XVII IMEKO World Congress
Metrology in the 3rd Millennium
June 22–27, 2003, Dubrovnik, Croatia

ELECTRONIC CIRCUITS PARAMETRIC MEASUREMENT: A NEW APPROACH IN ELECTRICAL METROLOGY

*José Carlos Oliveira**
*Diamantino Silva Freitas***
*Artur Capelo Cardoso***

*Instituto Superior de Engenharia do Porto, Porto, Portugal
**Faculdade de Engenharia da Universidade do Porto, Porto, Portugal

Abstract – The measurement of electrical quantities in electronic circuits is inscribed in the non-parametric world for a long time. Multimeters, Oscilloscope and Network Analysers are the preferred tools used by engineers. Parametric identification allows a new step in circuit characterization. This approach opens new perspectives in the process of electronic circuit's design. The present paper presents a solution to the problem of parametric characterization. To allow the simulation and validation of the results a new simulation tool is also presented. The proposed simulation solution uses modified off-the-shelf standard software packages, integrates them closely, offering an expeditious way of bringing mathematical post-processing power to a standard electronic simulation package, in the framework of circuit design and characterization or measurement.

Keywords: Electronic Circuit characterization, simulation, parametric measurement.

1. INTRODUCTION

The measurement of an electronic black box circuit and quality assessment is one of most difficult tasks in a post-design environment or in a validation phase. Usually this task is performed measuring electrical quantities with representation of its time or frequency response or both.

The measurement of other electrical characteristics like input and/or output impedances can reveal a lot more about the circuit but are often neglected. The characterization of the electrical black box circuits usually benefits from measurement of these quantities. This approach reveals the four-terminal model as a fundamental tool to completely describe the circuit. Moreover, this theory can be used to describe the circuit using parameters and opening new perspectives over non parametric approaches.

The simulation of electronic circuits has been for some years a standard practice in the industry, as mean of reducing costs with experiments. The applications used for such simulation techniques are mainly based on two platforms: SABER an SPICE. The original SPICE and subsequent commercial packages lack several functionalities that were needed for system simulation. In particular, post-processing mathematical tools and controlled power sources with arbitrary waveforms were not readily available. This is needed for parametric measurement.

These requirements are fulfilled by the solution presented here, which combines the simulation power and large models library of SPICE with the mathematical toolbox capabilities of Matlab

2 -CURRENT SOLUTIONS AND PROPOSED APPROACH

The electronic circuit characterization and associated metrological approach must take into account the frequency band in use and the provided functionalities. This a priori knowledge leads to a "Grey Box" problem in contrast to the "Black box" one. This approach leads to the conclusion that the measure and consequent circuit parameter identification shouldn't be universal. Two examples could be: High frequency and reverberant systems. When measuring high frequency circuit, the use of "vector network analyser" is the standard solution; the industry is already offering "off the shelf" systems like the HP8510B. By incorporating other measurement equipments some work performed in this field [1] extends the original capabilities

Some electronic circuits employed in audio systems, also have specific problems [2]. The speaker loading effect imposes some particular problems leading to specific solutions like the one based on the non-parametric Time Delay Spectrometry approach.

The proposed solution primarily aims at the medium frequency measurements of electronics circuit's characteristics. Well established circuit characterization tools like dual channel FFT analysis and network analysis offered by commercial solutions as the Hewlett Packard 3562A give a non parametric circuit characterization with several disadvantages for the project and test. We aim a new paradigm in circuit analysis based on the parametric transfer function measure, analysis and identification: The transfer function definition is employed on a wider sense than the classic one. The definition is extended to reflect the need for measuring more circuit elements. This approach, primarily, characterizes not the circuit elements but its parametric transfer function. This representation and the circuit topology knowledge together with some algebraic manipulation can lead to the components values, aiming at fields like component testing and measurement, widening the initial objective.

The proposed platform: is a PC based data acquisition system integrated with some widely accepted powerful mathematical and simulation software packages also offer a new concept and possibilities in the project phase. The results between simulation and measurement can be readily available and compared since both can use the same excitation signal.

3. PROPOSED SIMULATION AND MEASUREMENT SYSTEM

This new approach leads to a PC data acquisition based measurement system. The simulation of electronic circuits has been for some years a standard practice both in the industry, as mean of reducing costs, and in the teaching of electronics in undergraduate studies. SPICE was originally developed at Berkley University and today is one of the most widely accepted electronic circuits simulation program, forming the core of the several commercial products like PSPICE from Orcad or ISPICE from Intusoft. Recently other approaches to the problem emerged like the Mathwoks Simulink/Matlab toolbox.

The Matlab toolbox lacks the wide base of models that SPICE offers. Manufacturers usually offer micro-models for SPICE, making the results very reliable and consistent with the final product. There is very little offer, if any, of similar models for Matlab. Post-processing mathematical tools and controlled power sources with arbitrary waveforms were not readily available in SPICE.

These requirements are fulfilled by an original approach, developed as part of the work presented here, which combines the simulation power and large model base of SPICE with the mathematical toolbox capabilities of Matlab.

The experimental results were obtained using an Analog Devices 16-bit data acquisition board. The Electro Magnetic Compatibility was carefully controlled in order to minimize extraneous input signals. These results were compared with the simulated ones in order to validate the theory and the simulation.

The functionalities that were added to the SPICE package to allow checking between simulated and experimental results were:

- absolute control on the sampling periods;

- the addition of controlled power sources with arbitrary waveforms.

Since these two functionalities were impossible to achieve with closed commercial packages, the solution was to adopt Berkley SPICE [4], version 3, and modifying it accordingly.

In addition to these two additional functionalities added to SPICE, there was also the implicit requirement for calculation power and ease of use. Here, Matlab emerged as the most obvious solution. This approach allowed the full and tight integration between the three system components: Data acquisition, simulation and processing algorithm. In fact, it's possible to feed the data acquisition D/A and simulation with the same data

To demonstrate that the modified simulator version was still converging as the initial one, several tests were conducted, including passive circuits like our RC network or active circuits like analogue filters. Fig 1 represents one of those circuits. Fig 2 shows compared results between our package and ORCAD PSPICE to a step input.

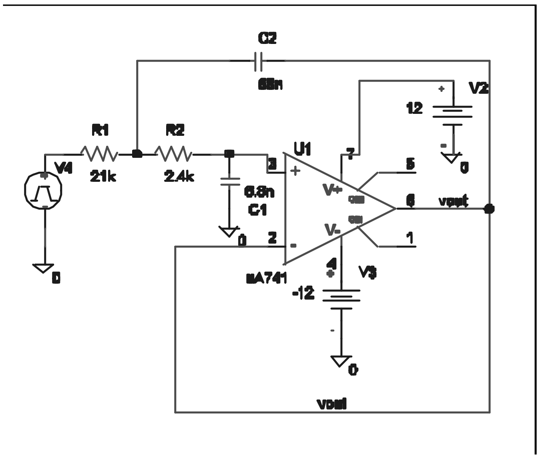

Fig. 1– Circuit used to compare the results between our solution and commercial ones.

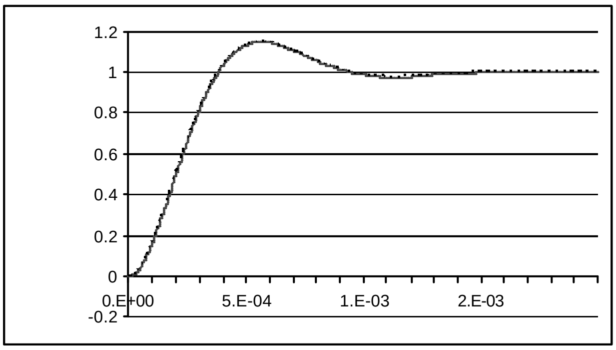

Fig. 2 – Compared results from the circuit in Fig1 to a step excitation.

In order to compare the results several constraints were respected:

- voltages are plotted only for the time points for which coincide in both simulations

- the operational amplifier micro - model was the same, a National Semiconductors 741

The results are very similar and the added simulation points didn't add any convergence problems.

This platform also offers flexibility not available on off-the-shelf instrumentation methods. Expansion of this platform entails only, at most, software evolutions instead of a new instrument.

4. PARAMETRIC ELECTRONIC CIRCUIT MODELLING AND IDENTIFICATION

The four-terminal, or two-port network, theory has been applied for a long time in the modelling of electrical systems. In the particular case of electronic circuits one of the most frequently used models is the so-called hybrid model [3]. If the generic theory is applied, and choosing as input dipole the Norton equivalent form and as output dipole the Thevenin equivalent form, the generic circuit equations are:

$$i_1(t) = H_{11}v_1(t) + H_{12}i_2(t)$$
$$v_2(t) = H_{21}v_1(t) + H_{22}i_2(t)$$

With $v_1(t)$, $v_2(t)$ the input/output circuit voltage and $i_1(t)$, $i_2(t)$ the input/output circuit current. Considering i_2=0 we have:

$$H_{11} = i_1(t)/v_1(t) \quad H_{21} = v_2(t)/v_1(t)$$

and v_1=0

$$H_{12} = i_1(t)/i_2(t) \quad H_{22} = v_2(t)/i_2(t)$$

In the classic approach the so-called hybrid parameters are modelled as electrical components. We took this theory and generalized it, representing each parameter as a transfer function.

If the electromagnetic interference is neglected and the circuit is linear, each transfer function H12, H22, H11, H21 of the four-terminal circuit can be modelled by an ARMA (Auto Regressive moving Average) transfer function [7]; this is one of the key features in our parametric approach.

Because the objective is to demonstrate that the simulation is correct it must be possible to theoretically model the circuit. Let's considerer the chosen circuit represented in fig 3.

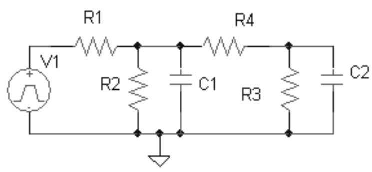

Fig. 3 – Circuit under identification.

In this case: R_1=177KΩ, C_1=2.9ηF, R_4=12 MΩ, R_2=179KΩ, C_2=22ηF, R_3=1.9MΩ and V_2 is the voltage across C_2.

4.1. Mathematical modelling

The circuit was simulated using our modified software and each transfer function of the four-port was identified using a least-squares method [9] with an independent uniformly distributed sequence [10] as input signal. The least-squares method was chosen since other non-standard methods like higher-order analysis [11] proved to be less adequate [12] in this particular case. This four-terminal model requires two simulations, one for each pair of transfer functions (H_{11}, H_{21} *and* H_{12}, H_{22}.) Both simulations used as input the same independent uniformly distributed sequence.

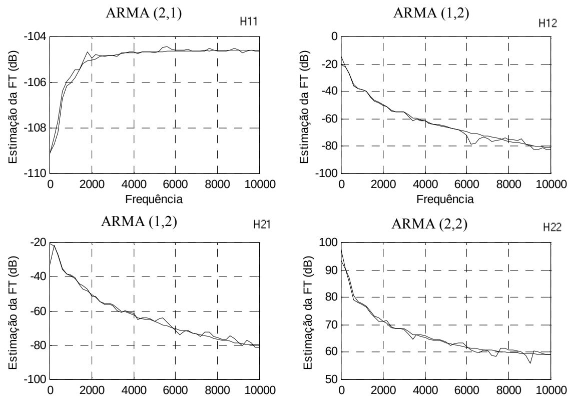

Fig. 4 – Simulated and theoretical compared results.

The compared results are depicted in Fig. 4, for the four transfer functions in the four-terminal model. The order, for each z-domain transfer function, can be found above each chart.

4.2. Experimental confirmation

The presented results were confirmed by a practical test using real signals picked-up from the circuit. The chosen circuit was implemented and tested by means of a 16-bit data acquisition board.

The electromagnetic interference was carefully controlled. However some lack of equipment didn't allow the necessary tests in order to obtain all of the 4 transfer functions. We can now show the results for the case of H_{21}. In Fig 5 the chart compares between the practical and the simulated tests results, showing a quite good agreement. This H_{21} transfer function has two real poles on frequencies P_1=119 Hz and P_2=4 KHz. The results agreed within a ±5% error margin around the dominant pole frequency value.

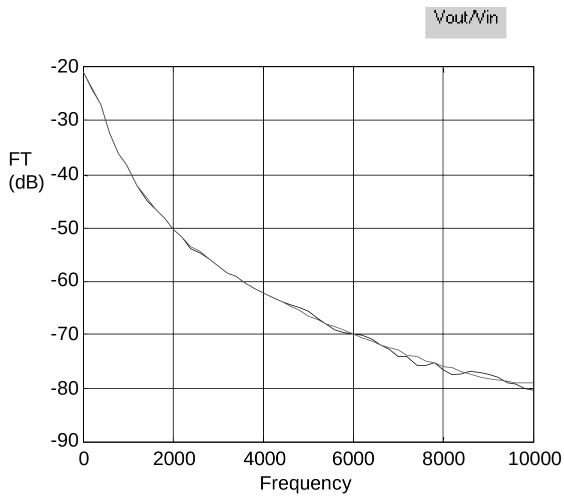

Fig.4 – Compared results for H_{21} between simulation and acquired signals.

4. CONCLUSIONS

A new approach to electronic circuit simulation modelling and measurement was presented. This approach is based on parametric modelling and transfer function identification and uses a four-terminal (two-port) model structure. To support our study a new simulation tool was developed based on standard software packages, integrating the simulation features with extensive mathematical processing.

Theoretical and experimental results were obtained showing a good agreement. The approach is being expanded to new fields such as modelling of non-linear circuits. New approaches are now being evaluated, such as electronic circuits modelled as Multiple Input Multiple Output systems. Inclusion of EMC effects in the model is also under consideration.

The modified SPICE simulation package is available for download at http://www.dee.isep.ipp.pt/~jcarlos/matlab.

REFERENCES

[1] L. Dunleavy, “Mixer Measurements using Network and Spectrum Analysis”, 48thARFTG Conference *Dec. 5-6, 1996 Clearwater FL*

[2] Cabot Rich, “Fundamentals of Modern Audio Measurement”, Journal of the Audio Engineering Society, September 1999

[3] Mason Samuel, “Electronic Circuits, signals and systems”, The MIT press, 1970

[4] T. Quarles, “SPICE3 Version 3f3 User’s Manual”, Berkley, CA, 1993

[5] DJ Delorie, “Cygwin User's Guide”, http://sources.redhat.com/cygwin/cygwin-ug-net/cygwin-ug-net.html

[6] Donald L. Schilling, “Electronic Circuits, discrete and integrated”, MCGraw-Hill, 1989

[7] Lennart Ljung, “System identification, Theory for the user”, Upper Sadlle River, NJ: Prentice-Hall, 1999

[8] John Tsimbinos, “Identification and compensation on nonlinear distortions”, PHD Thesis, University of South Australia; 1995

[9] Graham Goodwin, “Dynamic system identification – Experiment design and data analysis”; Academic Press; 1977

[10] Yannis Viniotis, “Probability and Random Processes, for electrical engineers”, McGraw-Hill, 1998

[11] Mendel J. M. “Tutorial on higher – order statistics (spectra) in signal processing and system theory: Theoretical results and some applications”, Proc. IEEE, Vol. 79, 1991, 278-305

[12] *Boss et al,* “Statistical analysis of cumulant estimation errors with regard to an efficient blind ARMA identification “, Proc. ATHOS Workshop on HOSA in signal processing Edinburgh, 1994, 9-14

Authors:

José Carlos Barros Oliveira, Instituto Superior de Engenharia do Porto, Rua António Bernardino de Almeida, 4200 Porto, Portugal, +351228340500, jcarlos@dee.isep.ipp.pt.

Diamantino Rui da Silva Freitas, Faculdade de Engenharia da Universidade do Porto, Rua Roberto Frias, 4200 Porto, Portugal, +351225081400, dfreitas@fe.up.pt.

Artur Capelo Cardoso, Faculdade de Engenharia da Universidade do Porto, Rua Roberto Frias, 4200 Porto, Portugal, +351225081400, acardoso@fe.up.pt.