Performance Evaluation of Analog Systems Simulation Methods for the Analysis of Nonlinear and Chaotic Modules in Communications

J. C. Chedjou, K. Kyamakya
Institute for Smart-Systems Technologies, University of Klagenfurt, Austria
e-mails: jean.chedjou@uni-klu.ac.at, Kyandoghere.Kyamakya@uni-klu.ac.at

Van Duc Nguyen
Faculty of Electronics and Telecommunications, C9-P403 Hanoi University of Technology
e-mail: van_duc_nguyen@yahoo.com

Ildoko Moussa, J. Kengne
Doctoral School of Electronics, Information Technology, and Experimental Mechanics (UDETIME), University of Dschang, Dschang, Cameroon
e-mail: kengnemozart@yahoo.fr

Abstract—The revolutionary idea of setting Analog Cellular Computers based on Cellular Neural Networks systems (CNNs) to change the way analog signals are processed is a proof of the high importance devoted to the analog simulation methods. This paper provides basics of the methods that can be exploited for the analog simulation of very complex systems (an implementation on chip using CNN technology is possible even on FPGA). We evaluate the performance of analog systems simulation methods. These methods are applied for the investigation of the nonlinear and chaotic dynamics in some modules of communication systems. We list some problems encountered when using this approach and propose appropriate techniques to tackle them. The overall motivation is to encourage the research community to use analog methods for systems’ analysis, despite the strong focus on the numerical approaches that kept analog simulation alternatives in the dark during the past decades. Both advantages and limitations of the analog modelling schemes are discussed versus those of their numerical counterpart. To illustrate the concepts, a communication module consisting of a shunt type Colpitts oscillator is considered. The electrical structure of the oscillator is addressed and the modeling process is performed to derive the equations of motion. The numerical analysis is carried out to obtain various bifurcation diagrams showing scenarios leading to chaos. Both PSPICE based simulations and laboratory experimental realizations (with analog circuits) are considered to validate the modeling and to confirm the numerical results. Near-sinusoidal oscillations, sub-harmonics and chaos are observed. The bifurcation study reveals that the system moves from near-sinusoidal regime to chaos via the usual paths of period doubling and sudden transitions. One of the interests of this work (amongst many others) is to prove that the analog systems simulation approach is more suitable than its numerical counterpart for the analysis of the striking and complex dynamics of non-linear parts/module of communication systems.

Keywords—Communication Systems; Shunt Colpitts Oscillator; Bifurcation; Chaos; Analog Systems Simulation Methods

I. INTRODUCTION

The last decade has witnessed a tremendous attention on the effects of nonlinearity in sinusoidal oscillators [1-23]. The interest devoted to these effects is explained by the rich and complex behaviour the oscillators can exhibited in their nonlinear states and also by the various technological and fundamental applications of such oscillators. Indeed, in their nonlinear states these oscillators can be exploited in many applications such as measurement, instrumentation and telecommunications. In their regular states, the oscillators can be used for instrumentation and measurements while the chaotic behaviour (irregular state) exhibited by the oscillators can be used in chaotic secure communication [24] just to name a few.

Concerning either shunt [1-6] or non-shunt [7, 8] structures of the Colpitts oscillators, some interesting works have been carried out. Reference [1] does consider an analytical approach based on asymptotic method to analyse the dynamics of the Colpitts oscillator. Bifurcation scenarios are obtained numerically to confirm the richness of the modes exhibited by the Colpitts oscillator. The extreme sensitivity of this oscillator to tiny changes in its parameters is shown. Reference [2] deals with the observation of chaos in the Colpitts oscillator. A model (set of equations) describing the autonomous states of the oscillator is proposed. A piecewise-linear circuit model is considered. Chaotic behaviour is observed numerically and experimentally. The authors of reference [3] focused on the relationship between chaotic Colpitts oscillator and Chua’s circuit. They showed that the Colpitts oscillator might be
mapped to the Chua's oscillator with an asymmetric nonlinearity. Some bifurcation structures of the oscillators are obtained and the structure under consideration exhibits striking phenomena amongst which period doubling scenarios to chaos are observed. Reference [4] develops a methodological approach to the analysis and design of a Colpitts oscillator. A nonlinear approach for the two quasi sinusoidal and chaotic operating modes was considered. In particular, the generation of regular and irregular (chaotic) oscillations in terms of the circuit parameter was shown. Reference [5] considers non-smooth bifurcations in piecewise-linear model of the Colpitts oscillator. An approximate 1D map was proposed for predicting border collision bifurcation (common in power electronics) of the Colpitts oscillators. In reference [6] one considers the modelling of chaotic generators using microwave transistors. The transition from a simplified mathematical model to a model of RF chaotic source or microwave band is discussed. Reference [7] exploits a Lur'e system form to clarify the occurrence of chaos in the Colpitts oscillator. Components of the autonomous chaotic Colpitts oscillator causing the variation of equilibrium points are identified. This study was extended by the same authors to the case of the forced (non-autonomous) Colpitts oscillator. Both amplitude and frequency of the external excitation were used for chaos control in the oscillator. The dynamic maps of locking, transition, and normal areas, with their related frequencies and output powers, were depicted by measurements. Simulation and experimental results of injection-locked behaviour were discussed and presented.

The works summarised above show the Colpitts oscillator (either a shunt or non-shunt structure) as a chaotic generator. It appears that the non-shunt structure of the Colpitts oscillator has been intensively considered while the literature is very poor concerning information related to the shunt structure of such an oscillator. It has been shown that tiny changes (imperfection or instability in the shunt structure of the oscillator) in the parameters values can generate the chaotic behaviour of the oscillator. One of the advantages of the shunt Colpitts oscillator (amongst many others) can be found in practical realisation. Indeed, this structure is simple to be realised. Moreover, the good stability of the fundamental characteristics (that is the amplitude and phase) of the waveforms generated by such a structure is due to the fact that the biasing current is not flowing in the oscillatory network as observed in the non-shunt structure. These are some advantages of the shunt structure of the Colpitts oscillator.

This paper considers the shunt structure of the Colpitts oscillator. To the best of our knowledge approximate analytical results available in the literature concerning such a structure are obtained from analysis tools based on Lur'e system forms. These models were used to study the stability conditions of regular oscillations and the possible appearance of chaos in the shunt structure of the Colpitts oscillator. Nevertheless the literature does not propose a direct modelling of the shunt structure of the Colpitts and does not analyse the chaoticity (degree of chaos) of the oscillator from the real model of the oscillator. Our aim in this paper is to list some difficulties that can be faced when performing analog experimental simulations and propose appropriated methods to tackle them. We also contribute to the general understanding of the behaviour of the shunt structure of Colpitts oscillator and complete the results obtained so far by a) carrying out a systematic and methodological analysis of its nonlinear dynamics; b) providing both theoretical and experimental (analogue) tools, which will be of precious use for design and control engineers since they can be used to get full insight of the nonlinear dynamical behaviour of the oscillator; c) pointing out some of the unknown and striking behaviour the shunt Colpitts oscillator.

One of the traditional properties of sinusoidal oscillators is the possibility of adjusting the frequencies of the waveforms generated starting from RC or LC resonators components. Nevertheless such an operation becomes very delicate or obsolete when the effects of nonlinearity are taken into account, since a rigorous analysis shows the strong dependence upon nonlinearity of the fundamental characteristics (that is both amplitude and frequency) of the waveforms generated. This can clearly be demonstrated by performing a rigorous analysis to express the fundamental characteristics of the waveforms generated, in terms of the parameters of the bipolar junction transistor responsible of the nonlinearity phenomena observed in the structural behaviour of such oscillators.

The structure of the paper is as follows. Section 2 presents the theoretical methods versus analogue methods. An evaluation of some difficulties currently faced when performing each of these methods is carried out. Some appropriate solutions are proposed to tackle these difficulties. In section 3 we use a simpler model of the bipolar junction transistor for modelling. The state equations of the circuit designed corresponding to the shunt structure of the Colpitts oscillator are obtained. The numerical simulation is carried out and various bifurcation diagrams associated to their corresponding graphs of largest one dimensional (1D) Lyapunov exponent are obtained showing both complex and striking scenarios to chaos. Section 4 exploits the Psipice software to simulate the dynamics of the shunt structure of the Colpitts oscillator using the "trial and error" approach. The chaotic behaviour of the oscillator is observed. In section 5, experimental measurements on a real circuit are performed to confirm the results from both numerical and Psipice simulation tools. Section 6 deals with conclusions and proposals for further works.

II. EFFICIENCY OF THE THEORETICAL METHODS VERSUS THE ANALOG METHODS: EVALUATION AND SOME PROPOSALS

A. Theoretical methods Vs. Analog methods

We briefly describe and compare both analog and numerical methods. Invariably the question arises - Which is better, analog or numerical method? Our wish is to present both limits and advantages of each of the methods in order to maintain opened the answer to the above question. We present
and prescribe some practical advises when dealing with analog implementation techniques. Our aim is to encourage engineers to use these techniques for the analysis of nonlinear models despite some practical difficulties faced (when performing these techniques) such as saturation and offset phenomena of the discrete components (diodes, transistors, operational amplifiers, and multipliers) of the electronic circuits. In addition to these difficulties is the dependence of the accuracy of the analog techniques upon the precision and stability of the electronic components. Hopefully, by proposing some techniques to tackle the difficulties encountered during analog implementations this will encourage interested researchers to use analog systems simulation techniques for the analysis of nonlinear problems.

Theoretical approaches (analytical and numerical methods) are commonly used to investigate the dynamics of nonlinear systems. However, the problems faced when performing numerical simulation methods are well-known: a) lack of method to choose the appropriate numerical integration step size, b) lack of method to determine the duration of the transient phase of a numerical simulation, and c) numerical simulation of complex dynamical systems is very time consuming while compared to its analog counterparts that are very fast [24]. Though the analog implementation is always limited by the saturation and offset phenomena of analogue devices such as operational amplifiers (LM741 and LF351) and multipliers (AD-6333JN), it does however offers good ways to tackle the above difficulties faced by the numerical analysis. Analytical methods can provide only approximate solutions of nonlinear dynamical models while analog methods give exact solutions. These are some major reasons for the increasing interest devoted to this type of simulation for the analysis of nonlinear and chaotic physical systems [24 – 28]. In fact, a properly designed circuit can provide sufficiently good real-time results faster than a numerical simulation on a fast computer [24]. Such a circuit must use high precision resistors and capacitors. In addition, the offset voltage of the operational amplifiers and multipliers must be well controlled.

The analog techniques do not take into account the notion of “algorithm” and there is no need to translate quantities into appropriate symbolic forms. For these techniques, variables are represented by physical quantities on which the operations are performed. The simulation is carried out by some physical systems that obey the same mathematical relations that control the physical or technical phenomenon under investigation [29]. This procedure is in some sense more natural to both physicists and engineers [30]. A virtue of the analog techniques is that their basic design concepts are usually easy to recognize. What goes on inside is understandable since it is an analog of the real system whereas the numerical type simulator is a product of pure logic. It cannot be described as similar to something with which we are familiar [31, 32].

Therefore, although the numerical analysis had long ago superseded the analog techniques due, to a large extent, to the spectacular development of digital technology, we still believe that analog techniques might bring some fresh air to the theory of computation in the field of nonlinear dynamics.

B. Practical problems and advices

Concerning the effects of nonlinearity in an unspecified sinusoidal oscillator, we mention that they come mainly from the discrete components (bipolar junction transistor (BJT), operational amplifier (Opamp), or analog circuits multipliers) constituting the gain elements. Thus, the effects of nonlinearity will be differently perceived depending upon the type of gain element. Some recent scientific contributions [1-4] have presented analytical approaches to explain the nonlinear behavior exhibited by classical oscillators using bipolar junction transistors. The exponential dependence of the current flowing through the collector of the BJT with respect to the voltage drop between its base-emitter regions was presented as the origin of nonlinearity in the oscillators. Bifurcation structures were presented, showing the coexistence between the regular states (limit cycles) and the irregular states (chaotic) of the oscillators.

The difficulties faced by theoretical (analytical and numerical) analyses methods are avoided when performing analog implementation techniques. Nevertheless various practical problems are currently encountered during the implementation of analog circuits. Among these problems are some that automatically induce errors in analog calculations. The development below lists some practical problems encountered and proposes solutions to tackle each of them.

- Offset phenomenon:

This phenomenon is the presence of a static voltage at inputs of analog devices (such as operational amplifiers (UA741), circuits multipliers (AD6333JN, ....) when they are biased by a DC voltage source. The offset phenomenon that occurs at outputs of the analog devices usually[39]. We have demonstrated the cancellation technique of this phenomenon in Ref. [24]. Indeed, such phenomenon can be compensated by using a compensation array that consists of monitoring a precise potentiometer to reduce the effect of the phenomenon on the dynamical behavior of the analog devices [24, 39]. The steps to perform offset cancellation are threefold. We use a potentiometer (P) having three points amongst which the middle point is movable between the two others that are fix points. We connect the fix points of (P) to the pin numbers 1 and 5 of the Opamp (for example: UA741 or LF 351). The second step is the connection of the movable point of (P) to a DC voltage source (biasing for instance). By monitoring a potentiometer (P), we measure the evolution of the DC voltage both at inputs and output of the Opamp. The situation where the movable point is very close to the pin numbers 1 and 5 should be avoided. This can lead to the destruction of the device (Opamp) due to a simultaneous presence of the entire value of the bias both at pins 1 and 5. The potentiometer is monitored to transform the magnitude of the input voltages of Opamps into almost the same order. The offset cancellation becomes very complex when the electronic circuit is of a self-
sustained type. In this case, the voltages at inputs of the analog device can be a direct consequence of the self-sustained character of the circuit. When the value of the self-sustained voltage at inputs of analog devices can be predicted (by a circuit theory analysis or by taking into account the expected performances of the circuit that may be defined before the realization of the circuit) the offset cancellation method can be used to fix the predicted values.

- Saturation phenomenon:

The dynamics of analog circuits is limited by the value(s) of the DC voltage(s) source(s) used for biasing. The saturation phenomenon occurs when a signal of magnitude greater than the value(s) of the DC voltage(s) source(s) used for biasing is found at a given point in the electronic circuit. To overcome the saturation problem, the scaling factor process is applied by rescaling the state voltages at different points of the electronic circuit in order to fit within the biasing range. We use a “Static Check” to verify if the system has been wired correctly. By tracing through the system we can calculate what the output voltage of each component should be. If it is determined that all outputs are of correct magnitude and sign (when measuring them), it can be safely assumed that the system is wired correctly.

- Power transfer:

When the electric current is flowing from one electronic network (transmitter) to another (receiver), the power is transferred in the same direction. A problem may arise where the power is not transferred. This can be explained by the fact that the dynamical resistor at the output of the transmitter is not of the order of that at the input of the receiver. Such a problem can be solved by adapting the total dynamical resistor (impedance) between the two electronic networks. This is generally achieved by adding, in parallel, a dynamical resistor at the output of the first electronic network (transmitter) or at the input of the second electronic network (receiver). Also, the power may not be transferred because the connection between the transmitter and the receiver is open. This problem can be detected by measuring the voltage at each point of the analog circuit.

- Defective electronic devices:

Damage of analog devices is generally caused by their wrong supply (biasing) or by a complete involuntary shunt performed between their inputs. When defective, the temperature within the analog devices may be very high. This may be a usual checking for some analog devices such as Opamps and analog multipliers AD633JN. Modules for a direct test of defective components are available. Concerning Opamps, they can be tested as voltage device followers. Analog circuit multipliers can also be tested by loading their inputs by well-known electrical signals. Nevertheless, a situation may occur where despite the fact that the analog devices (Opamps and analog multipliers) are not defective there is no signal at their outputs or the signals at outputs are not those expected. This is a classical problem related to the bandwidth of the analog devices used that may be controlled when performing analog experimentation.

- Time scaling:

It is well-known that the state accuracy of electronic circuits depends on the accuracy of their electronic components (Opamps, analog multipliers, resistors, capacitors, ...) [24, 39]. Yet, the dynamics of electronic circuits is limited by the frequency bandwidth of the analog devices (Opamps, multipliers, ...). When an analog device operates within a range of frequency not included in its bandwidth, this affects the behavior of the electronic circuit containing this device and, consequently the results obtain are not correct. The time scaling process offers to the analog devices (e.g. Opamps, Circuit multipliers, ...) the possibility to operate under their bandwidth. This process is currently used to restrict the high frequencies into low frequencies and inversely, this depending upon the frequency bandwidth of the analog devices in order to expect their good functioning. The time scaling process is also of high importance while performing analog simulation. It offers the possibility to simulate the behavior of the system at very high frequencies by performing an appropriate time scaling that consists of expressing the real time variable \( t \) versus the analog simulation time variable \( \tau \) (e.g. \( t = 10^{-\alpha} \tau \)), allowing the simulation frequency to be \( 10^{12} \) times less than the real frequency. Here, \( \alpha \) is positive integer depending on the values of the resistors and capacitors used in the analog simulator. One of the advantages of time scaling amongst many others is the possibility it offers to the integrators to manage both high and low frequencies signals. Time scaling also allow the simulation of either high frequency or large broadband phenomena using analog devices (Opamps, analog multipliers, ...) that operate in a restricted frequency bandwidth [39].

### III. IMPORTANT CONTRIBUTION

Some interesting proposals were presented to tackle problems encountered by numerical simulation. Concerning the problem due to the integration discontinuities related to the choice of the numerical integration step size, Thomas Rüthner-Peterson [33] proposed an efficient algorithm using backward time-scale differences for solving stiff differential-algebraic systems. The proposed approach has computational advantages in simplicity and flexibility with respect to variations of the integration order. In fact, this algorithm allows the order within each step to be changed in an optimal way between \( k \) and \( k + 1 \). The implementation of the algorithm is described as part of a nonlinear analysis program, which has proved to be quite efficient for simulations of electronic networks. This program provides parameters in the DC analysis mode to be varied with automatic control of the step size. We have found that though the proposed method is very interesting in solving the numerical convergence problem since it varies automatically the step size to obtain an appropriate converging one, it requires very long integration time. Moreover, the
integration duration may become even much larger because it increases with increasing nonlinearity in the system under investigation.

The community providing usable technical solutions for computer-based design (or CAD) has proposed the possibility of using GEAR algorithm [34] in Spice to overcome the divergence problem due to an inappropriate choice of the integration step size [35]. The proposed method, though quite interesting, is limited by the fact that the simulation using Spice is still a theoretical analysis because the characteristics (or the internal parameters) of the analog components (Diodes, Transistors, Operational Amplifiers, and Multipliers) are chosen to be ideal (that is, are transportable to real components which are generally far from being ideal). In addition to this, Spice is emulation and the calculations it performs are done through algorithmic processes on a computing platform of the Von Neuman type.

Mention that Pspice and Simulink are calculation tools that are currently used for analog analysis rather than a real physical implementation (see the subsection below). These simulation tools are purely theoretical and still rely on some form of numerical computation in the background. Further, the analog components they use are generally considered in the states where their characteristics are ideal. This would have been an advantage since the results obtained are of very good accuracy. Unfortunately the simulation of complex dynamical systems using these simulation tools is very time consuming (due to the still numerical computations in the background). Nevertheless, it is clear and sufficiently convincing that analog systems simulation (either analog simulation of the circuits or the direct implementation of the circuits) is more suitable than its other counterparts for the analysis of complex nonlinear phenomena. It is a very precious tool for reliably detecting some strange phenomena such as chaos, modulation, demodulation and also synchronization, to name a few.

IV. SAMPLE RESULTS TO ILLUSTRATE THE CONCEPTS

We consider the shunt type structure of the Colpitts oscillator. The interest devoted to this oscillator is its possibility to behave chaotically both at low and high frequencies. The stability of the shunt type oscillator is also of high importance since it allows an efficient exploitation of such a structure in instrumentation, measurement and telecommunication. Our aim is to propose a shunt type structure of the Colpitts oscillator of practical interest to enrich the literature concerning nonlinear oscillators. The proposed structure might fulfill the above requirements. We show that the proposed structure can be realized experimentally. We also show that analog systems simulation (that is both analog simulation design and direct implementation with analog electronic components) are very suitable to get full insight of the behavior of the oscillator.

A. Circuit description

Fig. 1 is the design of the shunt structure of the Colpitts oscillator under investigation. The bipolar junction transistor $Q_1$ used in the common-base-configuration, plays the role of nonlinear gain element. The feedback network consists of the inductor $L$ and the capacitors $C_1$ and $C_2$. These capacitors act as voltage divider. $C_3$ is a coupling capacitor which may be of very low impedance within the frequency bandwidth in which the oscillator operates. The biasing is provided by the DC voltage source $V_{CC}$. $I_0$ is an ideal current source. The difference between the series type and the shunt type Colpitts oscillators is that the biasing current doesn’t flow through the feedback network in the latter type.

![Circuit diagram of the shunt type Colpitts oscillator](image)

The fundamental frequency of a shunt type Colpitts oscillator can be estimated as follows:

$$f_0 = \frac{1}{2\pi} \sqrt{\frac{C_1 + C_2}{LC_1C_2}} \quad (1)$$

In the structure of Fig. 1, the nonlinear device is the bipolar junction transistor $Q_1$, which nonlinear character is responsible of the chaotic behavior exhibited by the electronic circuit. The transistor Q2N3904 is chosen for the investigations. The choice of this type of transistor is motivated by the fact that the dynamical input impedance

$$Z_{input} = \frac{h_{11}}{h_{21} + 1} \quad (2)$$

is expressed in terms of the hybrid parameters which values give an appropriate value of the dynamical input impedance that allows a good power transfer. Another interest on the Q2N3904 is its availability in Pspice simulation package. Therefore the results from Pspice can be compared with experimental results. Three steps are considered for the investigation of the dynamical behaviour of the shunt type Colpitts oscillator: (1) modelling of the oscillator and analysis of chaotic behaviour exhibited by the oscillator; (2) Pspice simulation of the oscillator using the 'trial and error'
approach; (3) direct implementation of the oscillator. These steps allow the confirmation or validation of the results obtained concerning the behaviour of the oscillator.

B. Circuit modelling and chaotic behavior

- Circuit modeling

The BJT operates in just two regimes namely either the forward conducting or the non-conducting one. Therefore, for theoretical analysis, a simplified model consisting of one current-controlled source and a single diode with exponential characteristic is convenient. Under these assumptions, the emitter and collector currents are defined as follows:

\[ I_E = I_S \left( \exp \left( \frac{V_{BE}}{V_T} \right) - 1 \right) \]  

\[ I_C = \alpha_F I_E \]  

where \( I_S \) is the saturation current of the base-emitter junction and \( V_T = 26mV \) is the value of the thermal voltage at room temperature. \( \alpha_F \) denotes the common-base short-circuit forward current gain of the BJT.

We emphasize that the idea is to select the simplest possible model which maintains the essential features exhibited by the real circuit. If we denote by \( I_L \) the current flowing through the inductor \( L \), and \( V_i \) \((i = 1, 2, 3)\) the voltages across the capacitors \( C_i \), the Kirchhoff Current Laws (KCL) can be applied to the circuit of Fig. 1 to obtain the following set of differential equations describing the evolution of the voltages \( V_i \) within the electronic circuit:

\[ L \frac{dI_L}{dt} = V_1 + V_2 \]  

\[ RC_1 \frac{dV_1}{dt} = V_{CC} - V_1 - V_2 - V_3 + \]  

\[ - R \alpha_F I_E - RL \]  

\[ RC_2 \frac{dV_2}{dt} = V_{CC} - V_1 - V_2 - V_3 + \]  

\[ - R \left( 1 - \alpha_F \right) I_E - I_L - I_0 \]  

\[ RC_3 \frac{dV_3}{dt} = V_{CC} - V_1 - V_2 - V_3 - R \alpha_F I_E \]

Assuming \( \alpha_F = 1 \) (this is well justified for most transistors with \( 100 \leq \beta_F \leq 200 \) ) and introducing the following dimensionless quantities:

\[ x = \frac{\rho I_L}{V_T} \]  

\[ y = \frac{V_1}{V_T} \]  

\[ z = \frac{V_2}{V_T} \]  

\[ \theta = \frac{V_3}{V_T} \]  

\[ \rho = \frac{L}{C_T} \]  

\[ \varepsilon_1 = \frac{C_2}{C_1} \]  

\[ \varepsilon_2 = \frac{C_2}{C_3} \]  

\[ \gamma = \frac{\rho I_L}{V_T} \]  

\[ \delta = \frac{\rho}{R} \]  

\[ \sigma = \frac{V_{CC}}{V_T} \]  

\[ \zeta = \frac{\rho I_0}{V_T} \]  

Eqs. (4) can be transformed into the following first order differential equations:

\[ \frac{dx}{d\theta} = y + z \]  

\[ \frac{dy}{d\theta} = \varepsilon_1 \left[ - x + \delta \left( \sigma - y - z - \theta \right) - f(z) \right] \]  

\[ \frac{dz}{d\theta} = \delta \left( \sigma - y - z - \theta \right) - x - \zeta \]  

\[ \frac{d\theta}{d\theta} = \varepsilon_2 \left[ \delta \left( \sigma - y - z - \theta \right) - f(z) \right] \]

where \( f(z) \) is the exponential function derived from Eqs. (3) and expressed as follows:

\[ f(z) = \gamma \left( \exp(-z) - 1 \right) \]

The model described by Eqs. (6) is the one we propose for the investigation of the dynamical behaviour of the structure under consideration of the shunt type Colpitts oscillator.

- Chaotic behavior

Eqs. (6) are solved numerically to define routes to chaos. We use the fourth-order Runge-Kutta algorithm [36, 37] for the sets of the parameters used in this work, the time step is always \( \Delta t \leq 0.005 \) and the calculations are performed using real variables and constants in extended mode. The integration time is always \( T \geq 10^6 \). Here, the types of motion are identified using two indicators. The first indicator is the bifurcation diagram, the second being the largest 1D numerical Lyapunov exponent denoted by

\[ \lambda_{\text{max}} = \lim_{\tau \to \infty} \left[ \frac{\ln|d(t)|}{\tau} \right] \]  

where
\[ d(t) = \sqrt{\left(\Delta x\right)^2 + \left(\Delta y\right)^2 + \left(\Delta z\right)^2 + \left(\Delta \theta\right)^2} \]

and computed from the variational equations obtained by perturbing the solutions of Eqs. (6) as follows: \( x \rightarrow x + \Delta x, \ y \rightarrow y + \Delta y, \ z \rightarrow z + \Delta z, \) and \( \theta \rightarrow \theta + \Delta \theta. \) \( d(t) \) is the distance between neighbouring trajectories [38]. Asymptotically, \( d(t) = \exp(\lambda_{\text{max}} t). \) Thus, if \( \lambda_{\text{max}} > 0, \) neighbouring trajectories diverge and the state of the oscillator is chaotic. \( \lambda_{\text{max}} < 0, \) these trajectories converge and the state of the oscillator is non-chaotic. \( \lambda_{\text{max}} = 0 \) for the torus state of the oscillator [38].

Setting the values of the components of Fig. 1 \( (V_{CC} = 12V, V_T = 26mV, C_2 = 100nF, C_3 = 22\mu F, L = 470\mu F, I_5 = 6.734mA, I_6 = 5mA, \) and \( R = 318\Omega) \) we analyze the effects of the capacitor \( C_1 \) (or the parameter \( \epsilon_1 \)) on the behaviour of the oscillator. Therefore, a scanning process is performed to investigate the sensitivity of the oscillator to tiny changes in \( C_1 (\epsilon_1). \) The investigations are carried out in the following windows: \( 5\mu F \leq C_1 \leq 15\mu F, \) and \( 15\mu F \leq C_1 \leq 25\mu F. \)

![Bifurcation diagram of the current $i_L$ flowing through the inductor $L$ in terms of the feedback divider capacitor $C_1$](image)

Figure 2. a) Bifurcation diagram of the current $i_L$ flowing through the inductor $L$ in terms of the feedback divider capacitor $C_1$ for $5\mu F \leq C_1 \leq 15\mu F$.

Considering the effects of the capacitor \( C_1, \) it appears that the structure of the oscillator in Fig. 1 leads to complex dynamical behaviour, such as torus, multi-periodic, quasi-periodic, and chaotic states. We observe various routes to chaos (such as sudden transition, period adding, period-doubling, torus breakdown, or quasi-periodic routes) with several kinds of periodic and multi-periodic windows. Fig. 2 provides some sample results, showing the bifurcation diagram \([C_1(n\mu F), i_L(mA)]\) when \( 5\mu F \leq C_1 \leq 15\mu F. \) Period-doubling routes to chaos are shown both with increasing and decreasing \( C_1. \) Also shown is a period 2 sudden transition route to chaos.

![Bifurcation diagrams for $C_1$ and $i_L$](image)

Figure 3. a) Bifurcation diagram of the current $i_L$ flowing through the inductor $L$ in terms of the feedback divider capacitor $C_1$ for $15\mu F \leq C_1 \leq 25\mu F$.

Various tiny windows of chaotic states of the oscillator are shown alternating with windows of regular motion. The weak chaoticity (degree of chaos) of the oscillator is shown. This is clearly demonstrated by the small values of the largest 1D numerical Lyapunov exponent that were always less than 0.0185 for \( 5\mu F \leq C_1 \leq 15\mu F. \) Fig. 3 shows the bifurcation diagram \([C_1(n\mu F), i_L(mA)]\) for \( 15\mu F \leq C_1 \leq 25\mu F. \) A period-adding scenario to chaos (period 5→ period 7→ chaos) is shown. Also shown is the period 4 sudden transition route to chaos that occurs in a tiny window of \( C_1 \) found between 17.5nF and 18nF. The weak chaoticity of the oscillator is also shown.

We have drawn in Fig. 4 some phase portraits of the current \( i_L \) flowing in the inductor \( L \) for sample values \( C_1. \) This figure confirms the period-doubling scenario to chaos shown by the bifurcation diagrams. The following transition: period 1→ period 2→ period 4→ period 8→ chaos is clearly shown. The attractor of period-8 is not shown because of its high instability due to the tiny window within which it co-exists both with period 4 and chaotic attractors situated respectively at left and at right of the value \( C_1 = 9nF \) as clearly shown in Fig. 2b.
Figure 4. Numerical phase portraits of $i_L$: a) Period-1 or limit cycle, \( C_1 = 4.7\, nF \), b) Period-2 \( C_1 = 8.0\, nF \), c) Period-4 \( C_1 = 8.5\, nF \), and d) Chaos \( C_1 = 18.0\, nF \).

Different routes to chaos observed in the shunt structure of the Colpitts oscillator are commonly observed in nonlinear systems, such as forced systems, coupled autonomous systems, and coupled forced systems [38], to name a few. This serves to justify the richness of the bifurcations in the shunt Colpitts oscillator and also the striking phenomena exhibited by such oscillators.

The model proposed for the shunt colpitts oscillator has been computed numerically to have full insight of the behaviour of the oscillator. The simulation on Pspice is performed to verify the numerical results obtained and also to validate the proposed model for the shunt type Colpitts oscillator.

C. Pspice simulation of the oscillator

We use the same model of the bipolar junction transistor defined in the preceding section namely the Q2N3904. The circuit of Fig. 1 is implemented in Pspice. Here, the "trial and error" approach [12-13] is substantially exploited. The following values of the circuit components are defined in Pspice to obtain some phase portraits showing the evolution of the typical phase-space trajectories of the current $i_L$ flowing through the inductor: $V_{CC} = 12\, V$, $V_T = 26\, mV$, $C_2 = 100\, nF$, $C_3 = 22\, \mu F$, $L = 470\, \mu F$, $\beta_T = 210$, $I_0 = 5\, mA$, and $R = 600\, \Omega$. The numerical phase portraits shown in Fig. 5 are obtained from the Pspice simulation. These phases portraits are qualitatively similar to those obtained numerically. Moreover, the sequence of bifurcation shown numerically (period-1 → period-2 → period-4 → period-8 → chaos) is confirmed by the Pspice simulation. The results from Pspice simulation were generally in very good agreement with those from the numerical analysis despite the divergence observed in the values of the bifurcation points (values $C_1$).

This divergence can be explained by the (real) characteristics of the bipolar transistor used for the Pspice simulation. It is well-known that the characteristics of the transistor used (that is the Q2N3904) are predefined and stored in Pspice simulation package some of them being considered as ideal. Moreover, in order to understand the operation mode of the BJT in the oscillator, further simulations were performed using two models of the BJT: a) the Ebers-Moll model and b) the even simpler transistor model consisting of a simple diode and a current controlled source. The results obtained in both cases were similar to the one previously obtained using Pspice own model for the BJT. Thus the simpler model is adequate for investigating the essential behaviour of the system. This makes it possible later to adopt relatively simple state equations to describe the oscillator. The divergence between both numerical and Pspice simulation results were explained by the non-real characteristics of the BJT used. This justifies the interest devoted to the real physical implementation of the shunt Colpitts oscillator since this method uses real electronic components and consequently the characteristics of the electronic components are real.

D. Real physical implementation of the oscillator

According to the previous results, the shunt type Colpitts oscillator can exhibit complex and striking bifurcation scenarios leading to chaos, when the feedback divider capacitor $C_1$ is monitored. The study here is focussed on both
design and analogue experimentation of the shunt type Colpitts oscillator. The experimental results obtained from a real implementation of the oscillator are compared with the results obtained by both numerical and *Pspice* simulation methods.

**Figure 6.** Experimental setup for measurements on the Shunt Colpitts oscillator.

Figure 6 is the proposed experimental setup for measurements on the shunt type Colpitts oscillator. This circuit is built on a breadboard. Fig. 6 shows the basic scheme of the shunt type Colpitts oscillator shown in Fig.1 with the following values of the circuit components: $R = 600\Omega$, $C_2 = 100nF$, $C_3 = 22\mu F$, $L = 470\mu F$, and $V_{CC} = 12V$. The network consisting of the operational amplifier $U_1$ with related resistors is an implementation of the ideal current generator. If the following condition is fulfilled:

$$
\frac{R_1}{R_2} = \frac{R_3}{R_4 + R_5}
$$

(8a)

The current $I_0$ pulled from the load is given by:

$$
I_0 = \frac{R_2}{R_1}V_i
$$

(8b)

where $V_i$ is the output voltage of the network using the operational amplifier $U_2$ with related resistors which electronic function is an inverting amplifier. Therefore, with the values of the components in Fig. 6, the relationship between the control voltage $V_i$ and the current $I_0$ is:

$$
I_0 = \frac{V_i}{1000}
$$

(8c)

Thus, $I_0$ is supposed to vary between 0 and 12mA as $R_{10}$ varies between 0 and 100 kΩ since the inverting input of $U_2$ is connected to -12V.

![Figure 7. Experimental phase portraits of $i_L$: a) Period-1 or limit cycle, $C_1 = 4.7nF$ ($X: 5mA / div.$ and $Y: 2V / div.$), b) Period-2 $C_1 = 22nF$ ($X: 5mA / div.$ and $Y: 2V / div.$), c) Period-4 $C_1 = 25.3nF$ ($X: 5mA / div.$ and $Y: 2V / div.$), and d) Chaos $C_1 = 47nF$ ($X: 5mA / div.$ and $Y: 2V / div.$)](image)

In order to investigate how the feedback ratio affects the dynamics of the circuit, $C_1$ is chosen as a control parameter. The variation of $C_1$ is performed by connecting in parallel standard capacitor components to obtain the desired value. The value of the biasing current $I_0$ is set to 5mA (as in the above *Pspice* simulations) using the resistor $R_{10}$. A 1Ω resistor is added in series to the inductor $L$ to sensor its current $i_L$. The experimental results are obtained by observing as a function of time the voltage across the inductor and by plotting phase-space trajectories ($i_L$, $v_L$) using the oscilloscope in the XY mode.

As in the case of *Pspice* simulations the dynamical behaviour of the oscillator changes substantially as $C_1$ is monitored. This is clearly demonstrated by the experimental pictures in Fig. 7 showing the real behaviour of the shunt type Colpitts oscillator proposed in this paper. As it appears in Fig. 7 the real circuit shows the same bifurcation scenarios as observed using both numerical and *Pspice* simulation methods. Figure 7 shows an evolution of $i_L$ starting from normal near-sinusoidal oscillations to chaos via a period doubling sequence when $C_1$ is increased. This evolution shows identical bifurcation scenarios (period 1→ period 2→ period 4→ period 8→ chaos) with those form the preceding simulation methods. Note that pictures in Fig. 7 are very close to the numerical phase portraits. This can be considered to validate the proposed model (Eqs. 6) for the investigation of the dynamical behaviour of the shunt type Colpitts oscillator. During our experimental investigations, we have also found
period-adding and sudden transition scenarios to chaos exhibited by the system. These scenarios were also reported using both numerical and Pspice simulation methods. The experimental results were generally very close those obtained from these methods. A very good agreement is obtained while comparing the experimental values of the bifurcation parameter $C_1$ with the values from Pspice simulation.

V. CONCLUSION

This paper was motivated by the wish of encouraging engineers to deal with analogue simulation. Nowadays, the revival of this method is encouraged due to the technological exploitation of analogue systems simulation in various fields namely telecommunication, biocomputing, traffic management, electronics (instrumentation and measurements), to name a few. The advantages and limits of analogue simulation were discussed and compared with those of its numerical counterpart. Some proposals to tackle some problems faced during the experimental realisation were presented. The concepts were illustrated by proposing a structure of the shunt type Colpitts oscillator. The choice of this type of oscillator was motivated by our wish to enrich the literature by showing the capability of the proposed oscillator to exhibiting very complex and striking phenomena. Three methods were considered during our investigations: the numerical, the Pspice simulations, and finally the real physical implementation of the proposed oscillator. These methods were compared to validate the results obtained. The KCL theorem was used to derive a model describing the dynamical behaviour of the oscillator. Taking one feedback divider capacitor $C_1$ as control parameter, bifurcation diagrams associated to their corresponding graphs of largest 1D numerical Lyapunov exponent were plotted to summarise the scenario leading to chaos. The studies revealed that the proposed configuration of the Colpitts oscillator can exhibit near-sinusoidal oscillations, quasi-periodic, multi-periodic, and chaotic oscillations. Very complex bifurcation structures were obtained: torus, period-adding, period-doubling, and sudden transition scenarios to chaos. The results from different methods were compared and a very good agreement was observed.

An interesting question under investigation is that of finding the relationship between the loop gain and the dynamics of the oscillator. Another problem under consideration is that of coupling two identical chaotic oscillators of this type and searching for the synchronization threshold. Such an investigation is of high importance in many application areas such as chaotic secure communications where chaos synchronisation is being exploited in wave coding processes. It is also of particular interest to consider the implementation of analog methods exploiting the CNN technology for the analog simulation of very complex systems especially on VLSI chip implementations (for example on FPGA). This is particularly necessary when the number of analog nodes (needed for simulating a given very complex system) is very high (many orders of magnitude).

ACKNOWLEDGMENT

J. C. Chedjou would like to acknowledge the financial support from both the Swedish International Development Cooperation Agency (SIDA) through the International Centre for Theoretical Physics (ICTP), Trieste, Italy. Further, he does express his profound gratitude to the Institute for Smart-systems Technologies (IST), Faculty of engineering, University of Klagenfurt, Austria.

REFERENCES


Jean Chamberlain Chedjou received in 2004 his
degree in Electrical Engineering at the
Leibniz University of Hannover, Germany. He
has been a DAAD (Germany) scholar and
also an AUF Research Fellow (Postdoc.). From 2000 to date he has been a
Junior Associate researcher in the
Condensed Matter section of the ICTP
( Abdus Salam International Centre for
Theoretical Physics) Trieste, Italy.
Currently, he is a senior researcher at the Institute for Smart
Systems Technologies of the Alpen-Adria University of
Klagenfurt in Austria. His research interests include
Electronics Circuits Engineering, Chaos Theory, Analog
Systems Simulation, Cellular Neural Networks, Nonlinear
Dynamics, Synchronization and related Applications in
Engineering. He has authored and co-authored 2 books and
more than 22 journals and conference papers.

Kyandoghere Kyamakya obtained the M.S. in Electrical
Engineering in 1990 at the University of
Kinshasa. He obtained his Doctorate in Electrical Engineering at
the University of Hagen in Germany.
He then worked three years as a post-
doctorate researcher at the Leibniz
University of Hannover in the field of
Mobility Management in Wireless
Networks. From 2002 to 2005 he was
junior professor for Positioning Location
Based Services at Leibniz University of Hannover. Since 2005
he is full Professor for Transportation Informatics and Director
of the Institute for Smart Systems Technologies at the
University of Klagenfurt in Austria.

Van Duc Nguyen received the Bachelor and Master of
Engineering degrees in Electronics and
Communications from the Hanoi
University of Technology, Vietnam, in
1995 and 1997, respectively, and the
Doctorate degree in Communications
Engineering from the University of
Hannover, Germany in 2003.
From 1995 to 1998, he worked for the
Technical University of Hanoi as an Assistant Researcher. In
1996, he participated in the student exchange program between
the Technical University of Hanoi and the Munich University
of Applied Sciences for one term. From 1998 to 2003, he was
with the Institute of Communications Engineering, University
of Hannover, first as a DAAD scholarship holder and then as a
member of the scientific staff. From 2003 to 2004, he was
employed with Agder University College in Grimstad,
Norway, as a Postdoctoral Researcher. He was with
International University of Bremen as a Postdoctoral
Fellow. In 2007, he spent 2 months at the Sungkyungwan
University, Korea, as a Research Professor. His current
research interests include Mobile Radio Communications,
especially MIMO-OFDM systems, and radio resource
management, channel coding for wireless networks.

Moussa Ildoko holds an MSc. in control and signal
processing and a Doctorate degree in
Electronics from the University of
"Valenciennes et du Hainaut-
Cambrésis" in France, respectively in
He is currently an Associate researcher
at UDETIME (Doctorate School of
Electronics, Information Technology,
and Experimental Mechanics) at the
University of Dschang, Cameroon. Besides, he is a Senior
Lecturer at the University of Yaoundé 1, Cameroon. He
research interests are related to Nonlinear Dynamics, Analog
Circuits Design, Chaos based Secure Communications.

Kengne Jacques obtained the diploma of Technical
High School Teacher (DIPET II) from the Department
Electrical Engineering (ENSEP/ University of Douala / Cameroon)
in 1995 and the Master of Science (M.Sc) degree from the faculty of
Sciences/University of Dschang in
2007 both in Electronics. From
1995 up to now he has been working as a Technical high school
teacher. He is currently an associate
researcher at UDETIME (Doctorate
School of Electronics, Information Technology,
and Experimental Mechanics) at University of Dschang. Mr.
Kengne is a doctorate student in Electrical Engineering in the
field of Nonlinear Dynamics and its application in
Communications.