Agilent Technologies

Appendix

Circuit Envelope Simulator

Circuit Envelope Simulator

Circuit envelope simulation offers designers an efficient way to analyze amplifiers, mixers, oscillators, and feedback loops in the presence of modulated and transient high-frequency signals. This simulator allows the sophisticated modulated signals found in today's communication circuits and subsystems to be efficiently and accurately analyzed. This simulation technology combines the advantages of both the time- and frequency-domain techniques to overcome the limitations of both harmonic balance and SPICE simulators.

Circuit Envelope Simulation Process

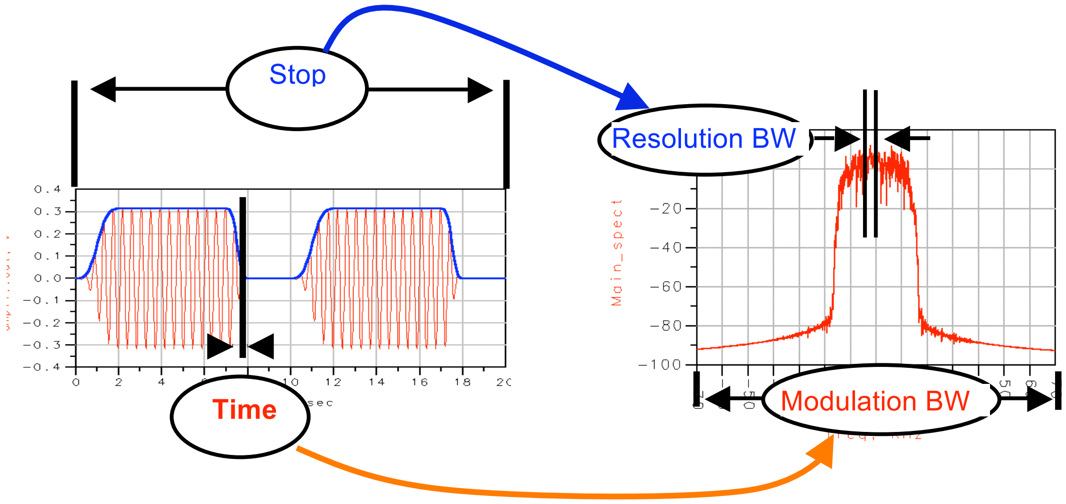
Circuit envelope simulation combines elements of harmonic balance and time-domain simulation techniques.
Like harmonic balance, circuit envelope simulation describes the nonlinear behavior of circuits and the harmonic content of the signals. Unlike harmonic balance, however, circuit envelope simulation extends over time. It is not limited to describe only steady-state behavior.
In effect, circuit envelope simulation depicts a time-varying series of harmonic balance results.

$$\boxed{v(t) = real\left(\sum_{k=o}^{N} V_k(t) e^{j\omega_k t}\right)}$$

In circuit envelope simulation, input waveforms are represented as RF carriers with modulation envelopes that are described in the time domain. The input waveform consists of a carrier term and a time-varying term that describes the modulation that is applied to the carrier.
Amplitude, phase, and frequency modulation, or combination of these can be applied, and there is no requirement that the signal be described as a summation of sinusoids or in steady-state. This makes it possible to represent digitally modulated (pseudo random) input waveforms realistically.

Step Size and Stop Time

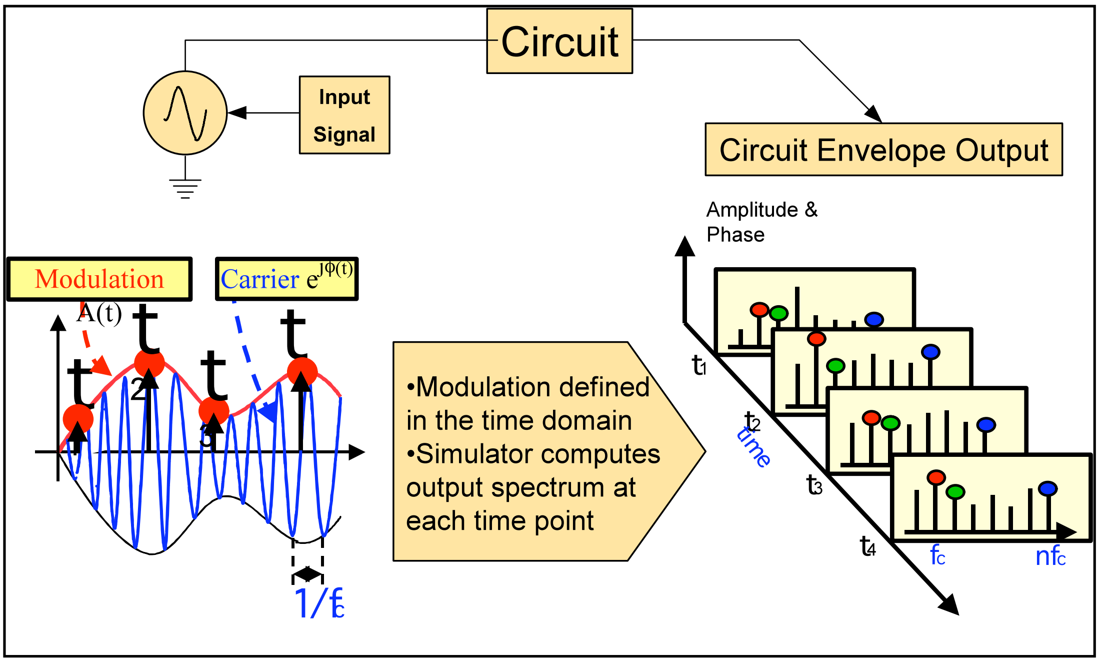
In the circuit envelope simulation, both the step size (Time step) and the total time interval of the simulation (Stop Time) affect simulation time and memory requirements. Both of these parameters affect the number of points in the simulation.
The step size can also affect accuracy if the step size is too large to sample the waveform accurately. In some circuit envelope simulations, the overall time interval must include exactly an integer multiple or multiples of a complete wave cycle to avoid mathematical errors in the Fourier analysis.



One advantage of circuit envelope over SPICE is that the time step is determined by the bandwidth of the modulation. In circuit envelope simulation, the bandwidth is relatively narrow, generally only a few Megahertz. Therefore, relatively large time steps can be used without the risk of sampling errors.
In SPICE, the time step depends on the frequency of the RF carrier frequency or its harmonics. These frequencies are very high. As a result, many more time steps must be used to cover the interval with comparable accuracy. This greatly increases computation time and memory requirements.

Simulator Theory

In a circuit envelope simulation, the input signal is defined in terms of the modulation of the carrier in the time domain, *V(t)*, and the RF carrier and its harmonics, in the frequency domain. Time steps and the total time interval of the simulation are also defined.
When the simulation is run, the spectrum at the output and at other nodes, if desired, is computed at each time point. The result makes available the amplitude and the phase of any spectral component as a function of time.



Computation of the Fourier Coefficients

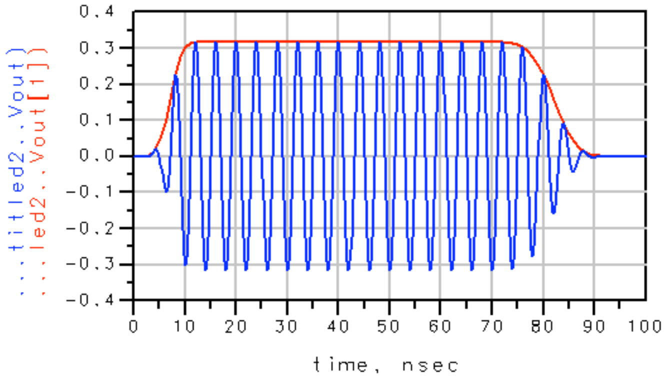
In circuit envelope simulation, the Voltage *V(t)* at any node of the circuit is given by the following equation:

$$\boxed{v(t) = real\left(\sum_{k=o}^{N} V_k(t)e^{j\omega_k t}\right)}$$

Where *v(t)* is a voltage at any node in the circuit, including the input. The Fourier coefficients, $V_k(t)$, are allowed to vary with time and may represent an arbitrary modulation of each carrier. Since each time-varying spectrum $V_k(t)$, can be thought as a modulation waveform with a center frequency f_k, these are often referred to as "envelopes". This spectrum may represent transient signals with continuous spectra, such as digital modulation envelope over a RF carrier, or periodic signals with discrete spectral lines, such as the two RF tones required for intermodulation distortion analysis.
In ordinary harmonic balance simulation, the coefficients are complex constants.
When a circuit envelope simulation is run, the simulator samples the modulation envelope of the carrier in time and calculates the discrete spectrum of the carrier and its harmonics at each time point over the time interval. If the circuit includes frequency mixing, intermodulation terms are also computed at each time point.
In harmonic balance simulation, only one spectrum, the steady-state spectrum, is computed.

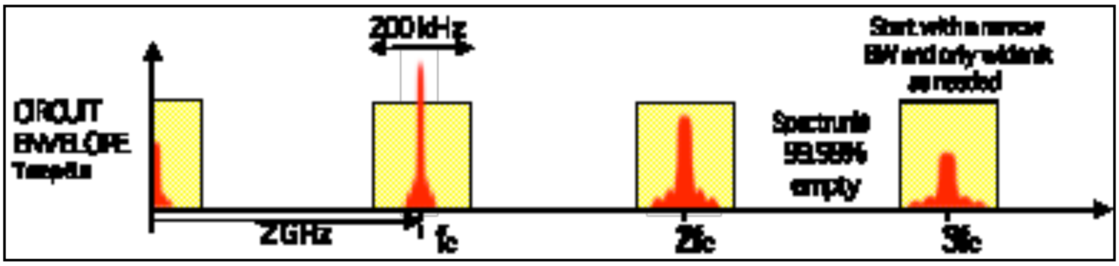
Circuit Envelope vs Carrier

Circuit envelope simulation is computationally efficient, both in time and in memory use. The modulation waveform is described directly in the time domain rather than as a sum of sinusoids. The sampling rate (time step) is governed by the relatively narrow modulation bandwidth rather than by the RF signal frequency.
In harmonic balance simulation the modulation must be described as a sum of sinusoids. The more complicated the modulation waveform is, the larger must be the number of sinusoids that are summed. The result, especially when higher-order products must also be considered, is that the amount of memory required for the simulation can easily exceed the amount available.


Circuit Envelope simulation shows the envelope of the modulated waveform not the carrier

In SPICE, the signal must be sampled at a rate fast enough to capture the highest frequency present. Because this frequency can be very high, many short time steps are required, and this can require much time and memory for the simulation. If only the modulation envelope is of interest, circuit envelope simulation is a better choice.

In an Envelope simulation each node voltage is represented by a discrete spectrum having time varying Fourier coefficients. The set of spectral frequencies is user-defined; the amplitude and phase at each spectral frequency can vary with time. The signal representing the harmonics is no longer limited to a constant, as it is in harmonic balance. Each spectral frequency can be thought of as a center frequency of a spectrum; the width of each spectrum is *± 0.5/Time step*.

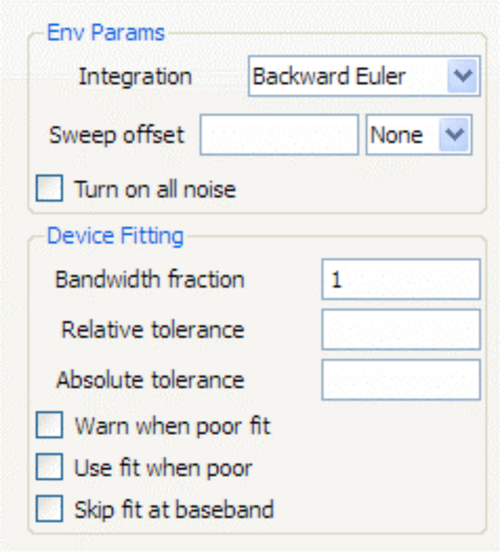


In most cases the bandwidth of the modulation signal is much smaller than the lowest user-defined spectral-frequency (which corresponds to the "carrier" frequency). The bandlimited signal within each spectrum can contain periodic, transient, or random tones.

One limitation of time domain based simulators, like SPICE, is that the accuracy of any numerical integration routine degrades at higher frequencies, relative to their low frequency or DC accuracy. For example, using a standard trapezoidal second order integration technique, just to compute the current through a capacitor at a given frequency requires that the sample rate be 10 times the carrier frequency for one percent accuracy and about 32 times the carrier frequency for 1 percent accuracy. For modulation carrier applications, the best accuracy should be at the carrier frequency, not DC, since that is where the information is centered. For these time domain simulators, the accuracy is best at DC and falls off at higher frequencies.

For the envelope simulator, multiple integrations are being performed with each centered around a carrier frequency. So the best accuracy is achieved right at each carrier, and any decrease due to time domain integration only applies to the outer modulation frequencies around each carrier. The envelope time step must be set small enough to maintain the desired accuracy of the envelope spectrum, but the accuracy at and very close to the actual carriers are equal to the steady-state, full accuracy of the basic harmonic balance simulator.

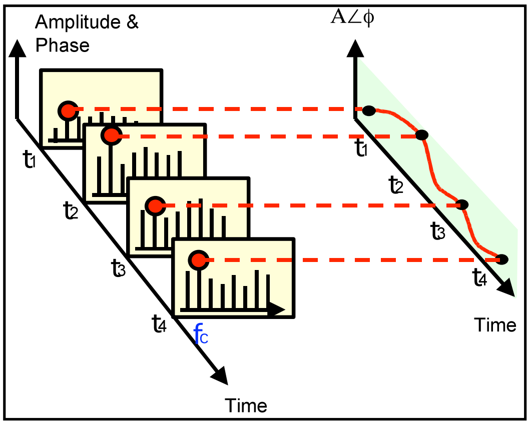
Another advantage the circuit envelope simulator is that the instantaneous amplitude, phase, and I/Q information of each carrier is immediately available. Therefore, it can be used to create efficient models of magnitude, power, and phase for various modulation formats. No additional, lengthy, or error introducing calculations are required to convert the time domain waveform into envelope information.


Env Params
Integration: Backward Euler
Sweep offset: None
☐ Turn on all noise
Device Fitting
Bandwidth fraction: 1
Relative tolerance:
Absolute tolerance:
☐ Warn when poor fit
☐ Use fit when poor
☐ Skip fit at baseband

But this also means that it can start from the steady-state solution. It can start its simulation with the entire carrier, LO, RF, and DC waveforms at their steady state value. There is no need to wait, as with most time domain simulators, for the circuit to achieve steady state. Therefore the simulator can start its simulation of the modulated envelope, as it varies with time, without having to wait for the transients to die out. However, it doesn't share the limitation of harmonic balance's inability to simulate these start-up transients, if desired. The RF, LO, and even the DC sources can be turned on at time t=0, so that the full start-up transients can be observed. Due to the combined use of time and frequency domain techniques, this envelope simulator shares most of the advantages of both time and frequency domain simulators.

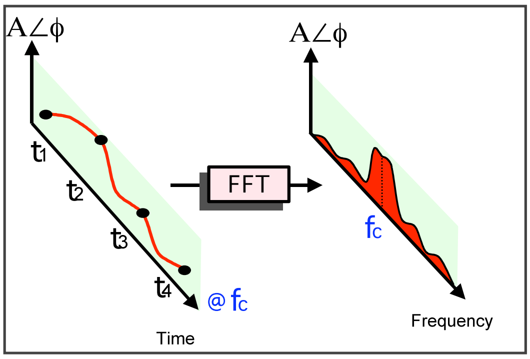
Time-Domain Outputs

In circuit envelope simulation, the discrete time-varying spectrum of the carrier and its harmonics is simulated at each time point over the time interval. If the circuit includes mixing, then inter-modulation terms are also computed at each time point.
Amplitude and phase data at each time point in the simulation is then read into the dataset.
These results, in the time domain, show amplitude, phase, I/Q, frequency, and harmonics as a function of time for the output and any other node of the circuit if desired.



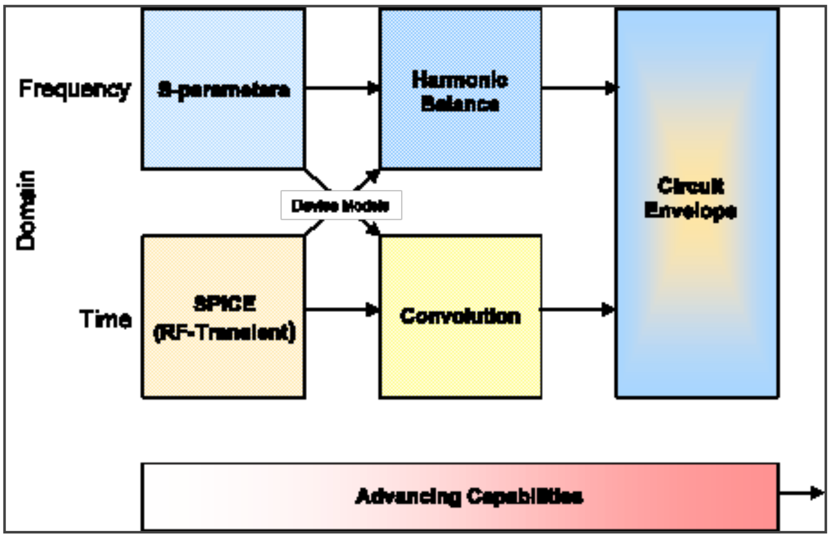
FFT Outputs

By taking the Fourier transform of the amplitude and phase data from the simulation of any spectral component (for example the fundamental), frequency domain results around that spectral component can be presented.
The Fourier transform is used (in effect) to convert the amplitude and phase data from the simulation back into the frequency domain.
This makes it possible to examine results such as spectral regrowth, adjacent-channel power, intermodulation distortion, and phase noise.



Circuit Envelope Simulation Technology

Circuit envelope simulation advances HP EEsof's circuit simulation capabilities by combining time domain and harmonic balance techniques. It is especially useful in examining transient behavior and responses to digitally modulated signals.
In the frequency domain, S-parameter analysis accounts for linear effects, and harmonic balance simulation accounts for nonlinear effects. Both simulation techniques show steady-state behavior.
Circuit envelope simulation extends harmonic balance into the time domain. It allows designers to examine time-varying output spectrums and transient behavior including harmonics. In the time domain, SPICE is effective for lumped elements, and convolution simulators are effective for distributed as well as lumped elements.
Circuit envelope simulation extends these techniques by allowing analysis of distributed elements and by decreasing computation time and memory requirements.



As a simulation tool, circuit envelope simulation has many of the same capabilities that other tools have. Like other tools, circuit envelope simulation overlaps with and adds to the capabilities of other simulators. It does not completely replace them. Nor do other simulation tools offer everything that circuit envelope simulation offers.

- Circuit envelope simulation can do all the same simulations that harmonic balance can do. For unmodulated signals, however, harmonic balance is more efficient.
- Circuit envelope simulation can do most of the same simulations that SPICE can do. For arbitrary loops and large logic and analog circuits, however, SPICE is more efficient.

Circuit envelope simulation has a wide range of applications. It is most useful in analyzing transient and modulated high-frequency signals, and it can be used on a wide range of circuits.

Among the most important types of circuits are the following:

Amplifiers

Amplifier start-up transient, including harmonics, can be analyzed as a function of time, and in response to pulsed and digitally modulated RF signals. Spectral regrowth and adjacent-channel power can also be studied.

Harmonics can be analyzed in the time domain only when circuit envelope simulation is used. Harmonics cannot be analyzed in the time domain using SPICE or harmonic balance.

Mixers

Mixer nonlinearities can be characterized in terms of third-order intercept (TOI or IP3) and higher-order intercepts. This is done by using cosine modulation to split the single RF input tone into two signals which are spaced at twice the modulation frequency. As with amplifiers, spectral regrowth and adjacent-channel power can also be studied.

Oscillators

Oscillator start-up behavior can be analyzed easily using circuit envelope simulation. Among these analyses are start-up time and amplitude and frequency as functions of time. It is also possible to sweep control voltage while examining oscillator frequency as a function of time.

Phase-Locked Loops

In phase-locked loops, frequency settlings as a function of time and noise spectra can both be analyzed.

Additional Applications

Circuit envelope can also be used in various other applications. Among these are time domain simulation of nonlinear noise and time domain optimization.