The use of Harmonic Balance in Wave Concept Iterative Method for Nonlinear Radio Frequency Circuit Simulation

Hicham MEGNAFI, Noureddine BOUKLI-HACENE
Telecommunication Laboratory of Tlemcen
University of Abou Bakr Belkaid
Tlemcen, Algeria

Nathalie RAVUE, Henri BAUDRAND
Laboratory of Laplace
LAPLACE, University of Toulouse, CNRS
Toulouse, France

Abstract—This paper presents the birth of the new hybrid method for the non-linear Radio frequency circuits' simulation. This method is based on the combination of the wave concept iterative procedure (WCIP) and the harmonic balance (HB) for their advantages. It consists also the development of an application based on this method for the simulation of nonlinear planar radio frequency circuits. The simulation of the Radio frequency diode implemented in micro-strip line is done. The validations are obtained by the comparison of the results obtained by our new hybrid method and Harmonic Balance under Advanced Design System (ADS) software.

Keywords—WCIP; harmonic balance; nonlinear circuits; planar radio frequency circuits; radio frequency diode

I. INTRODUCTION

The Technological advances have enabled to establish an efficient microwave circuit. To analyze and simulate them in nonlinear regime, several methods of analysis have been implemented to achieve the required performances [1], [2], the basic differences between these methods that exist in the domain of analysis are temporal or frequency.

The nonlinear system response is periodic that can be determined by integrating the differential equations that govern the circuit to steady state.

The Time-domain analysis, if it allows to take in consideration a high number of non linearties, then it is not suitable for circuits having relatively large time constant elements in front of the period of the applied signal. In the other hand, if this approach allows the distortions study, it only applies to circuits with low nonlinearties [3].

To remove the main problems caused by the two previous methods, an alternative approach has been proposed and become the most commonly used method in the analysis of nonlinear circuits. This method is called the harmonic balance (HB) [4].

The basic principle of the harmonic balance is to split the circuit into two sub-circuits; a linear and nonlinear sub-circuits. The linear sub-circuit is necessary to be studied in the frequency-domain and the non-linear sub-circuit is well described in the time-domain. The results of the time-domain analysis are then converted into the frequency-domain by the Fourier Transform method [4]. The convergence is obtained, only if the interconnection currents between the linear and nonlinear sub-circuits, for each harmonic, are the same. The different currents must be balanced for each harmonic [5].

In the other hand, another method that has already shown its effectiveness in the simulation of RF circuits is called the wave concept iterative process (WCIP); this method is based on the concept of waves. It’s applied in the planar structures of resolution [6], the micro-strip antennas [7] and the planar filters [8].This method has the advantage of simplicity because it does not involve the use of basic functions and matrix inversion as in other calculation methods. Therefore, it is possible to analyze a large complex planar microwave structures [9], [10]. Moreover, a high computational speed can be achieved by using the two dimensional the fast Fourier transformation algorithm known as fast modal of transformation (FMT) [11].

The classical wave concept iterative process was not applied in the simulation of nonlinear circuits because this method is based on frequency domain analysis, for which we proposed a combination of HB and WCIP to solve the problem of limited simulation; HB and WCIP are based on the use of the Fourier transform and its inverse. The balance between the two domains related to the Fourier transform is fundamental to develop a model of nonlinear electromagnetic problems and to qualify circuits in microwaves.

The rest of the paper is organized as follows: The next one (Section II) contains a definition of the wave concept, then we describe the iterative process and its algorithm. We are going to show the interest and the principle of the harmonic balance and its algorithm. In Section III, we propose a new approach of WCIP (hybrid method of HB and WCIP) to get the advantages of the two methods for the simulation of nonlinear circuits, we propose also a nonlinear RF diode modeling in small signal regime to validate this new hybrid approach. Where in Section IV, we present the obtained results validated by the comparison with the results obtained by ADS software. And in the last Section V, we present the perspective of this new hybrid method as an improvement of the convergence and the simulation of new non-linear circuit (such as transistor, non-linear capacitor) and a conclusion.
II. METHODS

A. Wave Concept Iterative Method

The main goal of this part is to propose a global electromagnetic analysis for characterizing microwave planar linear circuits based on the use of the iterative method. This method is based on the wave concept. It has the advantage of simplicity because it does not involve the use of basic functions and matrix inversion as in other calculation methods. Therefore, it is capable of analyzing large complex planar microwave structures [12].

1) Waves Definition

The wave concept is introduced by writing the transverse electric field $\vec{E}_i$ and transverse current density $\vec{J}_i$ in terms of incident $\vec{A}_i$ and reflected $\vec{B}_i$ waves in each medium $i$. It leads to the following set of equations [10], [11]:

$$\begin{align*}
\vec{A}_i &= \frac{1}{2Z_0} (\vec{E}_i + Z_0 \vec{J}_i) \\
\vec{B}_i &= \frac{1}{2Z_0} (\vec{E}_i - Z_0 \vec{J}_i)
\end{align*}$$

(1)

Where, $Z_0$ is an intrinsic impedance of the medium.

B. Principle of Wave Concept Iterative Method

According to the criterion defining the wave concept, we can define the electric field and the current density values in each point of the surface $\Omega$ by the determination of the incidental and the reflective waves values. So, the iterative process is based on the creation of a recurrence relation between the incidental and the reflective waves values. Therefore, the iterative method can define the electric field and the current density values in each point of the surface $\Omega$ by the determination of the reflection operator representing the half medium around the interface $\Omega$, and transverse current density.

The iterative method can be summarized by the following equations system [3], [4]:

$$\begin{align*}
\vec{A} &= \hat{S} \vec{B} + \vec{A}_0i \\
\vec{B} &= \vec{F} \cdot \vec{A}
\end{align*}$$

(2)

Where,

$\hat{S}$: The transmission operator at the interface $\Omega$,

$\vec{F}$: The reflection operator representing the half medium around the interface

$\vec{A}_0i$: The source wave that can be defined by:

$$\vec{A}_0i = \frac{\vec{E}_0}{\sqrt{2}\,\omega_0}$$

(3)

$\vec{E}_0$: The total electric field produced by the excitation source.

The first system equation (2) describes the electromagnetic behavior at the circuit interface in spatial domain, while the second system equation (2) describes the waves’ reflection in modal-domain.

The iterative process use successively these two equations through a Fast Modal Transform (FMT: from spatial domain to modal domain) and its inverse (conversion from the modal domain to the spatial one) [13].

Wave propagation described by incident and reflected waves, is presented in the planar structure. We can see that the waves will be reflected continuously, as shown in Fig. 1 [14].

![Fig. 1. Wave propagation in planar circuit [5].](image)

1) Wave Concept Iterative Procedure

The procedure of the wave concept iterative method is summarized by the following steps which are shown in Table I:

<table>
<thead>
<tr>
<th>TABLE I: STEPS OF WAVE CONCEPT ITERATIVE PROCEDURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steps</td>
</tr>
<tr>
<td>-------</td>
</tr>
<tr>
<td>First step</td>
</tr>
<tr>
<td>Second step</td>
</tr>
<tr>
<td>Third step</td>
</tr>
<tr>
<td>Fourth step</td>
</tr>
<tr>
<td>Fifth step</td>
</tr>
<tr>
<td>Sixth step</td>
</tr>
<tr>
<td>Seventh step</td>
</tr>
</tbody>
</table>

C. Harmonic Balance Method

The Harmonic balance is a hybrid time and frequency domains analysis technique for simulating nonlinear circuits and systems. This hybrid analysis allows all the advantages of nonlinear temporal domain modeling, combined with the strength (efficiency) of the steady-state frequency technique [5].

1) The Standard Harmonic Balance Technique

The basic principle of HB approach is to split the circuit into two sub-circuits of $N$ common ports between the linear and non-linear sub-circuits. Each branch of the circuit circulates a current harmonic component [15], [16]. In this work, the nonlinear sub-circuit consists in a nonlinear diode represented by an algebraic equation $it(t) = F(v(t))$, so the circuit is decomposed as is shown in Fig. 2.
The currents flowing from nodes into linear sub-circuit \( \bar{I}_k \), including all distributed elements, are calculated by means of a frequency-domain linear analysis. And the currents from nodes into nonlinear sub-circuit \( \hat{I}_k \) are calculated in the time-domain. The balance between the time domain and frequency domains is obtained by Fourier transform; it is necessary to use a sinusoidal excitation to apply this technique. According to Kirchoff’s Current Law (K.C.L), the currents sum should be zero at all nodes (4) [17].

\[
\bar{I}_k(k, \omega_0) + \hat{I}_k = 0 \quad \forall k \in [0, \ldots, K]
\]

(4)

Where, \( K \) is the number of harmonic, \( \omega_0 \) is the pulsation of the source.

The currents’ sum computation gives the error of the method, called \( F_{\text{error}} \) (5). If the method converges (i.e. if the error function is driven to a given small value) then the resulting voltage amplitudes and phases approximate the steady-state solution [17], [18].

\[
F_{\text{error}} = \bar{I}_k(\omega) + \hat{I}_k(\omega) \quad \forall k \in [0, \ldots, K]
\]

(5)

Where; \( N \) is the number of harmonic, \( \omega \) is the pulsation of harmonic for each node.

2) Algorithm of Harmonic balance

The procedure of harmonic balance is summarized by the following steps which are shown in Table II:

![Image](image.png)

**Fig. 2.** The Harmonic Balance method split the circuit into linear and nonlinear subcircuits.

**TABLE II. STEPS OF HARMONIC BALANCE PROCEDURE**

<table>
<thead>
<tr>
<th>Steps</th>
<th>Development</th>
</tr>
</thead>
<tbody>
<tr>
<td>First step</td>
<td>Initial the estimation of ( V_0(t) ) (can be also in the frequency domain), we can estimate the initialization of the excitation by pulses with a broad spectral content or by a value equals to zero.</td>
</tr>
<tr>
<td>Second step</td>
<td>Apply nonlinearity in the time-domain, ( i(t) = f(v(t)) )</td>
</tr>
<tr>
<td>Third step</td>
<td>Convert the current in the time-domain to the frequency domain by the FFT.</td>
</tr>
<tr>
<td>Fourth step</td>
<td>In the frequency-domain, Check if the harmonics are balanced using the cost, (the use of Kirchoff’s Current Law equation).</td>
</tr>
<tr>
<td>Fifth step</td>
<td>Still in the frequency-domain, If the convergence criteria were not satisfied, updated the voltages using the cost function and its Jacobian. ( v(t)^{k+1} = v(t)^k - J^{-1}(v(t)^k) )</td>
</tr>
<tr>
<td>Sixth step</td>
<td>Convert the voltage in the frequency-domain to the time-domain by the FFT-1.</td>
</tr>
<tr>
<td>Seventh step</td>
<td>Repeat the second step to the sixth step until the convergence is done.</td>
</tr>
</tbody>
</table>

**Fig. 3.** Algorithm working principle.

III. NEW HYBRID APPROACH FOR NONLINEAR RADIO FREQUENCY CIRCUIT SIMULATION

In this part, we present the new approach of WCIP (hybrid HB-WCIP); it is a combination between WCIP classical and HB that we have proposed as well as their algorithm (hybrid HB-WCIP algorithm).

A. Principle

In the WCIP, a single frequency is used for the linear circuit excitation, so we can say that the wave is defined in space (pixel) and in the frequency at the circuit interface, hence the study is spatial-frequency at the circuit interface. To simulate the non linear circuit, the principle of HB is used to split the circuit into two sub-circuits at its interface; the linear sub-circuit is simulated by the classical iterative method, while the non-linear sub-circuit is solved by a temporal resolution. In this way, we took advantage of the WCIP in the simulation of the linear part of the circuit, and the principle of the harmonic balance in the simulation of the non-linear part.
B. Algorithm

The use of harmonic balance in the wave concept iterative method can be summarized by the diagram illustrated in Fig. 4.

![Harmonic Balance Diagram](image)

C. Nonlinear Radio Frequency Diode Modeling for Small Signal Regime

As it is noted previously, the modeling of nonlinear sub-circuit that we want to simulate is essential. In this part, we take a diode as an example of a nonlinear sub-circuit, and we propose a diode model that functions in the small signal regime.

The characteristic equation of a diode is given by the following formula [20]:

$$I(V) = I_s(e^{\alpha V} - 1)$$  \hspace{1cm} (6)

With: $V$ is the voltage across the diode junction (volts)  
$I_s$ Saturation Current (amperes)  
$\alpha = \frac{q}{\eta k T}$ \hspace{1cm} (7)

$q$ is the electron charge = 1.6*10^{-19}(coulomb),  
$\eta$ is diode Ideality Factor (generally between 1.1 and 1.2),  
$k$ is Boltzmann’s constant = 1.38*10^{-23} (joule/°K),  
$T$ is temperature in degrees Kelvin (°K).

Current flowing through the diode obtained by (6) contains two components, alternative $i(t)$ and continuous $I_0$, for this the total current $I = I_0 + i(t)$, the same for the voltage, $V = V_0 + v(t)$. Where $v(t)$ is the alternating voltage that consists of $n$ harmonics, hence the formula of the total voltage is as follows:

$$V(t) = V_0 + V_1 \cdot \cos(\omega_1 t) + \cdots + V_n \cdot \cos(n \cdot \omega_0 \cdot t)$$  \hspace{1cm} (8)

Where, $V_n$ is the nth harmonic component of the voltage.

In the approximation small signal (i.e.: $i(t) << I_0$), the diode current can be developed in Taylor’s series:

$$I(V) = I_0 + \frac{V}{V_0} \frac{di}{dv} + \frac{V^2}{2V_0^2} \frac{d^2i}{dv^2} + \cdots + \frac{V^n}{n!V_0^n} \frac{d^ni}{dv^n}$$  \hspace{1cm} (9)

with

$$\frac{di}{dv} = G_d, \frac{d^2i}{dv^2} = \alpha G_d, \frac{d^3i}{dv^3} = \alpha^2 G_d$$  \hspace{1cm} (10)

Where, $G_d$ is the dynamic conductance of the diode junction when the two different frequency voltages are applied across the diode.

The formula of current (9) will be:

$$I(V) = I_0 + G_d \left( \frac{V}{V_0} + \frac{V^2}{2V_0^2} \right) + \alpha G_d \left( \frac{V^3}{3V_0^3} + \cdots + \frac{V^n}{n!V_0^n} \right)$$  \hspace{1cm} (11)

To simplify the current (9) we use the Pascal triangle formula (11) to solve the current equation.

$$(a + b)^n = \sum_{k=0}^{n} C_n^k \cdot a^{n-k} \cdot b^k$$  \hspace{1cm} (12)

After calculating, the current formula is described as follows:

$$I(V) = I_0 + G_d \sum_{k=1}^{n} k \frac{a^{k-1}}{k!} \left[ V \right]_{m=0}^{k-2} \left[ C_m^{k-2} \cdot r_m^{k-2-k-1} \cdot \left[ V \right]_{m=1}^{k-1} \right]$$  \hspace{1cm} (13)

With $C_n^m$ is the coefficient obtained by the Pascal triangle, $n$ is the harmonic number, $V_m^n$ is the nth harmonic component of the voltage for $m$ iterations.

So we can implement this formula in the HB-WCIP algorithm, so that the RF diode can be simulated in the nonlinear regime.
IV. NUMERICAL RESULTS

The validation of the new approach algorithm of WCIP (hybrid HB-WCIP) can be confirmed by the simulation of a diode RF implemented in micro-strip line.

The RF diode model already proposed in Section C of Section III, works in the regime of small signals because we have used Taylor's development. Hence the input power of the source must be low enough to keep this regime.

The micro-strip line is deposited on a relative electric substrate permittivity $\varepsilon_r = 9.6$, thickness 0.635mm, and width $W = 0.375mm$. The ambient environment is air, therefore the relative electric substrate permittivity $\varepsilon_1 = 1$.

The structure shown in (Fig. 5) is enclosed in a box with electrical walls. The box dimensions and heights of the two medium are described as follows:

$$h_1 = 4mm, h_2 = 0.635mm, a = 6mm \text{ et } b = 6mm.$$

![Fig. 5. Radio frequency diode implemented in micro-strip line.](image)

The supply source type of this structure is bilateral and polarized along the axis Ox, and characterized by internal impedance $Z_0$ considered as the paralleling of the characteristic impedances of the two medium in which is delivered, i.e.:

$$Z_0 = Z_{01} // Z_{02} = \frac{Z_{01} \cdot Z_{02}}{Z_{01} + Z_{02}} = 90.57 \text{ Ohm}$$

To stay in small signal regime, we set the power of the source at -20 dbm. The frequency of the source is 2 GHz.

A mesh of 64x64 pixels has been chosen to approach the real dimensions (for a good description of the electromagnetic field in the micro-strip line, a mesh of 4 to 5 pixels in the width of the line must be established, one or two pixels for each end describe the boundary conditions, and 1 and more pixels in the middle to express its value inside the line).

The indicator functions of each medium (source, metal, dielectric and diode) are shown in the above (Fig. 6):

- Saturation Current $I_s : 5.10^{-8} \text{ A}$
- Diode Ideality Factor $\eta : 1.05$
- Temperature $T : 300 \degree K$

To validate the results obtained by the new algorithm implementation of WCIP (hybrid HB-WCIP) in Matlab, we compared these results with ADS software. The following figure shows the RF diode simulation implemented in the micro-strip line with these features under ADS software (Fig. 7).

![Fig. 7. Simulation by ADS software.](image)

With a maximum harmonic number equals to 5, the implementation results of the new algorithm (hybrid HB-WCIP) as obtained by ADS is described in the following figure (Fig. 8):

![Fig. 8. The indicator functions of all mediums.](image)
We can consider that the harmonic values of the output voltage less than -100 dbm are negligible noise.

In Table III, we can see clearly that the harmonic values of the output voltage obtained by ADS software are stable at a maximum harmonic number of three, hence three harmonic is necessary to present the output voltage of the diode. While in the HB-WCIP requires a number of harmonic equals to five. On the other hand, the values of the harmonics of the output voltage obtained by the two methods are really comparable between them.

According to the results obtained by the previous simulations, we can conclude that the simulation results are really comparable with those obtained by HB under ADS software. The small difference between the harmonic number is necessary to reach the convergence returns to the techniques of analysis used in each method, as well as WCIP, taking in consideration the box effect, on the other hand ADS neglects this effect.

V. CONCLUSION AND FUTURE SCOPE

In this study, we have started with the presentation of the non-linear circuits’ simulation interest, then the advantages and disadvantages of analysis methods of these circuits. After that we have presented the iterative method concept as well as the harmonic balance method. After that, we have proposed a new WICP approach (a hybrid HB-WICP) for the circuit non-linear simulation joined with RF model diode to validate this approach. The validations are obtained by the comparison of the results obtained by our new hybrid method and HB under ADS software.

Therefore, the future scope of the proposed approach can be developed in the minimization of the time convergence. We can also propose other model of non-linear circuit for small signal regime and also for large signal regime such as transistor, non-linear capacitor.

REFERENCES


Fig. 8. Radio Frequency diode simulation results implemented in the microstrip line by the change of the harmonic number used equals to 5.

The figure shows clearly the simulation with the new approach of WCIP (HB-WCIP) and the results obtained by ADS software are comparable, except in the last harmonics (harmonic 4 and 5). This difference comes back to the techniques of analysis used in each method.

A second comparison is needed between the new method (HB-WCIP) and HB implemented under ADS Software to study the convergence of results; the necessary harmonics number to reach a best presentation of the output voltage at the diode. For this, we simulate the RF diode implanted in the micro-strip line by HB-WCIP under Matlab and by ADS software for different number of harmonic.

The following table (Table III) shows the values in dbm of the output voltage harmonics (Fig. 7) obtained by the two methods; HB-WCIP and HB under ADS Software, for the number of harmonic used from 1 to 7.

<table>
<thead>
<tr>
<th>Number of harmonic used in each simulation</th>
<th>Number of harmonic (dbm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1st</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>-</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>4</td>
<td>-</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>-</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>-</td>
</tr>
<tr>
<td>ADS</td>
<td>-</td>
</tr>
<tr>
<td>HB-WCIP</td>
<td>-</td>
</tr>
<tr>
<td>7</td>
<td>-</td>
</tr>
</tbody>
</table>

TABLE III. RF DIODE SIMULATION RESULTS IMPLEMENTED IN THE MICRO-STRIP LINE BY THE CHANGE OF THE HARMONIC NUMBER USED

![Graph showing Diode RF implemented in microstrip line Simulation](image-url)


