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 $v(0) = v(T)$. In Fig. 3, the waveforms of $v(t)$ for five Newton iterations are shown and illustrate how the method converges to the periodic steady-state solution for a frequency multiplier. 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 [7].

In the above process, dense matrix manipulations are required for determining the solution. This places a restriction on the

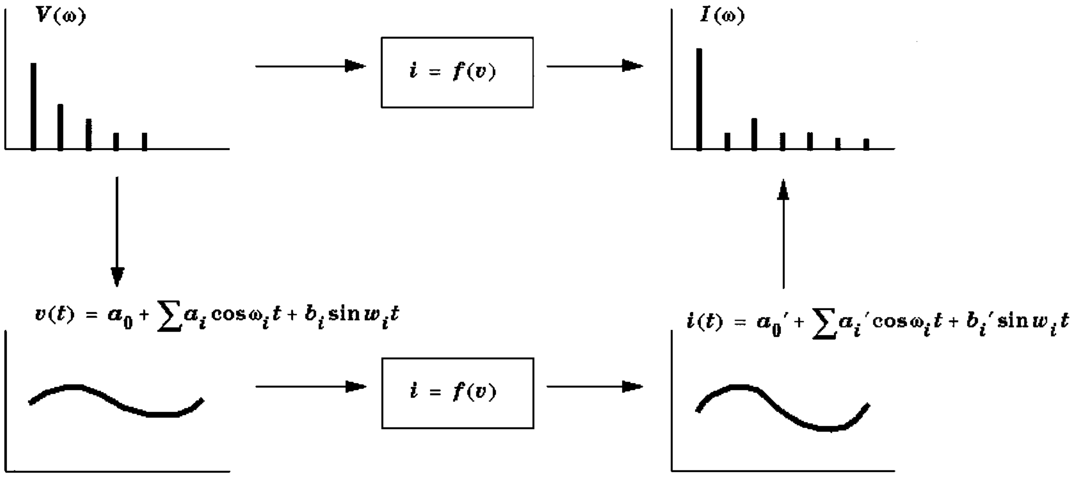


Fig. 5. Nonlinear frequency-domain analysis. The time-domain representations are used for determining the frequency spectrum of the output current for an applied periodic voltage.

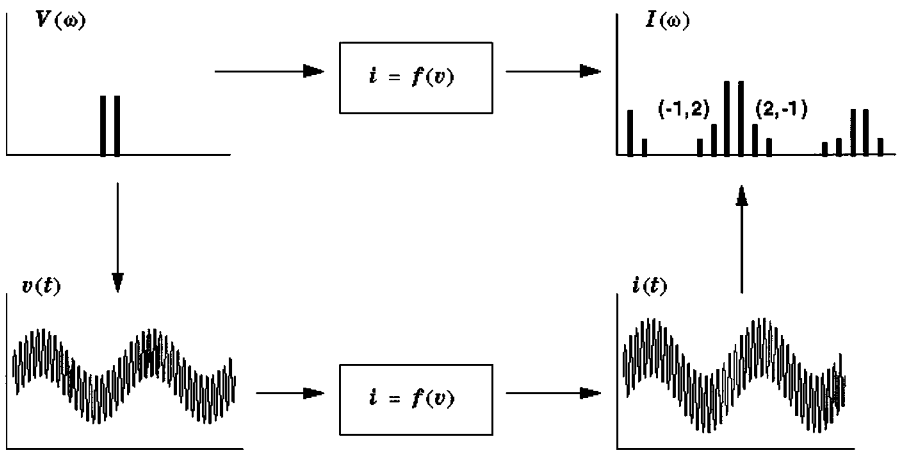


Fig. 6. Nonlinear frequency-domain analysis for two-tone signals. The time-domain representations are used for determining the frequency spectrum of the output current for two applied periodic voltages.

maximum size of the circuit that can be simulated as approximately 300 nodes. A recent method [20] overcomes these limitations allowing circuits with thousands of nodes to be simulated by use of efficient matrix solution algorithms. The periodic steady-state output voltage waveforms for a fifth-order CMOS switched-capacitor elliptic filter, as computed using the Newton shooting method in the Celerity simulator developed at Bell Labs, are shown in Fig. 4. It can be seen that the glitches that arise due to the clocked switches are captured in these simulations, indicating that the method is well suited for simulating highly nonlinear behavior.

As with any time-domain simulation method, accurate distortion calculation requires care in selecting simulation tolerances and the algorithm used for computing the harmonics. The Fourier integral method coupled with the time-domain shooting method with appropriate tolerances provides accurate distortion calculation [7].

Time-domain methods are however not well suited for multitone 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. When the RF signal is small then the linear time-varying method described in Section III-E can be used.

B. Harmonic Balance (HB) Method

The HB method [21]–[27] is a well-established frequency-domain technique for periodic and quasi-periodic steady-state analysis of mildly nonlinear circuits. It is used to analyze distortion and transfer characteristics of amplifiers, mixers, and oscillators.

Before describing the HB algorithm, we first explain the concept of nonlinear frequency-domain analysis. Consider a nonlinear resistor described by a current-voltage characteristic $i = f(v)$. Given a periodic voltage excitation with a frequency spectrum $V(\omega)$, we would like to find the frequency spectrum $I(\omega)$ of the resulting current, when the circuit is in its periodic steady-state. This is illustrated in the upper part of Fig. 5.

Since all nonlinear device models for SPICE-like circuit simulators are described in the time domain, the current-voltage calculations are performed in the time domain. $V(\omega)$ is first transformed to a time-domain signal $v(t)$ through its Fourier series representation

$$v(t) = a_0 + \sum_i [a_i \cos(\omega_i t) + b_i \sin(\omega_i t)], \quad \omega_i = i\omega_0$$

where ω_0 is the fundamental frequency of the input signal. Then the time-domain waveform is applied to the nonlinear resistor and the current waveform is determined in the time domain. The time-domain current waveform is then transformed to the frequency domain by use of a Fourier transform. The complete procedure is illustrated in Fig. 5.

When the circuit is excited by multitone signals, the above process works in a similar manner, except that a more general Fourier series representation is assumed for each signal. In this case, the above equation is used with ω_i defined to be the multiples of the excitation frequencies and the sums and differences of all these frequencies. The Fourier coefficients a_i and b_i, in this case, characterize harmonic and intermodulation distortions. The calculation procedure for a two-tone problem is illustrated in Fig. 6.

In the HB method, the equations are solved in the frequency domain. The key idea is the application of KCL at each node, assuming a nodal formulation is used. The frequency spectrum of all the currents at a node is balanced, i.e., KCL is applied for each independent frequency. This is illustrated schematically in Fig. 7 where a nonlinear diode is connected to linear elements at a circuit node.

In a circuit simulator, the HB method is formulated by expressing the circuit differential equations in terms of the Fourier coefficients, 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 method.

Most implementations of the HB method require the manipulation of relatively dense matrices, consuming excessive CPU times and large amount of memory for even medium-sized circuits. Recent work [28]–[31] has incorporated efficient matrix algorithms that extend HB 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.

The HB method has several salient features, as summarized below.

1) 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 HB is dictated by the number of Fourier coefficients, and not by the Nyquist rate as in the conventional transient analysis. This makes HB particularly attractive for circuits driven by multitone excitations.
2) HB is very accurate and fast when all signals in the steady-state solution can be approximated using a small number of Fourier coefficients. Examples include a nearly linear (low distortion) circuit or a weak stimulus applied to a nonlinear circuit. Naturally, when the circuit response is strongly nonlinear for any signal, 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.
3) Frequency-dependent distributed elements are handled with no difficulty.
4) Parameter sweep (distortion and/or gain compression versus 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.

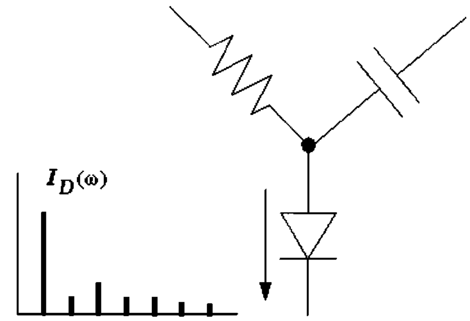


Fig. 7. Application of KCL in the frequency domain at a circuit node.

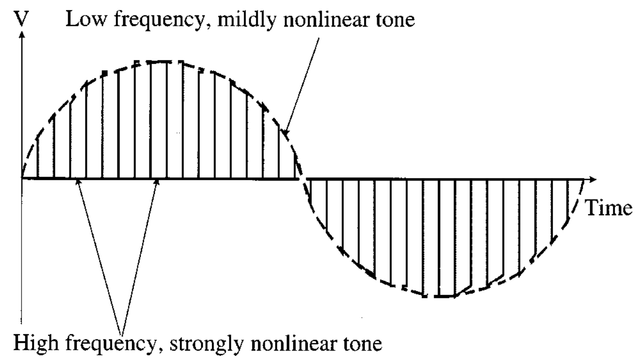


Fig. 8. A typical waveform that can be efficiently simulated using a mixed frequency-time method.

Oscillators are handled by treating the oscillation frequency as an unknown in the system of nonlinear equations, and setting one of the Fourier coefficients of one signal to zero.

C. Mixed Frequency-Time Methods

For a special class of multitone circuits, *mixed frequency-time* approaches are superior to both time and frequency-domain techniques. Two broad classes of methods, exemplified by [32] and [33] and the more recent [34]–[36] are available, differing in equation formulation and in numerical properties. The method of [33] 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. The methods in the second class are applicable to 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 these methods. An example waveform is shown in Fig. 8, where the slowly varying sinusoidal signal, the periodic envelope, is of interest. This envelope is determined in an efficient manner by use of the mixed frequency-time methods.

The method of [33] works by integrating the circuit's differential equations in the time domain for a *few* isolated and evenly distributed periods of the fast, strongly nonlinear tone. 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 multitone 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 algorithms of [36] and [37] are based on formulating the multitone problem as a partial differential equation (PDE) in multiple time scales. 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.

D. Envelope 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 as shown in Fig. 9. Often, the information signal is not periodic or 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.

The first method developed for solving this class of problems in the time domain is known as the *envelope-following method* [38]. This method has been used for the efficient transient simulation of switching circuits in [39]. When a waveform, such as the one in Fig. 9, is sampled periodically at the carrier frequency the discrete samples often change slowly and trace out a time varying envelope. If a low order polynominal can be used to approximate this envelope, then the change in the discrete samples several periods apart can be linked to the change in the discrete samples one period apart by means of a differential like equation. The overall envelope equation is solved using a Newton shooting method, while a SPICE-like transient analysis is used in the inner loop to simulate the circuit in detail over one isolated period. In this manner, a conventional transient analysis is performed only for a few isolated periods of the fast carrier

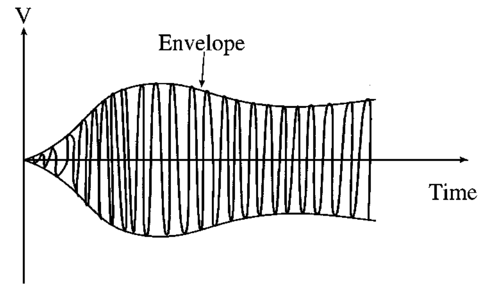


Fig. 9. A typical envelope that can be efficiently simulated using an envelope method.

which leads to the efficiency of this method. The number of periods that are skipped is controlled based on a local truncation error estimate.

A new set of "Fourier envelope" algorithms referred to as the *circuit envelope method*, *transient envelope method* or *modulation-oriented HB* have been developed to efficiently and accurately solve for the envelope using a combination of time-domain and frequency-domain techniques [40]–[43]. These algorithms operate at the time scales of the slowly varying envelope, and are relatively insensitive to the rate of the high frequency carrier. 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 using HB in the inner loop. Envelope methods can be used to simulate the turn on transient response of amplifiers, and the response of power amplifiers to digitally modulated RF signals. With the recent development of algorithms based on partial differential equations [36], [37], the inner-loop steady-state problem can also be addressed in the time domain, like the envelope-following method. This has made envelope simulation possible for a broader class of highly nonlinear circuits, including chopping power converters, digital PLL's, and pulsewidth-modulated communication circuitry [44].

As an example, consider the block diagram of Fig. 10 for an automatic gain control for a circuit that generates the quadrature signals for an in phase/quadrature phase (I/Q) modulation scheme. The simulated envelopes for the I/Q signals using the method of [43] are also shown in this figure. This simulation allows the designer to determine the stability of the control loop without resorting to very long transient simulations.

E. Linear Time-Varying Analysis

The linear time-varying (LTV) 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 analysis can be performed on circuits that exhibit frequency translation, such as mixers and switched-capacitor circuits.

As an example, consider a mixer represented by an ideal multiplier with two input ports, the RF and the LO, and one output port. This circuit is driven by two tones, the LO which is a large signal, and the RF which is a small signal. The large-signal response is determined by applying the large-signal excitation, and computing the steady-state response using HB or the

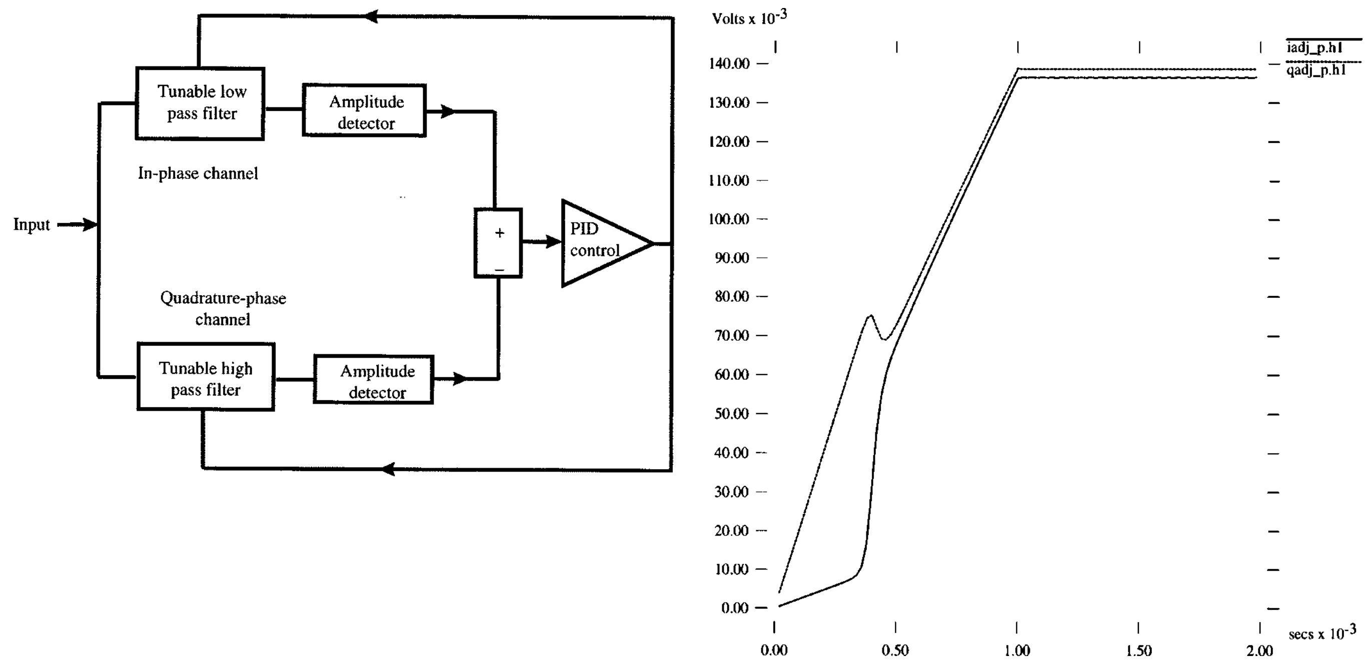


Fig. 10. An example of the application of the circuit envelope method. The block diagram of the AGC circuit and the simulated envelope waveforms are shown.

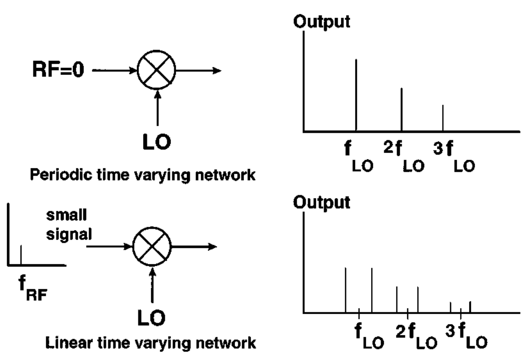


Fig. 11. The application of linear time-varying analysis to an ideal mixer example. The circuit is first simulated with the large-signal LO only and then the small-signal RF is applied.

time-domain shooting method described earlier (see upper half of Fig. 11). Linearizing the circuit about its periodic operating point results in a linear periodic time-varying circuit. Then the small-signal excitation is applied to the linear time-varying circuit and the complete response is obtained. The complete procedure is shown schematically in Fig. 11. In the literature, this method is known by a number of names including conversion matrix analysis, large-/small-signal analysis [27], mixer analysis, periodic ac analysis, or heterodyne ac analysis. An implementation of this method in conjunction with the time-domain shooting method is described in [45], and has recently been extended to large circuits in [46].

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:

1) the small-signal excitation does not perturb the time-varying operating point of the circuit;

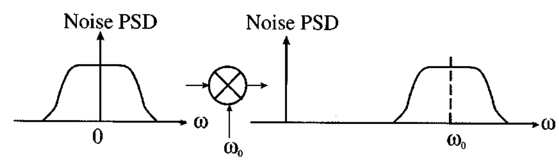


Fig. 12. Frequency translation of noise in a nonlinear circuit. An ideal mixer is used as an example.

2) 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 small-signal conversion gain for mixers can be calculated.

F. Summary of Periodic Steady-State Analysis Methods

The capabilities of the various analysis methods described above can be summarized as in Table II. In this table, the different methods and the circuits for which they are suited are shown. It is clear that none of the methods is general enough for all classes of problems that have to be solved for RFIC design.

G. RF Noise

Estimation of electrical noise is important in the design of RF circuits, since noise determines critical specifications like signal-to-noise ratio (SNR) and bit-error rate (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 as shown in Fig. 12, whereas the latter do not. Hence, conventional noise analysis in SPICE, which is based on stationary stochastic noise, does not suffice

TABLE II
SUMMARY OF VARIOUS PERIODIC STEADY-STATE ANALYSIS METHODS AND THE CIRCUIT APPLICATIONS FOR WHICH THEY ARE BEST SUITED

Circuit	Time-Domain Methods	Frequency-Domain Methods	Mixed Time Freq. Methods	Envelope Method	Linear Time Varying Method
Amplifiers (Mildly Nonlinear)	√	√			
Amplifiers (Highly Nonlinear)	√				
Mixers (Commensurate Tones)	√	√	√		√[3]
Mixers (Non-Commensurate Tones)		√	√		√[3]
Oscillators	√	√		√	
PLLs, AGCs				√	
Switched-Capacitor Circuits	√		√		√[3]

[3]Only a small-signal input can be applied.

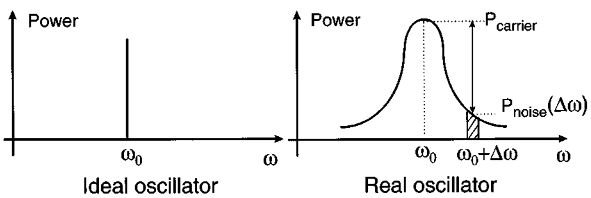


Fig. 13. Frequency spreading of the ideal frequency response of an oscillator due to phase noise.

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 nonoscillatory 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 (Fig. 13). 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 significant 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.

Mixing Noise: The Monte Carlo approach for noise simulation is popular [47]–[49]. 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. Furthermore, if low-frequency noise behavior is of interest, then a very long transient simulation is required. This makes the method even computationally more expensive, since the simulation interval must be hundreds to thousands times longer than the period of the lowest frequency noise. 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, a Monte Carlo approach is applicable to any circuit, needs no special algorithm beyond a transient simulation followed by some postprocessing, and gives results directly in the time domain.

In contrast to Monte Carlo simulation,which is a brute-force approach, 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 misleading, 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., [50], [51]). In [52], a time-domain technique was developed to compute the average output noise power for small circuits with a single-tone large-signal excitation. Recently, this method has been extended in [46] to handle large circuits efficiently.

Another approach employs a stochastic noise model based on cyclic spectral densities (CSD's) [53] or harmonic power spectral densities (HPSD's) [54], instead of correlated frequency-domain components. While the two methods are fundamentally equivalent, the CSD/HPSD approach is more convenient theoretically and also provides a relatively intuitive way of visualizing nonlinear noise propagation correctly. The technique of [54], 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 HPSD's, useful for system-

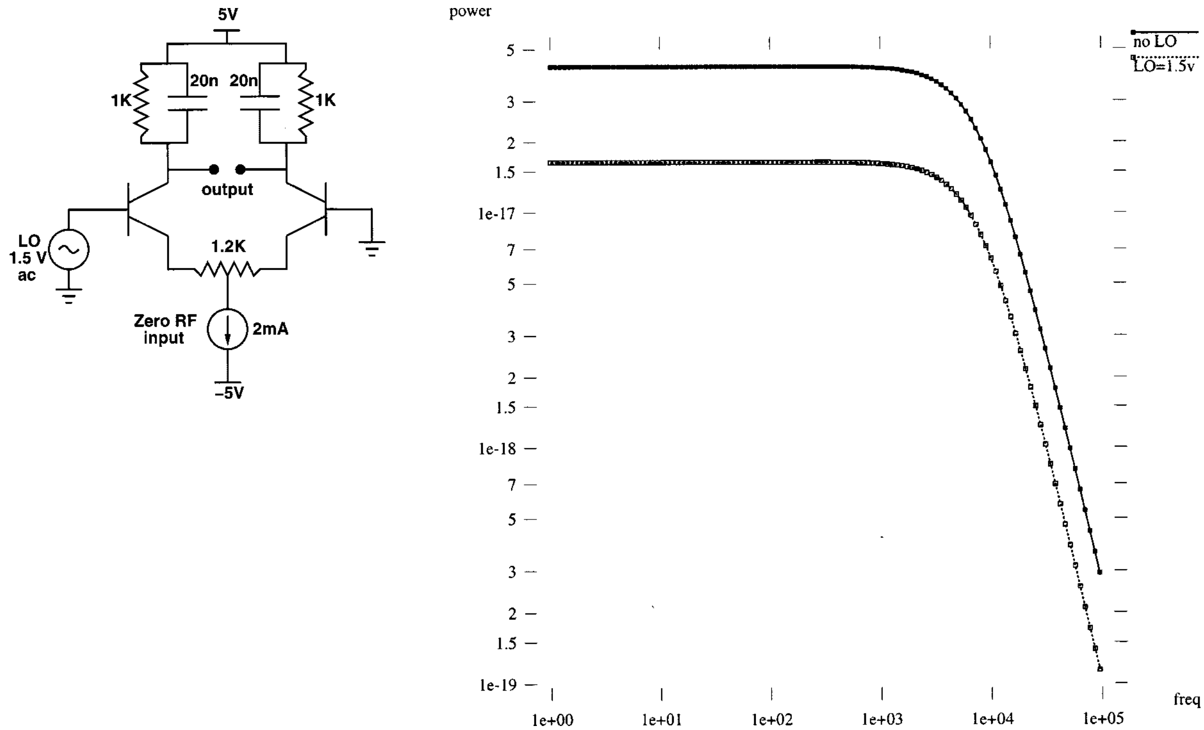


Fig. 14. Stationary noise power spectral densities for a switching mixer with and without the LO. The output noise reduces when the LO is included in the simulation.

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 (SDE's) [55]. This method is more general than the previous ones in the sense that it can calculate noise that is truly nonstationary, 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.

As an example of a noise simulation using the technique of [54], consider the switching mixer circuit of Fig. 14. The noise simulation results are also shown in this figure and it is seen that the noise power is reduced when an LO amplitude of 1.5 V is included in the simulation. This is to be expected, because the transistors are switched from an *on* to an *off* state, and an *off* transistor does not contribute to the output noise. On the other hand, a noise simulation that does not include the LO predicts higher noise because both transistors are on at all times.

The output noise voltage spectrum of a complete upconversion mixer as computed using the HB method in the Celerity simulator is shown in Fig. 15. With just the LO turned on, the noise performance is acceptable. However, the presence of a strong input signal can inadvertently activate supposedly noise-free bias circuitry. This significantly worsens the mixer noise performance as seen in Fig. 15. For this reason, complete verification of a mixer often requires noise simulations that include bias circuits, all parasitic and package effects, and strong input signals.

An issue peculiar to nonlinear (RF) noise analysis, as opposed to analysis about quiescent operating points (e.g., SPICE .noise) is that of modeling device noise sources under time-varying biases. Thermal and shot noises under time-varying biases are

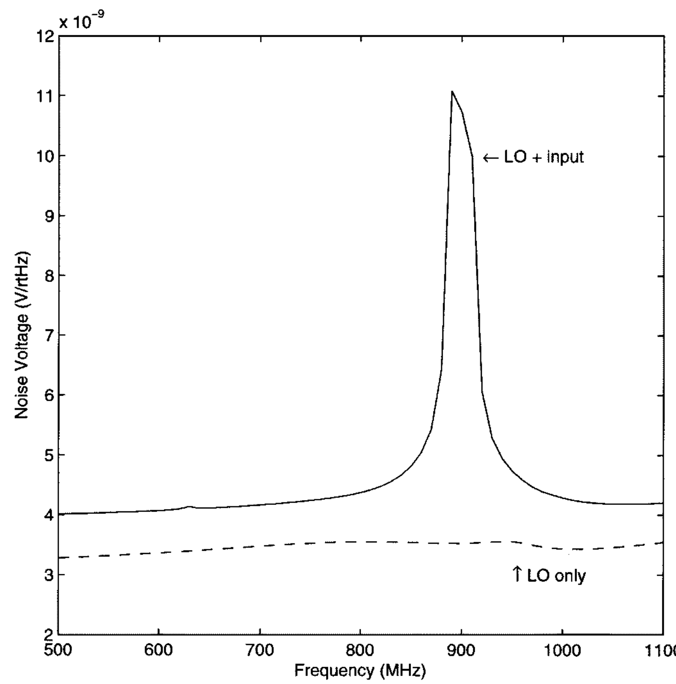


Fig. 15. Output noise voltage spectrum for a mixer with and without a strong input signal. The output noise significantly worsens in the presence of a strong input signal.

generally modeled 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., [54] and [55]) because of the lack of an alternative. At the moment, bias-dependent flicker noise modeling remains an open theoretical and experimental problem.

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, routinely performed 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° in phase to the noiseless signal, and hence, to large differences. The method based on stochastic differential equations above is essentially a small-signal analysis that calculates the growing nonstationary 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., [56]). Approaches for mixing noise [51], [54], [57] 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 [54] 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 [58].

IV. 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 emphasis is on the tools that are more commonly used for transistor level RFIC simulations. The following simulators have been considered: Advanced Design System (ADS) from Agilent EEsof [59], SpectreRF from Cadence Design Systems [60], and SP/XL-RF from Avista Design Systems [61]. The harmonica tool from Ansoft Corporation is similar in nature to the circuit-level simulator in ADS, and thus, will not be discussed here. All of these simulators support the standard SPICE analyses, and our focus is only on the RF specific capabilities that these simulators provide. New RF features are continually being added to these simulators. We have listed the features that are currently available to the best of our knowledge.

A. ADS

The ADS simulation environment from HP EEsof provides access to a number of simulation engines and algorithms through what is referred to as test benches. Here, we describe the features available in the HP RFIC Designer Premier product.

An HB simulator is used for the quasi-periodic steady-state analysis of circuits with up to twelve tones of excitations. This simulator employs fast, memory efficient matrix algorithms and can simulate larger circuits than the previous generation of HB simulators. Other capabilities include mixer noise analysis, large-signal s-parameter analysis that calculates s-parameters as a function of input power, and oscillator analysis.

HP EEsof offers the first commercial implementation of the Fourier envelope method, called Circuit Envelope. While HB is used to simulate a circuit in its steady state, Circuit Envelope can simulate amplifier harmonics versus time, oscillator turn on amplitude and frequency versus time, amplifier spectral regrowth due to digital modulation of the RF carrier, and PLL transient response.

B. SpectreRF

SpectreRF is available as an option in Spectre, a SPICE-like simulator from Cadence Design Systems. While the main focus in ADS is on frequency-domain algorithms (HB), SpectreRF extends the traditional time-domain algorithms to handle RFIC simulation. SpectreRF uses the Newton shooting method to calculate the periodic steady-state (PSS) response of circuits including oscillators. A recent version of the simulator also supports the mixed frequency-time method [62],[63].

SpectreRF can handle circuits with strong nonlinearities efficiently. It cannot handle distributed components except for 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 as described earlier. For circuits with a single tone large-signal excitation the periodic ac analysis [64] and the mixed frequency-time method [62] can be used. The first method allows calculation of the small-signal conversion gain for mixers and mixing noise, whereas the second method allows computation of intermodulation distortion using two small-signal RF tones.

C. SP/XL-RF

SP/XL-RF from Avista Design Systems is a simulator for RF and microwave circuits that embeds the Spectre simulation engine [65] within the Excel spreadsheet for doing what-if analyses. It provides accelerated steady-state calculation using the HB method. A heterodyne ac analysis (periodic ac, as in SpectreRF) is available for calculating small-signal conversion gain in mixers, while a heterodyne noise analysis is provided for determining mixing noise in amplifiers and phase noise in oscillators. In addition, s-parameters can be used to model multiport networks. SP/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.

D. Summary of Commercial RFIC Circuit Simulators

The features of the various commercial simulators described above are summarized in Table III. In this table, the

TABLE III
SUMMARY OF VARIOUS ANALYSIS METHODS AVAILABLE IN COMMERCIAL RFIC SIMULATORS

Method	ADS	SpectreRF	SP/XL-RF
Time-domain PSS		√	
Harmonic balance	√		√
Linear time varying	√	√	√
Circuit envelope	√		
Distributed elements	√	√ [4]	√
Nonlinearities	Mild	High	Mild
Oscillators	√	√	√
Mixing noise	√	√	√
Phase noise	√	√	√
S-parameters	√		√
Maximum # tones [5]	12	1	1

[4] Transmission lines only.

[5] Noncommensurate large-signal tones.

different analysis methods and the circuit simulators which support them are shown. Most of the commercial tools support several analysis capabilities, however, with some restrictions on the size of the circuit and the number of tones that can be simulated.

V. CONCLUSION

In this paper, we have provided an overview of the circuit simulation needs for RFIC design and techniques currently available for addressing some of these needs. We have focused on circuit simulation, since it is indispensable for design. System simulation and behavioral modeling are useful only early in the design phase. 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.

Periodic steady-state analysis, multitone HB, mixed frequency-time methods, and envelope methods promise faster simulation of RFICs. These techniques have also provided the basis for simulating noise in nonlinear circuits. Commercial tools incorporating some of these techniques are available and are useful, provided the limitations of the tools are well understood by the designer. At present, no single tool or analysis method can deliver a comprehensive solution to the designer. Furthermore, these tools have not been benchmarked for accuracy and performance which are important considerations in a design environment. Efforts such as [66] are attempting to provide a comprehensive set of tools and methodology for RFIC design. However, complete simulation of an RF front-end including the LNA, mixer, IF strip, and frequency synthesizer is still a distant dream. This problem is particularly challenging if effects of the layout, substrate coupling, and the package are to be considered. Therefore, researchers and tool developers have years of challenging work ahead of them to improve the RFIC simulation environment for efficient and accurate simulation of a complete RF front end.

ACKNOWLEDGMENT

The authors would like to thank one of the reviewers for a careful reading of the paper, for providing detailed comments, and for suggesting the "Fourier envelope" terminology. The first author, K. Mayaram, would like to thank Dr. K. Singhal and Prof. T. Fiez for helpful discussions. The second author, D. C. Lee, would like to thank D. Long, R. Fratti, J. Havens, and M. Shoji for many discussions, particularly on noise, and T. Lenahan for technical discussions on envelope simulation. The fifth author, J. Roychowdhury, would like to thank P. Feldmann, D. Long, A. Demir, and L. Toth for useful discussions on envelope methods and noise.

REFERENCES

[1] B. Razavi, "Challenges in portable RF transceiver design," *IEEE Circuits Devices*, pp. 12–25, Sept. 1996.
[2] K. Kundert, "Simulation methods for RF integrated circuits," in *Proc. ICCAD*, Nov. 1997, pp. 752–765.
[3] L. Nagel, "SPICE2: A computer program to simulate semiconductor circuits," Electron. Res. Lab., Univ. California at Berkeley, UCB/ERL M520, 1975.
[4] T. Quarles, "The SPICE3 implementation guide," Electron. Res. Lab., Univ. California at Berkeley, UCB/ERL M89/44, 1989.
[5] C. Hull and R. Meyer, "A systematic approach to the analysis of noise in mixers," *IEEE Trans. Circuits Syst. I*, vol. 40, pp. 909–919, Dec. 1993.
[6] A. Hajimiri and T. Lee, "A general theory of phase noise in electrical oscillators," *IEEE J. Solid-State Circuits*, vol. 33, pp. 179–194, Feb. 1998.
[7] K. Kundert, *The Designer's Guide to SPICE & SPECTRE*. Norwell, MA: Kluwer, 1995.
[8] ——, "Accurate Fourier analysis for circuit simulators," in *Proc. IEEE CICC*, May 1994, pp. 25–28.
[9] D. Weiner and J. Spina, *Sinusoidal Analysis and Modeling of Weakly Nonlinear Circuits*. New York: Van Nostrand, 1980.
[10] S. Chisholm and L. Nagel, "Efficient computer simulation of distortion in electronic circuits," *IEEE Trans. Circuit Theory*, vol. CT-20, pp. 742–745, Nov. 1973.
[11] J. Roychowdhury, "SPICE3 distortion analysis," Electronics Research Lab., Univ. California at Berkeley, UCB/ERL M89/48, Apr. 1989.
[12] C. McAndrew, B. Bhattacharyya, and O. Wing, "A single-piece C_∞-continuous MOSFET model including subthreshold conduction," *IEEE Electron. Device Lett.*, vol. 12, pp. 565–567, Oct. 1991.
[13] R. van Langevelde and F. Klaassen, "Accurate drain conductance modeling for distortion analysis in MOSFETs," in *Proc. Int. Electron. Devices Mtg.*, Dec. 1997, pp. 313–316.
[14] S. Kapur, D. Long, and J. Roychowdhury, "Efficient time-domain simulation of frequency-dependent elements," in *Proc. ICCAD*, Nov. 1996, pp. 569–573.
[15] S. Lin and E. Kuh, "Transient simulation of lossy interconnects based on the recursive convolution formulation," *IEEE Trans. Circuits Syst.*, Nov. 1992.
[16] J. Roychowdhury, A. R. Newton, and D. O. Pederson, "Algorithms for the transient simulation of lossy interconnect," *IEEE Trans. Computer-Aided Design*, vol. 13, pp. 96–104, Jan. 1994.
[17] T. J. Aprille and T. N. Trick, "Steady-state analysis of nonlinear circuits with periodic inputs," *Proc. IEEE*, vol. 60, pp. 108–114, Jan. 1972.
[18] ——, "A computer algorithm to determine the steady-state response of nonlinear oscillators," *IEEE Trans. Circuit Theory*, vol. CT-19, pp. 354–360, July 1972.
[19] P. Ashar, "Implementation of algorithms for the periodic-steady-state analysis of nonlinear circuits," Electronics Research Lab., Univ. California at Berkeley, UCB/ERL M89/31, 1989.
[20] R. Telichevesky, K. Kundert, and J. White, "Efficient steady-state analysis based on matrix-free Krylov subspace methods," in *Proc. IEEE DAC*, 1995, pp. 480–484.
[21] M. Nakhla and J. Vlach, "A piecewise harmonic balance technique for determination of the periodic response of nonlinear systems," *IEEE Trans. Circuits Syst.*, vol. CAS-23, pp. 85–91, Feb. 1976.
[22] A. Ushida and L. Chua, "Frequency-domain analysis of nonlinear circuits driven by multi-tone signal," *IEEE Trans. Circuits Syst.*, vol. CAS-31, pp. 766–778, Sept. 1984.
[23] K. Kundert and A. Sangiovanni-Vincentelli, "Simulation of nonlinear circuits in the frequency domain," *IEEE Trans. Computer-Aided Design*, vol. CAD-5, pp. 521–535, Oct. 1986.

[24] V. Rizzoli, C. Cecchetti, A. Lipparini, and F. Fastri, "General-purpose harmonic balance analysis of nonlinear microwave circuits under multitone excitation," *IEEE Trans. Microwave Theory Tech.*, vol. MTT-36, pp. 1650–1660, Dec. 1988.

[25] R. Gilmore and M. Steer, "Nonlinear circuit analysis using the method of harmonic balance—A review of the art. Part I: Introductory concepts," *Int. J. Microwave and Millimeter Wave CAE*, vol. 1, no. 1, pp. 22–37, 1991.

[26] K. Kundert, J. White, and A. Sangiovanni-Vincentelli, *Steady-State Methods for Simulating Analog and Microwave Circuits*. Norwell, MA: Kluwer, 1990.

[27] S. Maas, *Nonlinear Microwave Circuits*. Piscataway, NJ: IEEE Press, 1997.

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

[29] R. Melville, P. Feldmann, and J. Roychowdhury, "Efficient multi-tone distortion analysis of analog integrated circuits," in *Proc. IEEE CICC*, May 1995, pp. 241–244.

[30] P. Feldmann, R. Melville, and D. Long, "Efficient frequency domain analysis of large nonlinear analog circuits," in *Proc. IEEE CICC*, May 1996, pp. 461–464.

[31] D. Long, R. Melville, K. Ashby, and B. Horton, "Full-chip harmonic balance," in *Proc. IEEE CICC*, May 1997, pp. 379–382.

[32] L. Chua and A. Ushida, "Algorithms for computing almost periodic steady-state response of nonlinear systems to multiple input frequencies," *IEEE Trans. Circuits Syst.*, vol. CAS-28, pp. 953–971, Oct. 1981.

[33] K. Kundert, J. White, and A. Sangiovanni-Vincentelli, "A mixed frequency-time approach for distortion analysis of switching filter circuits," *IEEE J. Solid-State Circuits*, vol. 24, pp. 443–451, Apr. 1989.

[34] H. Brachtendorf, G. Welsch, and R. Laur, "A novel time-frequency method for the simulation of the steady state of circuits driven by multi-tone signals," in *Proc IEEE Int. Symp. CAS*, June 1997, pp. 1508–1511.

[35] J. Roychowdhury, "Efficient methods for simulating highly nonlinear multi-rate circuits," in *Proc. IEEE DAC*, June 1997, pp. 269–274.

[36] ——, "Analyzing circuits with widely-separated time scales using numerical PDE methods," *IEEE Trans. Circuits Syst. I*, May 1999.

[37] H. Brachtendorf, G. Welsch, R. Laur, and A. Bunse-Gerstner, "Numerical steady state analysis of electronic circuits driven by multi-tone signals," *Electron. Eng.*, vol. 79, pp. 103–112, 1996.

[38] L. Petzold, "An efficient numerical method for highly oscillatory ordinary differential equations," *SIAM J. Numer. Anal.*, vol. 18, pp. 455–479, June 1981.

[39] K. Kundert, J. White, and A. Sangiovanni-Vincentelli, "An envelope-following method for the efficient transient simulation of switching power and filter circuits," in *Proc. ICCAD*, Nov. 1988, pp. 446–449.

[40] E. Ngoya and R. Larcheveque, "Envelop transient analysis: A new method for the transient and steady state analysis of microwave communication circuits and systems," in *IEEE MTT Symp. Dig.*, June 1996, pp. 1365–1368.

[41] E. Rizolli, A. Neri, and F. Mastri, "A modulation-oriented piecewise harmonic-balance technique suitable for transient analysis of digitally modulated signals," in *Proc. 26th Microwave Conf.*, Sept. 1996, pp. 546–550.

[42] D. Sharrit, "New method of analysis of communication systems," presented at the MTTS WMFA: Nonlinear CAD Workshop, June 1996.

[43] P. Feldmann and J. Roychowdhury, "Computation of circuit waveform envelopes using an efficient, matrix-decomposed harmonic balance algorithm," in *Proc. ICCAD*, Nov. 1996, pp. 295–300.

[44] J. Roychowdhury, "MPDE methods for efficient analysis of wireless systems," in *Proc. IEEE CICC*, May 1998, pp. 451–454.

[45] M. Okumura, T. Sugawara, and H. Tanimoto, "An efficient small-signal frequency analysis method for nonlinear circuits with two frequency excitations," *IEEE Trans. Computer-Aided Design*, vol. 9, pp. 225–235, Mar. 1990.

[46] R. Telichevesky, K. Kundert, and J. White, "Efficient AC and noise analysis of two-tone RF circuits," in *Proc. IEEE DAC*, June 1996, pp. 292–297.

[47] P. Bolcato and R. Poujois, "A new approach for noise simulation in transient analysis," in *Proc IEEE Int. Symp. CAS*, June 1992, pp. 887–890.

[48] J. McNeill, "Jitter in ring oscillators," Ph.D. dissertation, Boston Univ., Boston, MA, 1994.

[49] T. Weigandt, "Low-phase-noise, low-timing-jitter design techniques for delay cell based VCO's and frequency synthesizers," Electronics Research Lab., Univ. California at Berkeley, UCB/ERL M98/5, 1998.

[50] A. Kerr, "Noise and loss in balanced and subharmonically pumped mixers: Part 1—Theory," *IEEE Trans. Microwave Theory Tech.*, vol. MTT-27, pp. 938–943, Dec. 1979.

[51] V. Rizzoli, F. Mastri, and D. Masotti, "General noise analysis of nonlinear microwave circuits by the piecewise harmonic-balance technique," *IEEE Trans. Microwave Theory Tech.*, vol. 42, pp. 807–819, May 1994.

[52] M. Okumura, H. Tanimoto, T. Itakura, and T. Sugawara, "Numerical noise analysis for nonlinear circuits with a periodic large signal excitation including cyclostationary noise sources," *IEEE Trans. Circuits Syst. I*, vol. 40, pp. 581–590, Sept. 1993.

[53] W. Gardner, *Introduction to Random Processes*. New York: McGraw-Hill, 1989.

[54] J. Roychowdhury, D. Long, and P. Feldmann, "Cyclostationary noise analysis for large RF circuits with multitone excitations," *IEEE J. Solid-State Circuits*, vol. 33, pp. 324–336, Mar. 1998.

[55] A. Demir, E. Liu, and A. Sangiovanni-Vincentelli, "Time-domain non Monte-Carlo noise simulation for nonlinear dynamic circuits with arbitrary excitations," *IEEE Trans. Computer-Aided Design*, vol. 15, pp. 493–505, May 1996.

[56] F. X. Kärtner, "Analysis of white and $f^{-\alpha}$ noise in oscillators," *Int. J. Circuit Theory Applic.*, vol. 18, pp. 485–519, 1990.

[57] W. Anzill and P. Russer, "A general method to simulate noise in oscillators based on frequency domain techniques," *IEEE Trans. Microwave Theory Tech.*, vol. 41, pp. 2256–2263, Dec. 1993.

[58] A. Demir, A. Mehrotra, and J. Roychowdhury, "Phase noise and timing jitter in oscillators," in *Proc. IEEE CICC*, May 1998, pp. 45–48.

[59] Agilent EEsof EDA. (2000) Advanced Design System overview. [Online]. Available: http://www.tmo.hp.com/tmo/hpeesof/products/ads/adsoview.html

[60] Cadence Design Systems. (2000) Affirma Analog Artist circuit design environment. [Online]. Available: http://www.cadence.com/technology/custom/products/analog_artist.html

[61] Avista Design Systems. (1999) SP/XL for RF and microwave design. [Online]. Available: http://www.avista.com/SPXL/index1.html

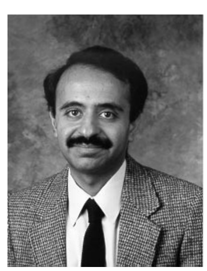
[62] J. Chen, D. Feng, J. Phillips, and K. Kundert, "Simulation and modeling of intermodulation distortion in communication circuits," in *Proc. IEEE CICC*, May 1999, pp. 5–8.

[63] D. Feng, J. Phillips, K. Nabors, K. Kundert, and J. White, "Efficient computation of quasiperiodic circuit operating conditions via a mixed frequency/time approach," in *Proc. IEEE DAC*, June 1998, pp. 635–640.

[64] R. Telichevesky, K. Kundert, and J. White, "Receiver characterization using periodic small-signal analysis," in *Proc. IEEE CICC*, June 1996, pp. 449–452.

[65] K. S. Kundert, "Spectre user's guide: A frequency domain simulator for nonlinear circuits," EECS Industrial Liaison Program, Univ. California at Berkeley, Apr. 1987.

[66] A. Dunlop, A. Demir, P. Feldmann, S. Kapur, D. Long, R. Melville, and J. Roychowdhury, "Tools and methodology for RF IC design," in *Proc. IEEE DAC*, June 1998, pp. 414–419.



Kartikeya Mayaram (S'82–M'89–SM'99) received the B.E. (Hons.) degree in electrical engineering from the Birla Institute of Technology and Science, Pilani, India, in 1981, the M.S. degree in electrical engineering from the State University of New York, Stony Brook, in 1982, and the Ph.D. degree in electrical engineering from the University of California at Berkeley in 1988.

From 1988 to 1992, he was a Member of Technical Staff in the Semiconductor Process and Design Center of Texas Instruments, Dallas, TX. From 1992 to 1996, he was a Member of Technical Staff at Bell Labs, Allentown, PA. He was an Associate Professor in the School of EECS, Washington State University, Pullman, from 1996 to 1999. Since January 2000, he has been an Associate Professor in the Electrical Engineering Department, Oregon State University at Corvallis. His research interests are in the areas of circuit simulation, device simulation and modeling, integrated simulation environments, and analog/RF design.

Dr. Mayaram received the National Science Foundation CAREER Award in 1997. He is an Associate Editor of IEEE TRANSACTIONS ON COMPUTER-AIDED DESIGN OF INTEGRATED CIRCUITS AND SYSTEMS.

David C. Lee (M'92) received the B.A.Sc. and M.A.Sc. degrees in systems design engineering from the University of Waterloo, Ontario, Canada.

He is a Scientist with Mentor Graphics, Allentown, PA, responsible for defining and developing future generations of CAD methods for verifying extremely large and complex mixed-signal and radio frequency (RF) chips. Since 1988, he has worked in the area of circuit simulation and modeling at Bell Laboratories, Lucent Technologies (1992–1998), and Bell Northern Research of Nortel (1988–1992). Since 1995, he has been researching and developing the new core technologies for modeling and simulating RF integrated circuits for the Celerity circuit simulator developed at Bell Labs Design Automation. His research interests include precision analog and RF simulation, behavioral modeling, and statistical design for manufacturability.

Mr. Lee is a member of SIAM.

Shahriar Moinian photograph and biography not available at the time of publication.

David A. Rich photograph and biography not available at the time of publication.

Jaijeet Roychowdhury received the B.Tech. degree from the Indian Institute of Technology, Kanpur, India, in 1987, and the Ph.D. degree from the University of California at Berkeley in 1992.

From 1992 to 1995, he was with the CAD Laboratory of AT&T Bell Laboratories, Allentown, PA. Since 1995, he has been with the Communication Sciences Research Division of Lucent Technologies Bell Laboratories, Murray Hill, NJ. His current interests are in multi-time simulation, reduced-order modeling, and noise issues in communications design.

Dr. Roychowdhury has received several awards for his research on analog and RF verification.