

Towards Precise, Scalable and Automatic Analysis of Analog and Mixed Signal Circuits

Faiq Khalid Lodhi, Nadra Ramzan and Osman Hassan

School of Electrical engineering and Computer Sciences

National University Of Science and Technology

Islamabad Pakistan

{faiq.khalid,nadra.ramzan,osman.hassan}@seecs.nust.edu.pk

Abstract— These days Analog and Mixed signal (AMS) circuits are widely being designed and used in many safety and financial critical application domains. This fact brings the challenge of efficiently and accurately analyzing these AMS circuits as an analysis error could result in a big financial loss. Traditionally, AMS circuits have been analyzed using paper-and-pencil proof methods or computer simulations. But these methods cannot cope with the growing complexities of AMS designs. In this paper, we propose a computer algebra system based analyzer for AMS designs. The distinguishing features of the proposed approach include its scalability, precision and automatic nature. For illustration purposes, we analyze the Impulse generator circuit using the proposed approach and compare our results with the ones obtained by simulation and paper-and-pencil proof methods.

Keywords—Computer Algebra Systems (CAS), MAXIMA; MATLAB;;SIMULINK

I. INTRODUCTION

The integrated circuits having both analog and digital circuits on a single semiconductor die are known as analog mixed signals (AMS) [1] circuits. They are usually needed as an interface between the electronic system and the real world. These days mixed signal circuits are widely used in smart sensors, low data rate RF devices and medical monitoring devices. Some examples of mixed-signal integrated circuits include data converters using delta-sigma modulation, analog-to-digital converter, digital-to-analog converter using error detection and correction, and digital radio chips. Digitally controlled sound chips are also mixed-signal circuits. With the advent of cellular technology and network technology this category now includes cellular telephone, WAN, LAN, software radio and router integrated circuits. With the system on chip designs getting more popular the importance of analyzing AMS circuits has increased.

AMS integrated circuits are usually designed for specific purposes and their design requires a high level of expertise and careful use of computer aided design (CAD) tools. The verification of AMS circuits is also very challenging, where we have to check if the circuit gives correct output for all possible inputs and none of the inputs drives the signal to an unstable state. The working of mixed signals is described in terms of continuous electrical quantities and is sensitive to

signal noise and leakage current, which makes the analysis process even more complex.

Different verification techniques have been used for AMS circuit verification in the past few decades. Some commonly used ones include paper-and-pencil proof methods and simulation. In paper-and-pencil proof method, we first model the circuit that has to be analyzed using a differential equation. The next step is to solve the differential equation in order to find the values of interesting circuit parameters. The main limitation of paper-and-pencil based analysis is the scalability of the approach. With the increase in circuit size, order of the corresponding differential equation also increases, which in turn complicates the process of solving a large high order differential equation. The chances of human error also increase in this approach when dealing with larger circuits. Simulation, which is the second most widely used AMS circuit analysis technique, allows us to observe the behavior of a circuit under a set of given parameters but it is not a very efficient way to estimate parameter values. We can simulate to estimate these parameters using a hit and trial approach by selecting a set of parameters and observing the behavior of the circuit and then repeating the same process for other combinations of interesting parameters. Obviously this would be very tedious for larger circuits where tens of parameters govern the operation of a circuit. Secondly, the results are never guaranteed to be the most optimized ones in this approach. Matlab is very efficient simulation software because of its built in functions but it can only solve linear circuits which have first order differential equations. Solving large circuits with higher order differential equations in MATLAB requires conversion to first order representation, which itself is a very time consuming and complex job with a significant risk of human error.

This paper presents a standalone and automatic AMS circuit analysis approach, which is more scalable and precise than the existing techniques. The proposed approach is based on a computer algebra system (CAS) [2]. A CAS is a type of software program that is used to manipulate mathematical formulae. The primary goal of a CAS is the automation of tedious and sometimes difficult task of algebraic manipulation. The primary difference between a CAS and a

traditional calculator is that CAS deals with equations symbolically rather than numerically. The use and capabilities of CAS vary greatly for one system to another but their purpose is essentially the same i.e. to manipulate symbolic equations. A CAS has the facilities for plotting mathematical equations as graphs and provides a programming language for the users to define their own procedures.

We utilize a combination of simulation and CAS in the proposed approach. The idea is to let the user draw the circuit schematic that he wants to analyze in a GUI. The given circuit is then translated to its corresponding differential equation automatically. The differential equation is then solved using the CAS and a general solution is presented graphically and in a tabulated form. The user can then view these results and pick the most appropriate set of parameters and plug these values in his schematic. This schematic is then simulated to see if the design meets the desired properties. It is important to note that the whole process after the schematic is drawn is automatic and there is no user interaction involved in the differential equation solutions.

In this paper, we demonstrate the effectiveness of the proposed approach by presenting a standalone tool that can analyze any AMS circuit with linear components but the idea can be easily extended to AMS circuits with non-linear components as well. For illustration and comparison purposes, we apply the proposed tool to analyze the impulse generator circuit.

The proposed tool utilizes Maxima [3], which is an open source CAS that is written in LISP [4] and emphasizes on symbolic computation. Its distinguishing features that make it the best choice for AMS circuit analysis include the manipulation of symbolic and numerical expressions, including differentiation, integration, ordinary differential equations (ODE's), systems of linear equations etc. It also has 3D plotting features. It yields high precision numeric results by using exact fractions, arbitrary precision integers, and variable precision floating point numbers. The most important factor is that it gives generic solutions to differential equations, a feature that facilitates the precision and accuracy of the proposed AMS circuit analysis results.

The rest of the paper is organized as follows: Section II presents some related work. Section III provides an overview of the proposed approach for analyzing AMS circuits. In Section IV, we present our experimental results for the impulse generator circuit. Section V includes comparison of the results that are obtained by using different approaches (paper-and-pencil proof method, simulation, CAS) for the impulse generator circuit. Finally, Section VI concludes the paper.

II. RELATED WORK

As discussed above generally three methods are used for analysis and verification of AMS circuits, which are paper-

and-pencil proof method, simulation and CAS. Some of the work that has been done in these directions is described below.

Satoru Iwata and Mizuyo Takamatsu [5] have proposed a combinatorial algorithm that minimizes the index of the hybrid equations in circuit simulation for linear time-invariant electric circuits. The optimal hybrid analysis can be shown to result in differential-algebraic equations (DAEs) with no higher index than modified nodal analysis (MNA) [6]. They have exhibited the effect of the index reduction through some numerical examples. Evaluating the practical effect of using the optimal hybrid analysis for large-scale circuits will be an interesting issue of future investigation.

Donald P Leach [7] concludes that modern Circuit Analysis Programs (CAY) such as PSPICE and MICROCAP offer efficient and accurate alternatives to ignore traditional paper-and-pencil techniques. Nevertheless, the ability to quickly analyze circuit behavior without the aid of a computer is an important skill that provides valuable insights into circuit performance. The paper-and-pencil technique presented here is easy to learn, easy to use and provides rapid estimate of circuit behavior.

In contrast, Marco V. Araujo [8] states that the ODE that results from Chua circuit aren't easy to solve without computational resources. It was shown that MATLABM can be a powerful tool to solve nonlinear circuits. It described the modeling of Chua's diode by an I-V piecewise-linear characteristic to allow the formulation of a system of state-variables equations to model the behavior of Chua oscillator. The chaotic behavior of the circuit was well presented in several simulations becoming more evident in the last ones.

Dejan V. Tošić [9] gave a review of very basic concepts of the state-variable characterization of lumped linear electric circuits. The state-space equations have been formulated using a CAS. A new algorithm for automated computer-aided symbolic computation of the state dynamics equation has been proposed. A Mathematica program SSALEC has been presented as a software implementation of the proposed algorithm. The SSALEC operation has been demonstrated step-by-step by an illustrative example.

Christopher Tocci [10] gave a useful approach for evaluating, solving and designing a common, but not so simple pulse-mode high-gain transit impedance amplifier or TIA circuit. It shows that Maple, which is a commercial CAS, gives the designer a tremendous insight into the gross and subtle aspects of this and other systems where multiple variables (electronic components) have interrelating effects.

As mentioned in Section I as well, paper-and-pencil proof method and simulation share the same limitations of scalability and precision. CAS based approaches overcome these issues, as has been demonstrated by the existing works done in this direction, some of which are outlined above. But

the existing approaches in this domain are either not automatic, requiring information from the user for solving differential equations, or do not offer general solutions, as the solutions are specific to some given initial conditions. On top of that, most of the CAS based approaches have been based on commercial CAS tools like mathematica or Maple that have huge license fees. The proposed approach tends to overcome these above mentioned limitations of the CAS based AMS circuit analysis. Our approach is completely automatic and it provides generalized solutions. We have used MAXIMA, which is an open source CAS with excellent ODE solving capabilities.

III. PROPOSED APPROACH

The proposed AMS circuit analysis approach is illustrated in Figure 1. It mainly comprises of three modules, i.e., a simulator (SPICE), a schematic to differential equation translator and a computer algebra system (MAXIMA).

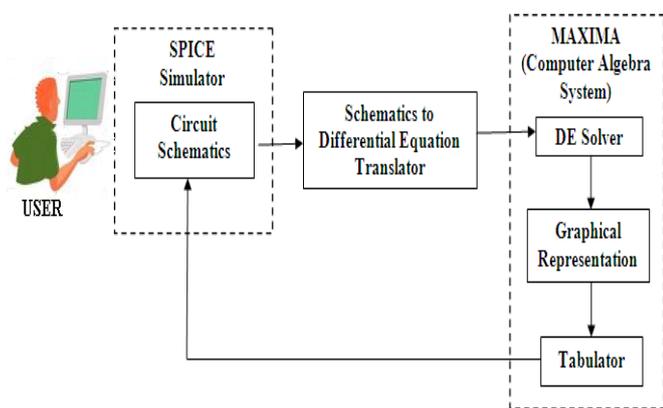


Figure 1. Basic Block Diagram of our Methodology.

The SPICE simulator provides an interactive GUI in which the users can draw and simulate their schematics. The schematics can be graphically plotted by dragging and dropping components on the schematic window. This simulator also provides the solution of circuit in the form of a graph that can be seen on the same GUI. It uses file extension .net in order to store the net list of the circuit drawn on the GUI. The user can also directly give the .net file as a text file to the simulator.

The second module, i.e., the schematic to differential equation translator, translates schematics to their corresponding differential equations and is one of the most important components in the proposed approach that lead to its automatic nature. This module is basically a C++ program that reads the .net file, which is generated by the SPICE simulator, and provides the corresponding differential in a format that is understood by the CAS and saves this information as a text file. Right now, the translator can handle linear components, like resistors, capacitors or inductors only, but the same concept can be extended later on to translate AMS circuits that contain non-linear elements, like transistors as well.

The third module is the computer algebra system MAXIMA. It supports two interfaces for evaluating the solution of a differential equation. First interface is wxMAXIMA which is for the user interaction. It is the GUI where the user can write differential equation in a specific format. Once the differential equation is finalized, MAXIMA solves it and displays the generalized results. The second interface is xMAXIMA, which extracts the differential equation from a text file. In our proposed approach, we use the text based interface because this can allow us to directly read the output of the translator that is based on the .net file for the given AMS circuit. Once a solution is found for the given differential equation, the results are graphically presented as well as tabulated in an organized way to provide a comprehensive account of the circuit behavior based on different parameter values. All of these tasks are performed in three steps. The first step is to solve the differential equation by a DE solver which extracts the differential equation from the text file that is created by the translator and then solves and gives the general solution of the differential equation. The next step is to plot the general solution, which is done by the graphical representation unit in xMAXIMA. The last step is to tabulate our results by using the same solution and the graphs that are obtained from the previous steps. The tabulator tabulates the results using different combinations of the inputs and outputs. This table is passed on to the simulator which prints these results and the user can choose values of components by observing the tables. A distinguishing feature of the proposed methodology is its automatic nature, as the user has to just draw the schematic of the AMS circuit and then initiates the analysis process. He is then given the analysis results in terms of useful graphs and tables. This information can in turn be used to evaluate parameter values for the AMS design under analysis. Once the final parameters are decided, the results of the circuit analysis can be double checked by simulating the circuit in the simulator by using the final values.

We believe that this approach is pretty scalable as it can handle very large circuits without any human interaction and will generate the most precise results since the differential equations are solved by a CAS, which is a well-known technique for solving differential equations.

IV. EXPERIMENTAL RESULTS

In this section, we illustrate the practical effectiveness of the proposed AMS circuit analysis approach by applying it to analyze the Impulse generator circuit. It is indeed a very basic linear AMS design but provides an excellent benchmark to compare all the existing approaches.

An impulse generator essentially consists of a capacitor which is charged to the required voltage and discharged through a circuit. The circuit parameters can be adjusted to give an impulse voltage of the desired shape. Basic circuit of a single stage impulse generator is shown in Figure 2, where the capacitor C_s is charged from a DC source until the spark gap G breaks down.

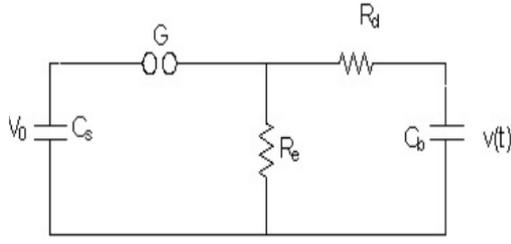


Figure 2. Basic Circuit of a Single Stage Impulse Generator.

We will now analyze the Impulse generator circuit using the two mainstream AMS circuit analysis approaches and the proposed approach. The intent is to highlight the main strengths and weaknesses of each one of these. The analysis results are presented graphically for better understanding.

A. Paper-and-Pencil Based Analysis

The mathematical model of the given impulse generator circuit can be expressed by the following set of differential equations:

$$\frac{dV_0}{dt} = \frac{V_0}{R_e C_s} + \frac{i_0}{C_s}$$

$$\frac{dV_0}{dt} = \frac{i_0}{C_b}$$

$$V_0(t_0 + \delta t) = v(t_0) + \delta t \left(\frac{dV_0}{dt} \right)_{t=t_0}$$

$$V_0(t_0 + \delta t) = v(t_0) + \delta t \left(\frac{dv(t)}{dt} \right)_{t=t_0}$$

Now the goal is to determine the values of R_d , R_e , C_s and C_b such that we get an impulse from this circuit. Figure 3 shows the developed mathematical model of the impulse generator. The components of impulse generator can be designed by conventional numerical methods. But it is complicated and time consuming as mentioned earlier. Moreover, the waveforms may not be accurate for the changes in values of components as the conventional solution incorporates a lot of simplification. So it is better to adopt mathematical modeling and computer simulation techniques.

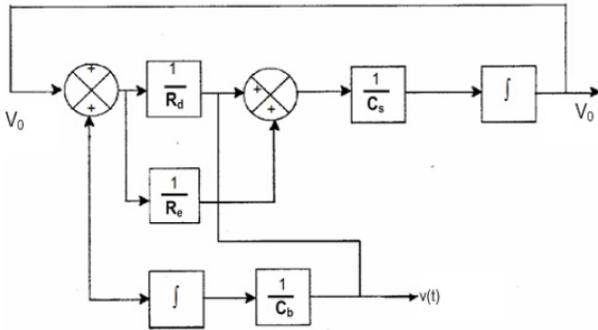


Figure 3. Block diagram of the Mathematical Model of Impulse Generator

In the paper-and-pencil based method, this can be done by obtaining a general solution to the above mentioned set of differential equations. We solved these equations and had to spend around half an hour. This statistic gives an idea about the tedious nature of this approach. Here we are dealing with linear components of a very small sized circuit. As the circuit size grows the complexity of solving the differential equations using paper-and-pencil based approaches rises exponentially. The obtained general solution can be plotted using any plotting software. So we used the plotter of MAXIMA for this purpose and the corresponding graph is given in Figure 4. This graph shows the changing values of impulse voltage with the passage of time. The circuit parameters are evaluated on the basis of user requirements. The user gives his specifications to the MAXIMA and it returns values according to the specifications. In this way different values of resistor and capacitor for impulse generator circuit are obtained.

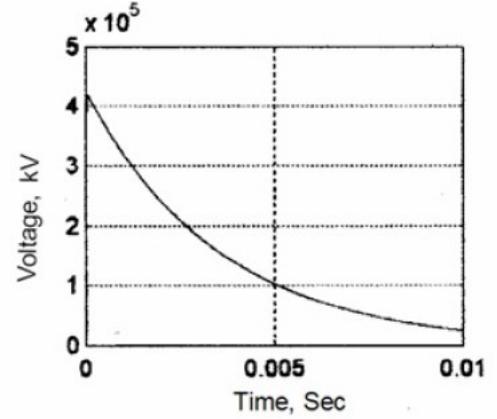


Figure 4. Impulse Generator Circuit Analysis using Paper-and-Pencil Proof Method

B. Simulation Method

Now, we use the simulation method to solve the same problem regarding the given Impulse generator circuit. For this purpose, we use the SIMULINK [11], which is a toolbox in MATLAB. The behavior of impulse voltage generator is again represented by differential equations but in order to simulate this circuit in SIMULINK, we have to somewhat rearrange the equations. The following set of equations suffices for this purpose:

$$V_0(t) = \frac{1}{C_b R_d} \int (V_0 - v_0(t)) dt - V_0(0)$$

$$i_0(t) = \frac{V_c - V_0(t)}{R_f}$$

To determine the state of the gap, a variable A is defined such that,

$$A = 0 \quad \text{When G is not conducting}$$

$$A = 1 \quad \text{When G is conducting}$$

The variable A stands for the state of active switch (spark gap G), and is a binary variable having values either 0 or 1.

Now, the Impulse generator can be modeled in the SIMULINK program with standard blocks available in Simulink as shown in Figure 5. The voltage across the object under test depends on the condition of the active switch G. When the switch is ON ($A = 1$) the voltage across the test object is given by $V_0(t)$ and when the switch is OFF ($A = 0$) the voltage is zero. The next step in the simulation process is to select the integration step size, t for the simulation. The solution becomes more accurate as the step size is reduced to smaller values.

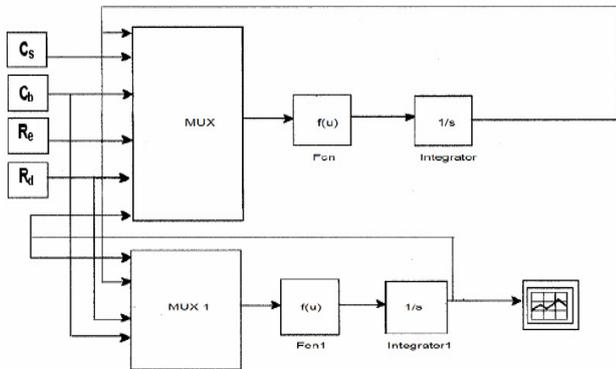


Figure 5. Simulink Model of the Impulse Voltage Generator

Simulating the SIMULINK model under the above mentioned conditions led to the results that are summarized in Figure 6. The graph given below shows the impulse voltage with respect to time using maximum step size.

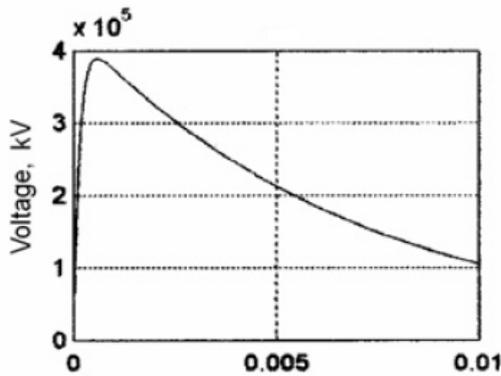


Figure 6. Simulink Result with a larger step size

In general, the step size will have to be taken much smaller than the natural time periods of the system. When we analyzed the circuit using minimum step size the following graph of impulse voltage was obtained.

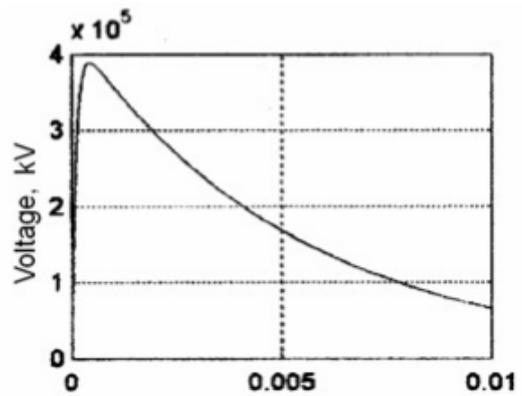


Figure 7. SIMULINK Results with minimum step size

C. Proposed Method using MAXIMA

We will now use the proposed approach to analyze the same Impulse generator circuit. The first step in this regard is to obtain the .net file of our circuit, which can be done by drawing the circuit given in Figure 1 in PSPICE [12]. The net list that we obtain by following this approach, for the circuit of Figure 1, is given in Figure 8.

```
* Schematics Netlist *
X_U1      $N_0001 $N_0002 Sw_tOpen PARAMS: tOpen=0 ttran=1u Rclosed=0.01
+ Ropen=1Meg
C_C1      $N_0003 $N_0001 1n
R_R2      $N_0003 $N_0002 1k
R_R1      $N_0002 $N_0003 1k
C_C2      $N_0003 $N_0003 1n
```

Figure 8. SPCIE Netlist for the Impulse Generator

Next we invoke the translator module that will convert the .net file of Figure 8 to the corresponding set of differential equations. The translator automatically reads the .net file and determines the components present between the nodes and on the basis of this data it constructs the differential equation of the given circuit utilizing the basic Electronics Circuit Analysis techniques like KCL and KVL etc and basic differential equations of the linear components e.g. for capacitor $i_c(t) = C \frac{dv_c(t)}{dt}$ and for inductor $v_l(t) = L \frac{di_l(t)}{dt}$. It then simplifies the equations to the lowest order possible. The software then converts the differential equation in a format that xMAXIMA can understand. The output of the translator is as follows:

$$\begin{aligned} \text{diff}(V_o,t) &= (V_o / (R_e * C_s)) \\ \text{diff}(I_o,t) &= (I_o / C_b) \end{aligned}$$

It is interesting to note that the above equations are the same as the manually derived ones, given in Section IV-A. Since, these equations are in the format of the xMAXIMA therefore it can read from the text file that is generated by the software interface. After obtaining these equations, MAXIMA solves

them and finds the corresponding general solution, which is then plotted as shown in Figure 8.

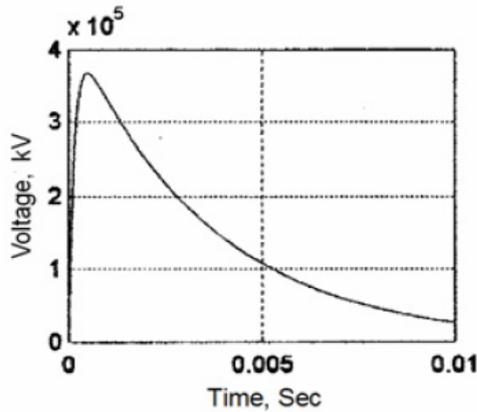


Figure 9. Analysis Results for the Impulse Generator using the Proposed Approach

V. COMPARISON

From the graphical results that we obtained using the three different approaches, we conclude that the graphical solution obtained by SIMULINK had large deviations from the graphical results that were obtained by manual Paper-and-Pencil based techniques. The deviation from the actual results is due to large step size. It has some limitations on the step size because decreasing step size increases the solution time. That is why the curve of the voltage pulse is not following the pattern of the actual curve. From Figure 6, we see that after 0.005sec the voltage values is 220kV but the actual value is supposed to be 100kV similarly after 0.01 the value should be 33kV but it is actually 100kV. Difference between the values is extremely large and in real circuits we cannot afford such deviations.

As we mentioned above that if step size decreases then difference from the actual value reduces. So by reducing the step size up to the lowest limitation we get the graph that is given in Figure 7. From this figure, we can observe that the curve is following the trajectory that is closer to actual results. From the results of this figure, we observe that after 0.005sec the voltage value is 170kV rather than 100kV and after 0.01 sec its about 70kV rather than 33kV. The results in this case are comparatively closer to the actual value but still there are some discrepancies.

Now, the results using the proposed approach are given in Figure 9. We note that the trajectory of the MAXIMA graph is quite close to the actual trajectory. After 0.005 sec, the value of the Impulse voltage is 100.5kV that is quite close to 100kV. The difference being of only 0.5kV, which is very less than the difference that we obtained by using SIMULINK. Similarly, we can observe that after 0.01 sec value of impulse voltage is 33.2kV. Here the difference is 0.2kV which is again very less than the differences obtained using SIMULINK.

VI. CONCLUSIONS

The paper presents a CAS based automatic approach for the analysis of AMS circuits. Due to the excellent precision and scalability features of CASs, the analysis results of the proposed approach are more accurate than the existing alternatives. Similarly, we can handle larger circuits with much ease using the proposed approach. Another benefit of the proposed approach is its automatic nature, which would be very useful for industrial designers, who always prefer to have push button type analysis tools.

The paper also presents the implementation of a standalone tool based on the proposed approach. We utilized MAXIMA, which is open source software, as our CAS tool. The tool is capable of handling any AMS circuit that can be described using linear components. We are currently working on extending these capabilities such that circuits with non-linear elements could also be handled.

We demonstrated the effectiveness of our approach by analyzing the Impulse generator circuit. The results clearly indicate the usefulness of the proposed approach. It is very encouraging to note significant improvements for such a small circuit. This fact clearly indicates that the improvements would be much more enhanced as the circuit sizes increase. One of our future works is to analyze larger AMS circuits and present more case studies in support of the proposed methodology.

VII. REFERENCES

- [1] Mark Burns and Gordon W. Roberts "An Introduction to Mixed-Signal IC Test and Measurement" 2001
- [2] Richard J. Fateman. "Essays in algebraic simplification". Technical report MIT-LCS-TR-095, 1972. (Of historical interest in showing the direction of research in computer algebra. At the MIT LCS web site:
- [3] maxima.sourceforge.net
- [4] Guy L Steele "common LISP:the language" 2nd edition 1990
- [5] Dejan V. Tošić "Formulation of Dynamics Equation for Electric Circuits with Computer Algebra Systems" 16th Telecommunications forum TELFOR 2008 Serbia, Belgrade, November 25-27, 2008.
- [6] W. H. A. Schilders, Philippe G. Ciarlet, Jacques-Louis Lions, E. J. W. ter Maten "Handbook of Numerical Analysis: Special volume: Numerical methods in Electromagnetics" pp 537, 2005
- [7] Dr. Christopher Tocci "Symbolic Circuit Analysis and Design Using Maple"
- [8] Marco V. Araujo "Circuit Analysis and MATLABM Simulation of Chua Oscillator" Telecommunications institute, University of Aveiro, Aveiro PORTUGAL Received 12 January 2009; Accepted 10 February 2009
- [9] Satoru Iwata and Mizuyo Takamatsu "Index minimization of differential-algebraic equations in hybrid analysis for circuit simulation" by Received: 28 April 2007 / Accepted: 11 April 2008 / Published online: 4 June 2008
- [10] Donald P Leach "Active circuit analysis using pen and paper pencil method" January 18, 2005
- [11] Sansen, Willy M.C.; Huijsing, Johan H.; and van de Plassche, Rudy J. (Eds.) "Analog Circuit Design Mixed A/D Circuit Design, Sensor Interface Circuits and Communication Circuits" January 31, 1994
- [12] Bashir Al-Hashimi "The art of simulation using PSpice: analog and digital" 1st edition April 1995.