Tools and Methodology for RF IC Design

Al Dunlop Alper Demir Peter Feldmann Sharad Kapur
David Long Robert Melville Jaijeet Roychowdhury

Bell Laboratories, Murray Hill, NJ

Abstract

We describe powerful new techniques for the analysis of RF circuits. Next-generation CAD tools based on such techniques should enable RF designers to obtain a more accurate picture of how their circuits will operate. These new simulation capabilities will be essential in order to reduce the number of design iterations needed to produce complex RF ICs.

1 Introduction

Design methodology and superior computer-aided design tools are key to success in the integrated circuit (IC) business. They are particularly important in the case of radio-frequency (RF) IC applications, where the digital IC divide-and-conquer design style, based on partitioning by functional blocks and abstraction levels, does not apply.

The goal of an RF designer is to get a manufacturable design that meets the specifications with minimum cost, under severe time-to-market constraints. Unlike traditional discrete-component RF design, prototyping is practically impossible, and the validation of a design can only be done by extensive simulation. Undetected mistakes in the design result in costly and time-consuming iterations that involve silicon processing.

Design methodology is the approach or philosophy used in creating a design. This includes the steps associated with the synthesis of the circuit that implements the desired functionality. This constructive aspect needs to be complemented by a verification methodology required to assure, with a high degree of confidence, the validity of the design before it is committed to silicon. The design and verification methodologies are implemented with the help of design tools. These tools aid in carrying out design and verification steps, i.e., producing mask layout, model extraction, and the various performance verification steps.

Typical specifications which must be met by RF ICs and, therefore, supported by the design and verification methodologies, include sensitivity, linearity, adjacent channel interference, and power level. These specifications depend on other performance measures such as noise figure, intercept point, and 1dB compression point. Verification tools need to be able to analyze the design at its various stages and predict the performance measures as accurately as possible.

A number of factors characterize RF-IC applications and render the verification process particularly difficult. One is the extreme range of frequencies, or time-scales, over which the circuit is operating. Typical applications have carrier frequencies in the 1-2 GHz range with modulating signals in the KHz range, a span of 6 orders of magnitude. The signals have a tremendous dynamic range. A radio will often have 80 dB of gain in the receiver path. Signals that must be received may be 60 dB weaker than the ones in adjacent frequency bands. Attenuations of more than 100dB for unwanted interferences are not unheard-of. At GHz frequencies, the passive components of an RF circuit are very significant and must be carefully modeled. Coupling through the interconnect and the substrate are prone to cause problems such as undesired interference (cross-talk) from adjacent structures. At these high frequencies, capacitors, inductors, on-chip wires, the silicon substrate, package, sockets, etc., must be analyzed at the electromagnetic level in order to capture and model all the significant effects. Finally, device noise determines the fundamental bounds of circuit performance and plays a significant role in RF circuit design. Device noise (flicker noise, shot noise, thermal noise, etc.) is a stochastic phenomenon resulting from the physics of the device and should be distinguished from deterministic phenomena such as crosstalk.

The verification methodology must rely on tools that are appropriate for the problems enumerated above. These tools must be efficient enough to permit a sufficiently extensive verification of the design. In some cases, the capability to perform more simulation results in the uncovering and fixing of more design flaws, thus avoiding costly processing iterations.

The wide range of operating frequencies imposes an insurmountable burden on the SPICE-type, time-domain simulators familiar to IC designers. It would take the simulation of millions of carrier cycles to analyze only one period of the modulating signal. Therefore novel, multi-scale simulation techniques must be employed. One such technique, the method of harmonic balance, is widely used and very familiar to microwave RF designers. Harmonic balance analysis is suitable for the analysis of circuits operating at widely separated frequencies and provides the required dynamic range. Unfortunately, the existing commercial implementations of harmonic balance simulators rely on algorithms that are not capable of handling the large circuits, consisting mainly of nonlinear elements, which characterize typical RF-IC applications. Moreover, harmonic balance cannot address all RF-IC verification problems of interest. Section 2 surveys an entire family of multi-scale simulation techniques (including harmonic balance) capable of evaluating all RF-IC circuit performance measures of interest. A novel implementation of harmonic balance suitable for the simulation of a full RF chip is described.

Device noise being among the most important design considerations, any verification methodology must rely on adequate tools that simulate the effects of noise in RF circuits. Noise sources and signals in RF circuits are modulated by time-varying signals and can only be modeled by cyclo-stationary and non-stationary stochastic processes. Analyzing the effect of noise in oscillators (phase noise/timing jitter) is particularly difficult. Analyzing the effects of phase noise in autonomous oscillators is described in Section 3.

The accurate analysis of passive linear structures at GHz frequencies must rely on the solution of the full set of Maxwell's equations, in all three dimensions, on domains having varied and complicated geometries. Section 4 surveys the techniques available for this problem and describes in detail a novel, very efficient method-of-moments approach. The models resulting from the analysis of the linear structures must be combined with the active components into a comprehensive simulation of the RF circuit.

35th Design Automation Conference ®

1-58113-049-x-98/0006/$3.50 DAC98 - 06/98 San Francisco, CA USA

Unfortunately some of the linear analysis tools can only produce models in the frequency domain. Of all the analysis methods, only harmonic balance can naturally handle these models. For other types of analysis, the linear model must be translated into a form suitable for time-domain simulations. The methodology to compute reduced-order models that are compatible with both time-domain and frequency-domain simulations is described in Section 5.

2 Circuit Simulation for RF ICs

Simulating RF ICs places new demands on transistor-level simulation tools: (1) The need to find steady-state and transient/envelope response with stimuli at widely separated frequencies or time-scales. Mixer simulation and two-tone intermodulation studies are classic examples of this scenario. (2) The need for large dynamic range in numerical results; Radio designs commonly deal with intentional signals separated in amplitude by 60-80 dB. Accurate prediction of spurious signals and feed-through requires a dynamic range in excess of 100 dB. Ideally, the numerical methods used should not be the limiting factor. (3) The simulation should be accomplished with reasonable computer resources.

The method of *harmonic balance* (HB) [16, 24] is already familiar to microwave RF designers and is becoming more well known among RF IC designers. Section 2.1 contains a discussion of the application of HB to a complete RF transmitter chip.

A new formulation, introduced even more recently, provides a general mathematical framework for the aforementioned issues. The *multi-rate partial differential equation (MPDE)* technique [4, 39, 40] represents signals as functions of more than one time variable. HB is a special case of the MDPE, which also allows multi-tone analysis of strongly nonlinear circuits (in fact, non-RF circuits such as power converters and switched-capacitor filters can also be treated effectively with the MPDE). Section 2.2 contains a discussion of the MPDE for strongly nonlinear RF circuits.

2.1 Harmonic Balance

The method of Harmonic Balance represents all circuit waveforms in the frequency domain. The method is particularly natural in the case of incommensurate multi-tone drive [45]. Early implementations focused on microwave circuitry [29], which often has a relatively small number of nonlinear components embedded in a large collection of linear elements. Unfortunately, RF integrated circuits do not really fit this model, since sophisticated semiconductor device equations require nonlinear modeling of the majority of components. Recent work by various authors [3, 10, 28, 31, 37] has demonstrated that Harmonic Balance can handle integrated designs containing many more nonlinear components than traditional implementations of the technique. Specifically, iterative linear algebra techniques [12] have been used to solve the large Jacobian matrix which results from linearization of the nonlinear equations.

Of course a large circuit can always be simulated in pieces. However, there are several disadvantages to this approach: (1) Circuit partitioning is inconvenient and error-prone, especially for extracted networks. (2) It is often difficult to preserve source and load impedances across partition boundaries. Nonlinear effects like load pull can be missed entirely. (3) Leakage and feed-through are a crucial concern in integrated RF designs and, by definition, cross module boundaries.

Figure 1 shows a typical output spectrum from a Harmonic Balance simulation. The simulated circuit was a large dual-conversion quadrature modulator chip designed for cellular applications. The left-most spectral component is a weak, but significant, spurious response from the local oscillator which might

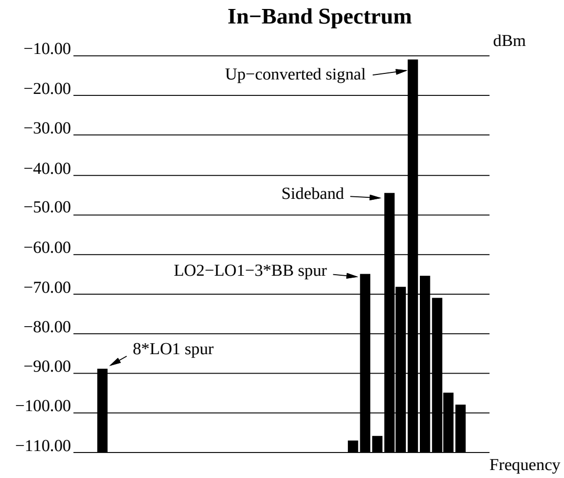


Figure 1: Modulator in-band spectrum

or might not be within the specifications of the chip. The spectrum display also shows a sideband component at a level of -35 dBc which was out of specification for this design, and was traced back to a layout imbalance. This effect was missed during conventional transient analysis. This was a big job—about 27 hours on a fast scientific workstation with approximately 500MB of memory—but still significantly faster and more useful than a transient analysis.

To summarize the significance of Harmonic Balance simulation for this example:

- The large range in driving frequencies [80 KHz and 1.62 GHz] would require a conventional transient analysis to run for several hundred *thousand* cycles. A conventional transient run *was* performed on this design, but with the base-band frequency set to 1 MHz (rather than 80 KHz) and required approximately the same amount of time as the Harmonic Balance run using the appropriate base-band frequency.
- The numerical dynamic range of the transient simulation was insufficient to pick up a weak spurious response at -78 dBc.

On the other hand:

- The memory and time required for Harmonic Balance simulation increase rapidly as more "tones" are added, i.e., driving sources at frequencies which are not in a simple harmonic relationship to one another. For example, predicting the intermodulation distortion of the entire modulator chain would require two different fundamental frequencies at base-band for a total for *four* tones; such a simulation would probably exceed available memory for this design, even with the memory compression techniques which were used in this example.
- Within the bounds of available numerical dynamic range, the time and memory requirements of transient simulation are not sensitive to the number of fundamental frequencies applied to the circuit.

2.2 Multi-Time Methods

The key to the MPDE formulation is the use of multivariate functions (functions of several time variables) to represent signals with separated time scales efficiently. To understand the concept, consider the product of a 1 Hz sine wave and a 1Ghz pulse train, given by:

$$y(t) = \sin(2\pi t)\,\mathrm{pulse}\left(\frac{t}{10^9}\right) \tag{1}$$

Figure 2 depicts $y(t)$, with the pulse period of 10^9 changed to 50 for viewing convenience. This quasi-periodic signal is expensive to represent in the time domain because 10^9 pulses of different shapes need to be sampled before the waveform repeats. It is this problem that makes traditional time-domain techniques like SPICE's transient analysis inefficient for such signals. Representation in the frequency domain as a two-tone signal is also inefficient because the pulses require many Fourier components for accuracy.

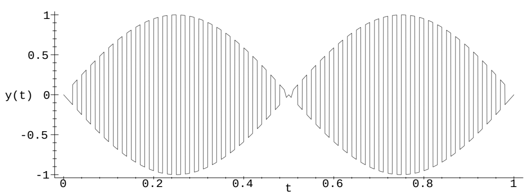


Figure 2: $y(t)$

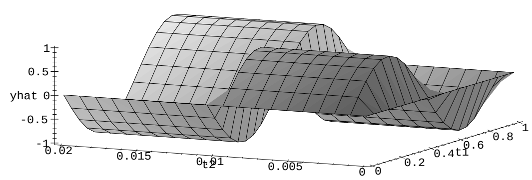


Figure 3: $\hat{y}(t_1,t_2)$

Consider, however, the function of two variables obtained by replacing the 'slow' time component by t_1 and the 'fast' time component by t_2:

$$\hat{y}(t_1,t_2) = \sin(2\pi t_1)\ \text{pulse}\left(\frac{t_2}{10^9}\right) \tag{2}$$

$\hat{y}(t_1,t_2)$, a *bi-variate form* of $y(t)$, is shown in Figure 3. Notice that it is easy to represent $\hat{y}$ accurately using relatively few numerical samples, in contrast to $y(t)$ in Figure 2. The number of samples does not depend on the separation of the two time scales, which merely determines the scaling of the axes. Moreover, $y(t)$ can be easily obtained by interpolation from samples of $\hat{y}(t_1,t_2)$, using the fact that $y(t) = \hat{y}(t,t)$ and that $\hat{y}(t_1,t_2)$ is periodic in each argument.

This observation is the basis of the MPDE formulation, in which all the waveforms in a circuit are represented in their bivariate forms (or multivariate forms if there are more than two time scales). The key to efficiency is to solve for these waveforms directly, without involving the numerically inefficient one-dimensional forms at any point. To do this, it is necessary to first describe the circuit's equations using the multivariate functions. The traditional form of a circuit's equations, used in all simulators, is the Differential-Algebraic Equation (DAE):

$$\dot{q}(x) + f(x) = b(t) \tag{3}$$

$x(t)$ is the vector of circuit unknowns (node voltages and branch currents); q denotes the charge/flux terms and f the resistive terms; $b(t)$ is the vector of excitations to the circuit (typically from independent voltage/current sources). It can be shown [40] that if $\hat{x}(t_1,t_2)$ and $\hat{b}(t_1,t_2)$ denote the bi-variate forms of the circuit unknowns and excitations, then the following MPDE is the correct generalization of (3) to the bi-variate case:

$$\boxed{\frac{\partial q(\hat{x})}{\partial t_1} + \frac{\partial q(\hat{x})}{\partial t_2} + f(\hat{x}) = \hat{b}(t_1,t_2)} \tag{4}$$

More precisely, if $\hat{b}$ is chosen to satisfy $b(t) = \hat{b}(t,t)$, and $\hat{x}$ satisfies (4), then it can be shown that $x(t) = \hat{x}(t,t)$ satisfies (3). Also, if (3) has a quasi-periodic solution, then (4) can be shown to have a corresponding bi-variate solution.

By solving the MPDE numerically in the time domain, strong nonlinearities can be handled efficiently. The following new methods have been developed for solving (4):

1. **Quasi-periodic time-domain methods (MFDTD and HS):** Quasi-periodic solutions are found by enforcing bi-periodic boundary conditions on the MPDE. In the Multivariate Finite Difference Time Domain (MFDTD), (4) is discretized on a grid in the t_1-t_2 plane by approximating the differentiation operators with a numerical differentiation formula. The resultant system of nonlinear equations, together with the bi-periodic boundary conditions, is solved using a nonlinear solution method. The grid is refined adaptively so that the solution is captured efficiently. Another purely time-domain method, Hierarchical Shooting (HS), is a generalization of the traditional shooting method to multiple time scales. Both MFDTD and HS are appropriate for circuits with no sinusoidal waveform components, such as power converters.

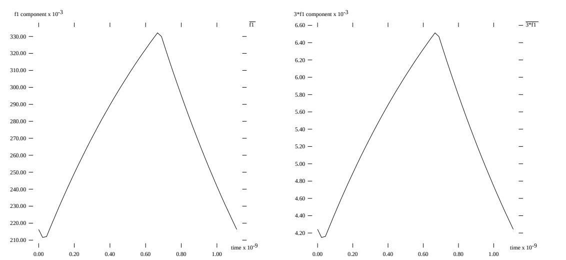
2. **Quasi-periodic mixed frequency/time method (MMFT):** In some circuits, the slow-scale signal path is often almost linear, while the fast-scale action is highly nonlinear. The linearity of the signal path can be exploited by expressing the slow scale components in a short Fourier series, and solving the mixed frequency/time system of equations. This Multivariate Mixed Frequency Time (MMFT) method is often more efficient for switched-capacitor filters and switching mixers.

3. **Time domain envelope methods (TD-ENV):** Envelope-type solutions can be generated from the MPDE by applying mixed initial/periodic boundary conditions. Novel time-domain methods based on FDTD or shooting along the fast time scale, and transient integration along the slow time scale, have been devised [40]. These techniques are capable of handling circuits with nonlinearities on a fast time scale, e.g., power converters, switched-capacitor filters, switching mixers, etc..

The above numerical techniques generate sparse matrices with near diagonal or block-diagonal structure, which makes it convenient to use iterative linear solution methods (e.g., [10, 31, 41])to solve large circuits efficiently.

The application of MMFT to a double-balanced switching mixer and filter circuit is described below. The RF input to the mixer was a 100kHz sinusoid with amplitude 100mV; this sent it into a mildly nonlinear regime. The LO input was a square wave of large amplitude (1V), which switched the mixer on and off at a fast rate (900Mhz).

The circuit was also simulated by univariate shooting for comparison. For MMFT, 3 harmonics were taken in the RF tone f_1 = 100kHz (corresponding to the t_1 variable). The LO tone at f_2 = 900MHz was handled by shooting in the t_1 variable. The output of the algorithm is a set of time-varying harmonics that are periodic with period $T_2 = \frac{1}{f_2}$. The first harmonic is shown in Figure 4(a). This plot contains information about all mix components of the form $f_1 + i f_2$, i.e., the frequencies 900.1 Mhz, 1800.1 Mhz, etc. The main mix component of interest, 900.1 Mhz, is found by taking the fundamental component of the waveform in Figure 4(a). This has an amplitude of 60mV.



(a) First harmonic (b) Third harmonic

Figure 4: Switching Mixer: MMFT output

The third harmonic is shown in Figure 4(b). It contains information about the mixes $3f_1 + i f_2$, i.e., the frequencies 900.3 Mhz, 1800.3 Mhz, etc.. The amplitude of the 900.3 Mhz component is about 1.1mV; hence the distortion introduced by the mixer is about 35dB below the desired signal.

The output produced by univariate shooting is shown in Figure 5. This run, using 50 steps per fast period, took almost 300 times as long as the new algorithm.

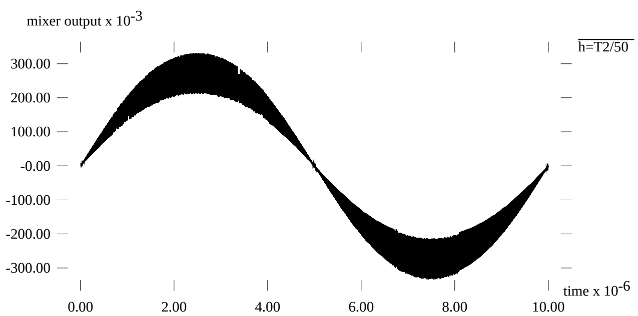


Figure 5: Mixer output from univariate shooting

3 Phase Noise in Oscillators

Oscillators are ubiquitous in physical systems, especially electronic and optical ones. In RF communication systems, they are used for frequency translation of information signals and for channel selection.

Noise is of major concern in oscillators, because introducing even small noise into an oscillator leads to dramatic changes in its frequency spectrum and timing properties. This phenomenon, peculiar to oscillators, is known as phase noise or timing jitter. A perfect oscillator would have localized tones at discrete frequencies (i.e., harmonics), but any corrupting noise spreads these perfect tones, resulting in high power levels at neighboring frequencies. This effect is the major contributor to undesired phenomena such as interchannel interference, leading to increased bit-error-rates (BER) in RF communication systems. Characterizing how noise affects oscillators is therefore crucial for practical applications. The problem is challenging, since oscillators constitute a special class among noisy physical systems: their autonomous nature makes them unique in their response to perturbations.

Considerable effort has been expended over the years [1, 18, 23, 25, 27, 30, 38, 46] in understanding phase noise and in developing analytical, computational and experimental techniques for its characterization. Despite the importance of the problem and the large number of publications on the subject, a consistent and general treatment, and computational techniques based on a sound theory, appear to be still lacking. We developed a novel, rigorous theory for phase noise which leads to efficient numerical methods for its characterization [5]. Our techniques and results are general; they are applicable to any oscillatory system, electrical or otherwise.

Understanding how perturbations affect stable oscillators is a crucial step in the analysis of phase noise. A nonlinear perturbation analysis that is valid for oscillators is required, in contrast to traditional linear perturbation analysis, which is not consistent for oscillators and results in non-physical predictions. With a rigorous nonlinear perturbation analysis, one can show that the effect of perturbations on an oscillator's response can be represented by a changing time shift, or phase deviation, in the periodic output of the unperturbed oscillator, and an additive component, called the orbital deviation, to the phase-shifted oscillator waveform. Moreover, the phase deviation and the orbital deviation can always be chosen such that the phase deviation will, in general, keep increasing with time even if the perturbation is always small, but the orbital deviation will always remain small. These results formalize existing intuition among designers about oscillator operation.

Considering random noise perturbations (e.g. thermal noise, $1/f$ noise), one sees that jitter and spectral spreading are closely related, and both are determined by the manner in which the phase deviation, a random process, spreads with time. The average spread of the jitter (mean-square jitter) increases without bound (precisely linearly for shot and thermal noise) with time. The power spectrum of the perturbed oscillator has a finite value at the carrier frequency and its harmonics, and the total carrier power is preserved despite spectral spreading due to noise. Previous analyses based on linear time-invariant (LTI) or linear time-varying (LTV) concepts erroneously predict infinite noise power density at the carrier, as well as infinite total integrated power. Furthermore, one can show that the oscillator's output is stationary. This might be surprising at first sight, since oscillators are nonlinear systems with periodic swings. Hence it might be expected that output noise power would change periodically as in forced systems. However, it must be remembered that while forced systems are supplied with an external time reference (through the forcing), oscillators are not. Cyclostationarity in the oscillator's output would, by definition, imply a time reference. Hence the stationarity result reflects the fundamental fact that noisy autonomous systems cannot provide a perfect time reference.

We obtain not only the above qualitative characterizations from a rigorous theory of phase noise in oscillators, but also correct computational techniques that are efficient for practical circuits. New numerical methods (in the time and frequency domains) for jitter/spectral dispersion, require only a knowledge of the steady state of the unperturbed oscillator and the values of the noise generators. Large circuits are handled efficiently. The separate contributions of noise sources, and the sensitivity of phase noise to individual circuit devices and nodes, can be obtained easily. We used the theory and numerical methods to analyze several oscillators, and compared the results against measurements. We obtained good matches even at frequencies close to the carrier, unlike most previous analyses.

4 Accurate Extraction

Extracting compact, accurate linear models for packages, interconnect, and components plays a significant role in modern RF designs. Models can be extracted in a variety of ways, but for the high accuracy that critical sections of RF designs demand, only direct numeric simulation suffices. At lower frequencies (far from resonance), capacitive or inductive coupling can be extracted with electrostatic or magnetostatic simulations. These simulations are generally simpler than full electromagnetic simulations, but at high frequencies (near resonances), electromagnetic simulation must be used. In either case, the problem is ultimately reduced to that of solving a linear system of equations $Ax = b$. This is accomplished by discretizing the physical structure into small elements. Interactions between elements give rise to the matrix A. Stimuli such as excitation voltages are reflected in the right-hand side b. Output from the simulator is typically an S parameter matrix, which can be used directly in a frequency-domain simulation. Alternatively, a circuit model can be constructed, using either parameter fitting or the model reduction techniques described in Section 5.

Simulation methods can be divided into two classes based on the type of matrix A that is involved. Methods in the first class use differential equation formulations. Finite-element (FE) [11], finite-difference (FD) [43], and finite-difference time-domain (FDTD) [44] approaches all fall into this class. In all of these, the matrix A is sparse. Methods from the second class use integral equations. The method-of-moments (MoM) approach [19] is based on integral equations. In this case, A is a dense matrix. However, an integral equation formulation allows us to apply Green's theorem, reducing volume integrals to surface integrals. This can reduce the matrix dimension significantly since the discretization only involves surfaces such as the boundary of a conductor or the interface between two dielectrics. The characteristics of the two classes are summarized in Table 1. Commercial tools such as Raphael and Ansoft are based on differential equations, while others such as Sonnet and Momentum use integral equations. For simulations that involve difficult-to-describe material variations (e.g., the doping profile of a MOSFET), the differential approach is clearly superior. For typical IC, board, or MCM simulations, where the material variation is simpler, the integral approach becomes attractive due to the use of surface discretizations. In these cases, the integral formulation often reduces the problem size by orders of magnitude. This reduction, combined with the advanced numerical methods described below, can yield MoM solvers that are ten or more times faster than finite-difference and finite-element tools. Thus, we believe that in the future integral equation methods will be the approach of choice for these problems.

	Differential	Integral
Matrix type	sparse	dense
Discretization	volume	surface
Matrix conditioning	poor	good

Table 1: Characteristics of classes of simulation methods

In recent years, numeric methods have been developed where the dense matrix A arising from an integral equation formulation can be represented implicitly and concisely. With these methods, the size of the representation for A is only $O(n)$ or $O(n \log n)$, where n is the dimension of the matrix. This is a significant reduction compared to the $O(n^2)$ storage that would be required to store A directly. Further, multiplication by A can be accomplished in time proportional to the size of the representation. Because integral equations usually give a relatively well-conditioned matrix A, Krylov-subspace iterative methods [41] can be used to quickly solve the linear system.

FastCap [33] and FastHenry [20] were the first electrostatic extraction tools based on this methodology. Multiplication by the matrix A in these tools is accomplished using the Fast Multipole Method [17]. The main weakness of these tools is that the interaction between discretization elements must have a $1/|\mathbf{r} - \mathbf{r}'|$ dependence, where $\mathbf{r}$ and $\mathbf{r}'$ are the positions of the elements in space. This "kernel" dependence means that all material boundaries must be discretized.

IES^3 [21] is a more recent kernel-independent scheme for compressing the matrix A. With IES^3, the matrix is recursively decomposed and compressed using the singular value decomposition. The interaction between well-separated groups of discretization elements is represented using a low-rank outer product. The interaction need not have a $1/|\mathbf{r} - \mathbf{r}'|$ dependence. In an extraction tool using IES^3, the effect of layered materials can be captured using a specialized Green's function [32]. IES^3 has been used in both electrostatic [21] and electromagnetic [22] simulators. Figure 6 shows how time and memory requirements scale only slightly faster than linearly with increasing problem size in an IES^3-based electromagnetic simulator. Comparisons of electromagnetic simulations to measurements for an integrated CMOS inductor on a lossy substrate are shown in Figure 7. We believe that in the near future, these techniques will make it possible to simulate critical multi-component assemblies such as the resonator shown in Figure 8.

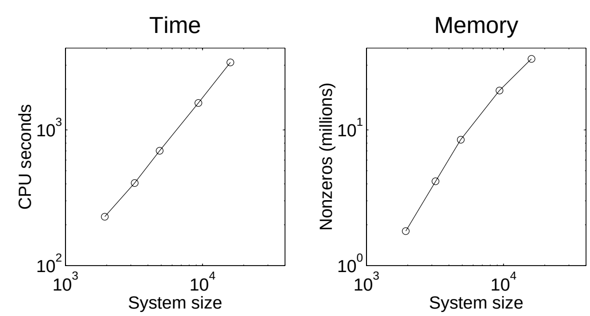


Figure 6: Electromagnetic simulation time and memory requirements

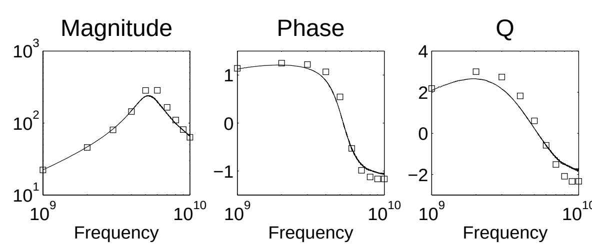


Figure 7: Comparison of inductor simulations and measurements

5 Reduced-Order Modeling

Integrated RF circuits often contain large linear sub-blocks. They can represent passive components such as capacitors, on-chip inductors, tapered RC lines, etc. They can also represent the parasitic effects of interconnect, substrate, power distribution lines, package, MCM, or board.

These linear components are sometimes modeled by equivalent lumped-element circuits composed of resistors, capacitors, inductors, and mutual couplings. Such equivalent models are generated by IC layout extraction tools. The lumped-element circuit model is, however, only adequate when signal wavelengths, (3cm for 10GHz) are large in comparison to circuit feature sizes.

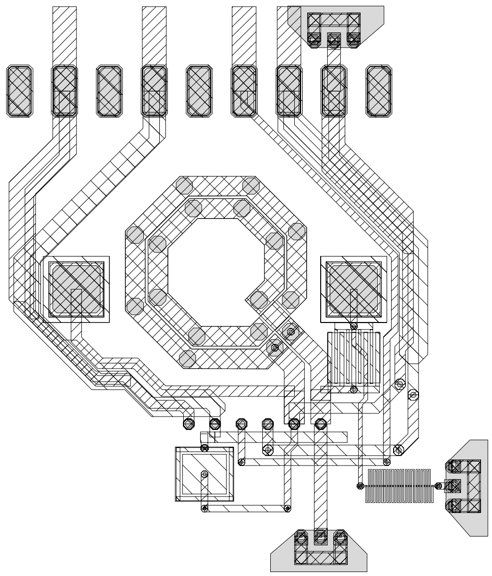


Figure 8: Resonator assembly

At current operating frequencies, 1-2GHz, such methods are adequate only at the chip level. As integrated circuits penetrate higher frequency applications, or even at current frequencies, at the package or printed circuit board levels, lumped-element circuit models become inaccurate. For these cases designers rely on frequency-domain models. These models are obtained either from measurements or, as described in the previous section, from field solvers.

The analysis of RF circuits containing such linear sub-blocks raises a number of problems and difficulties. First, the large size of the linear sub-blocks may render the total size of the problem infeasible for general RF circuit analysis of the type described in Section 2. For example, lumped element networks produced by layout extraction tools can reach sizes of millions of elements. Second, the linear sub-block may be modeled as a frequency-domain transfer function matrix while the rest of the nonlinear RF circuit is modeled as a set of nonlinear, time-domain, differential-algebraic equations. Of all the general RF circuit analysis methods, only the method of Harmonic Balance can efficiently handle a mixture of time-domain and frequency-domain methods.

A solution to both the size and the mixed domain problems is reduced-order modeling. The reduced-order model should capture with sufficient accuracy the interesting behavior of the original linear sub-block in the desired domain of interest, and, simultaneously, be much less expensive to evaluate. In addition, the reduced-order model should have efficient representations in both the time and frequency domains.

A methodology for reduced-order modeling that satisfies the above requirements is based on computing Padé-type approximations [15] to the frequency domain transfer function of the linear sub-block. The Padé approximation is a general method which attempts to approximate a function, which may not have an analytical expression and/or may be very difficult and expensive to compute, by a much simpler rational function. The coefficients of the approximating rational function are chosen such that the original function and the approximant match in the maximum number of the leading Taylor expansion coefficients.

The Padé approximation is particularly attractive for frequency-domain transfer functions because it captures very well the dominant poles and zeros of the linear system. In addition, the matched Taylor coefficients also have physical significance: they represent the moments of the time-domain circuit response. These properties, allow the substitution of the Padé approximant in the place of the original circuit with little significant loss of accuracy, but substantial efficiency gains.

The direct computation of Padè approximations [35, 36] is numerically unstable. Therefore, the preferred methods for the computation of Padé approximations of linear systems are Krylov-subspace techniques, in particular, variants of the Lanczos [26] and Arnoldi algorithms [2] which are efficient and numerically stable. Lanczos-type algorithms, as implemented in the Padé via Lanczos (PVL) algorithm [8, 9, 14, 13], produce the most efficient approximations. For the same order of approximation and computational effort they match twice as many moments as the Arnoldi algorithm [6, 34, 42]. However, in certain cases, Lanczos-based methods may produce non-passive reduced-order models of passive linear systems. In these cases post-processing is required to enforce the desired properties.

The Krylov subspace based reduced-order modeling algorithms produce one or two small matrices. These matrices can be used either to efficiently evaluate the linear sub-block's transfer functions at any desired frequency, or to formulate a small system of linear differential equations which model its time-domain behavior, and which can be solved in conjunction with the entire RF circuit.

Recently, reduced-order modeling techniques were also applied to the noise analysis problem [7]. The benefit is a significantly more efficient evaluation of noise power over a wide range of frequencies. Moreover, the entire noise behavior of a circuit block is captured in a compact form and can be used hierarchically in system-level simulations.

Conclusion

We have described powerful new techniques for the analysis of RF circuits. Next-generation CAD tools based on such techniques should enable RF designers to obtain a more accurate picture of how their circuits will operate. These new simulation capabilities will be essential in order to reduce the number of design iterations needed to produce complex RF ICs.

References

[1] A.A. Abidi and R.G. Meyer. Noise in relaxation oscillators. *IEEE Journal of Solid-State Circuits*, December 1983.

[2] W. E. Arnoldi. The principle of minimized iterations in the solution of the matrix eigenvalue problem. *Quart. Appl. Math*, 9:17–29, 1951.

[3] H. 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 App.*, 23:311–32, 1995.

[4] H.G. Brachtendorf, G. Welsch, R. Laur, and A. Bunse-Gerstner. Numerical steady state analysis of electronic circuits driven by multi-tone signals. *Electrical Engineering (Springer-Verlag)*, 79:103–112, 1996.

[5] A. Demir, A. Mehrotra, and J. Roychowdhury. Phase noise in oscillators: A unifying theory and numerical methods for characterisation. In *ACM/IEEE Design Automation Conference*, 1998.

[6] I.M. Elfadel and D.D. Ling. Zeros and passivity of arnoldi-reduced-order models for interconnect networks. In *Proc. 34nd ACM/IEEE Design Automation Conf*, 1997.

[7] P. Feldmann and R. W. Freund. Circuit noise evaluation by pade approximation based model reduction techniques. In *Tech. Dig. 1997 IEEE/ACM International Conference on Computer-Aided Design*, November 1997.

[8] P. Feldmann and R.W. Freund. Efficient linear circuit analysis by padé approximation via the lanczos process. *IEEE Trans. Computer-Aided Design*, 14:639–649, May 1995.

[9] P. Feldmann and R.W. Freund. Reduced-order modeling of large linear subcircuits via a block lanczos algorithm. In *Proc. 32nd ACM/IEEE Design Automation Conf*, pages 474–479, 1995.

[10] P. Feldmann, R. Melville, and D. Long. Efficient frequency domain analysis of large nonlinear analog circuits. In *Proc. of CICC*, 1996.

[11] R. L. Ferrari and P. P. Silvester. *Finite Elements for Electrical Engineers*. Cambridge Univ. Pr., 1996.

[12] R. Freund, G. Golub, and N. Nachtigal. Iterative solution of linear systems. *Acta Numerica*, pages 57–100, 1991.

[13] R. W. Freund and P. Feldmann. Reduced-order modeling of large linear passive multi-terminal circuits using matrix-pade approximation. In *DATE-98*, February 1998.

[14] R.W. Freund and P. Feldmann. Reduced-order modeling of large passive linear circuits by means of the sypvl algorithm. In *Tech. Dig. 1996 IEEE/ACM International Conference on Computer-Aided Design*, pages 280–287, November 1996.

[15] Jr. G.A. Baker and P. Graves-Morris. *Padé Approximants*. Cambridge Univ. Pr., New York, NY, 2nd edition, 1996.

[16] R. Gilmore and M. Steer. Nonlinear circuit analysis using the method of harmonic balance—a review of the art. *Int. J. on Microwave and Millimeter Wave Comp. Aided Eng.*, 1, Jan. 1991.

[17] L. Greengard and V. Rokhlin. A fast algorithm for particle simulations. *J. of Computational Physics*, 73(2):325–348, December 1987.

[18] A. Hajimiri and T.H. Lee. A state-space approach to phase noise in oscillators. Technical memorandum, Lucent Technologies, July 1997.

[19] R. F. Harrington. *Field Computation by Moment Methods*. IEEE Press, New York, 1991.

[20] M. Kamon, M. J. Tsuk, and J. White. FASTHENRY: A multipole-accelerated 3-d inductance extraction program. *IEEE Trans. on Microwave Theory and Techniques*, 42(9):1750–1758, September 1994.

[21] S. Kapur and D. E. Long. IES^3: A fast integral equation solver for efficient 3-dimensional extraction. In *Proc. of the Int. Conf. on Computer-Aided Design*, November 1997.

[22] S. Kapur, D. E. Long, and J. Zhao. Efficient full-wave simulation in layered, lossy media. In *Proc. of the 1998 Custom Integrated Circuits Conf.*, May 1998.

[23] F. Kärtner. Analysis of white and $f^{-\alpha}$ noise in oscillators. *International Journal of Circuit Theory and Applications*, 18:485–519, 1990.

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

[25] K. Kurokawa. Noise in synchronized oscillators. *IEEE Transactions on Microwave Theory and Techniques*, MTT-16:234–240, 1968.

[26] C. Lanczos. An iteration method for the solution of the eigenvalue problem of linear differential and integral operators. *J. Res. Nat. Bur. Standards*, 45:255–282, 1950.

[27] D.B. Leeson. A simple model of feedback oscillator noise spectrum. *Proceedings of the IEEE*, 54(2):329, February 1966.

[28] M. Rösch and K.J. Antreich. Schnell stationäre Simulation nichtlinearer Schaltungen im Frequenzbereich. *AEÜ*, 46(3):168–176, 1992.

[29] S. Maas. *Nonlinear Microwave Circuits*. Artech House, 1988.

[30] J.A. McNeill. *Jitter in Ring Oscillators*. PhD thesis, Boston University, 1994.

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

[32] K.A. Michalski and J.R. Mosig. Multilayered media Green's functions in integral equation formulations. *IEEE Transaction on Antenna and Propagation*, 45(3):508–519, March 1997.

[33] K. Nabors and J. White. Multipole-accelerated capacitance extraction for 3-d structures with multiple dielectrics. *IEEE Trans. on Circuits and Systems*, 39(11):946–954, November 1992.

[34] A. Odabasioglu, M. Celik, and L.T. Pileggi. Prima: Passive reduced-order interconnect macromodeling algorithm. In *Tech. Dig. 1997 IEEE/ACM International Conference on Computer-Aided Design*, November 1997.

[35] L.T. Pillage and R.A. Rohrer. Asymptotic waveform evaluation for timing analysis. *IEEE Trans. Computer-Aided Design*, 9:352–366, April 1990.

[36] V. Raghavan, R.A. Rohrer, L.T. Pillage, J.Y. Lee, J.E. Bracken, and M.M. Alaybeyi. Awe–inspired. In *Proc. IEEE Custom Integrated Circuits Conference*, May 1993.

[37] V. Rizzoli, A. Costanzo, P. Ghigi, F. Mastri, D. Masotti, and C. Cecchetti. Recent advances in harmonic-balance techniques for nonlinear microwave circuit simulation. *AEÜ*, 46(4):286–296, 1992.

[38] U.L. Rohde. *Digital PLL Frequency Synthesizers: Theory and Design*. Prentice-Hall, 1983.

[39] J. Roychowdhury. Efficient Methods for Simulating Highly Nonlinear Multi-Rate Circuits. In *Proc. IEEE Design Automation Conference*, 1997.

[40] J. Roychowdhury. Analysing Circuits with Widely-Separated Time Scales using Numerical PDE Methods. *IEEE Trans. Ckts. Syst. – I: Fund. Th. Appl.*, 1998. To appear.

[41] Y. Saad. *Iterative methods for sparse linear systems*. PWS, Boston, 1996.

[42] L.M. Silveira, M. Kamon, I. Elfadel, and J. White. A coordinate-transformed arnoldi algorithm for generating guaranteed stable reduced-order models of rlc circuits. In *Tech. Dig. 1996 IEEE/ACM International Conference on Computer-Aided Design*, pages 288–294, November 1996.

[43] G. D. Smith. *Numerical Solution of Partial Differential Equations: Finite Difference Methods*. Oxford Univ. Pr., 1986.

[44] A. Taflove. *Computational Electrodynamics: The Finite-Difference Time-Domain Method*. Artech House, 1995.

[45] A. Ushida and L. Chua. Frequency domain analysis of nonlinear circuits driven by multi-tone signals. *IEEE Trans. on Circuits and Sys.*, CAS-31:766–779, Sep. 1984.

[46] T.C. Weigandt, B. Kim, and P.R. Gray. Analysis of timing jitter in cmos ring-oscillators. In *Proc. IEEE ISCAS*, June 1994.