Simulation Bits©: Some Less Familiar Features of PSpice

Sam Ben-Yaakov and Mor Mordechai Peretz
Power Electronics Laboratory
Department of Electrical and Computer Engineering
Ben-Gurion University of the Negev
P.0.Box 653, Beer-Sheva 84105, ISRAEL
Tel: +972-8-6461561; Fax: +972-8-6472949
Email: sby@ee.bgu.ac.il ; Website: http://www.ee.bgu.ac.il/~pel

I. Introduction

SPICE simulation is already recognized as a viable tool in Power Electronics. It appears however, that some powerful features of modern simulators are not well known and as a result are seldom used to tackle Power Electronics Problems. In this article we demonstrate the use of the GFREQ (or EFREQ) behavioral dependent source [1] and the Optimization tool of PSpice (Cadence, USA) [2] to solve a modeling problem of a piezoelectric element. The main objective of the article is to alert workers in the Power Electronics field to the capabilities of these features that could be useful, of course, in many other cases.

II. The E/GFREQ behavioral dependent sources

These sources generate an output (voltage for EFREQ and current for GFREQ) that is a function of a PSpice 'expression' namely, a mathematical function of voltages and currents in the simulated circuit. The expression is then multiplied by a table to generate the output. Here the table is defined in the frequency domain as a discrete transfer function. The output voltage V(EFREQ) of the EFREQ will thus be:

$$V(EFREQ)=\{f[V(n),I(m)]\}[H(f_k)] \qquad (1)$$

where V(n) are voltages of various n nodes in the circuit, I(m) are currents through some m devices, and $H(f_k)$ is a discrete k points table given as:

$$H(f_k)=[f_1,G_1,P_1)(f_2,G_2,P_2)...(f_k,G_k,P_k)] \qquad (2)$$

in which 'f' is frequency, 'G' is gain and 'P' is phase

III. The Optimization Tool

PSpice (Cadence) optimization tool is an add-on package to the basic simulator that allows the selection the values of components to meet a specific goal function. A common example to such case will be the selection of the value of a capacitor in a feedback loop to meet a given phase margin criteria [2]. The initial data that are fed to the optimizer include an expression of the goal function, additional constraints, if any, and initial values of the components to be optimized. A detailed description of this tool is clearly beyond the scope (and space) of this article. Nonetheless, some features of the optimizer will become apparent when discussing the specifics of the example given here.

IV. The Example

We demonstrate the application of the above-mentioned features of PSpice by considering the problem of extracting the equivalent circuit of a piezoelectric element. The element under consideration is a Piezoelectric Resonant Blade (PRB) [3, 4] and the objective of the equivalent circuit is to emulate the impedance of the element as measured at its input terminals around the resonant frequency of the device. It is assumed that the PRB can be represented by the classical equivalent circuit of a resonator (Fig. 1).

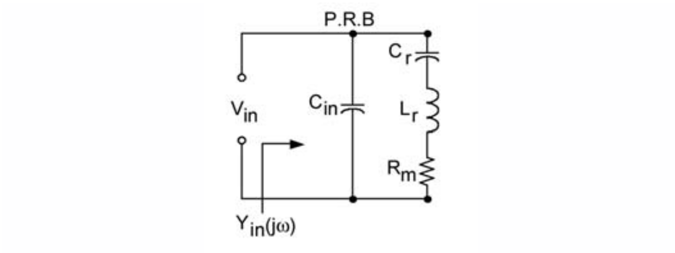


Fig. 1. Equivalent circuit of the Piezoelectric Resonator Blade (PRB)

The proposed model extraction procedure.

1. Measurement. The first stage of the proposed procedure is the admittance measurement. This was done in the present study by a network analyzer (HP4395A) coupled to a power amplifier so that the measurements could be carried out under high voltage excitations [3]. The outcome of the measurement is the admittance of the device as function of frequency. The results were saved as a table file that includes the information of the magnitude and phase for each frequency point over the measurement range (150Hz to 250Hz) around the resonance frequency of the device (180Hz).

2. Setting the GFREQ model. The table obtained from the admittance measurement was inserted into a PSpice GFREQ behavioral dependent source to create a subcircuit that emulates the admittance of the PRB (Fig. 2).

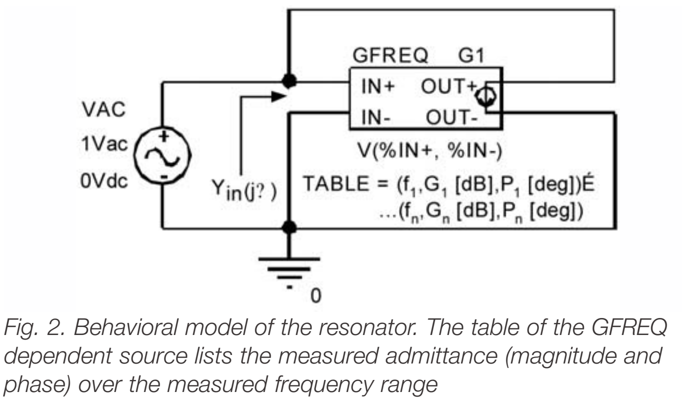


Fig. 2. Behavioral model of the resonator. The table of the GFREQ dependent source lists the measured admittance (magnitude and phase) over the measured frequency range

Notice that in this case, the 'expression' (V(%IN+,%IN-)) is the input voltage to the element and that the output is connected in parallel to the input. This way, the input current is a function of the input voltage times the admittance described by the table. Consequently, when the PSpice compatible equivalent circuit of Fig. 2 will be run on the simulator, it will duplicate the PRB input terminals admittance.

3. Setting up the optimizer. The objective of the optimization procedure was to find the values of C_{in}, R_m, L_r, C_r of the equivalent circuit (Fig. 1) such that it will faithfully represent the physical PRB. This was accomplished by running a set of AC simulations of a system that included the equivalent circuit (Fig. 1) and the GFREQ source (Fig. 2), each fed by a unity AC voltage source, and letting the optimizer choose the values such that the difference between the responses was minimized according to the least square error function. The error was therefore defined as:

$$err = min\ \{[Re(I_{ref}) - Re(I_{mod})]^2 + [Im(I_{ref}) - Im(I_{mod})]^2\} \quad (3)$$

where 'Re' and 'Im' designate real part and imaginary part operators, subscript I_{ref} is the current to the GFREQ model and I_{mod} is the current to the equivalent circuit.

While running, the optimizer calculates 'err' for each AC run and the C_{in}, R_m, L_r, C_r are iterated to minimize the 'err' cost function.

4. Estimating the initial values. To help the optimizer to converge to final values that make sense from the physical point of view, the initial values of the components should be close to the correct ones. The first guesses of the components' values were based, in this work, on an approximated calculation that also applied the results of the network analyzer measurements.

The admittance of the PRB model of Fig. 1 can be expressed as:

$$Y_{in}(j\omega C_{in})\, j\omega C_{in} + \frac{1}{R_m + j\omega L_r + 1/j\omega C_r} \quad (4)$$

where C_{in}, R_m, C_r, L_r refer to the values at around the resonance frequency of interest.

Around the fundamental series resonant frequency (L_r, C_r), ω_r, the admittance can be simplified as:

$$Y_{in}(j\omega) = \frac{1}{R_m} + j\omega C_{in} \quad (5)$$

Eq. (5) can therefore be used to obtain the approximated values of R_{m_int} and C_{in_int} by:

$$C_{in\_int} = \frac{Im(Y_{in}(j\omega_r))}{\omega_r} = \frac{|Y_{in}(j\omega_r)| sin(\varphi_r)}{\omega_r} \quad (6)$$

$$R_{m\_int} = \frac{1}{Re(Y_{in}(j\omega_r))} = \frac{1}{|Y_{in}(j\omega_r)| cos(\varphi_r)} \quad (7)$$

where φ_r is the phase when $Y(j\omega)$ is at the maximum value that is, around the series resonant frequency ω_r.

The resonant frequency is given by:

$$\omega_r = \frac{1}{\sqrt{L_r C_r}} \quad (8)$$

and the quality factor, Q, can be obtained from the measurements by:

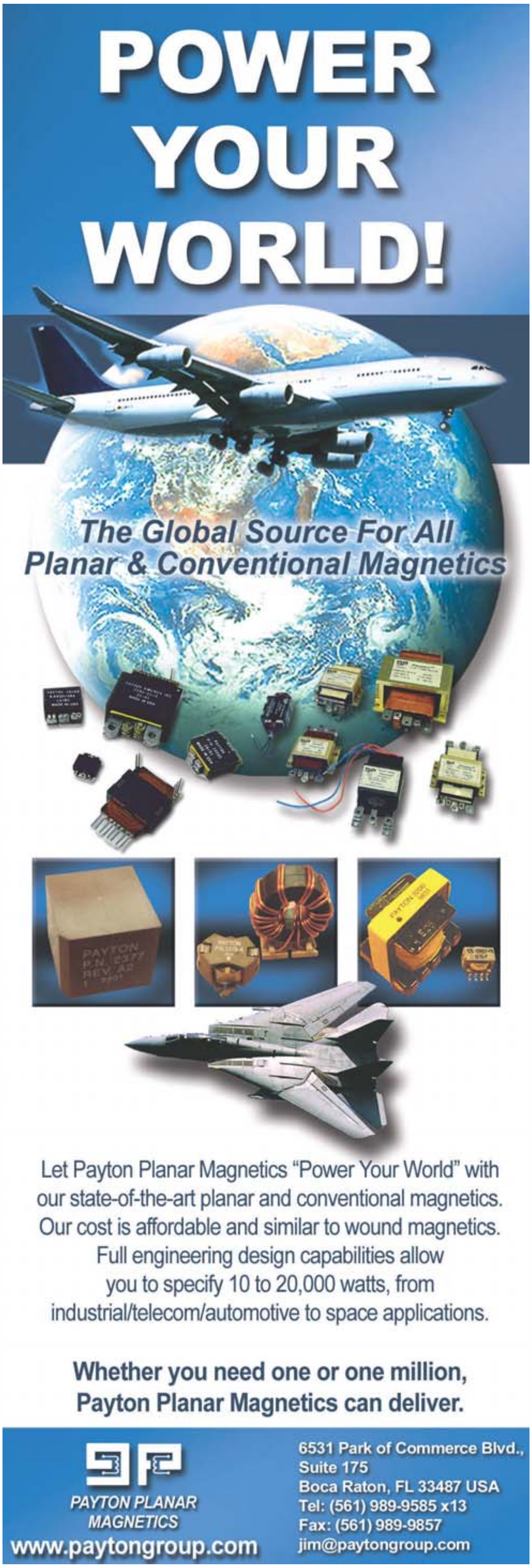
$$Q = \frac{f_r}{\Delta f} \quad (9)$$

where Δf is the 3-dB bandwidth of admittance around the resonance point.

The quality factor for the series resonance network is given by:

$$Q = \frac{\omega_r L_{r\_int}}{\omega_r} \quad (10)$$

From (10) L_{r_int} can be isolated, yielding the approximated initial value of the resonant inductor:



$$L_{r\_int} = \frac{1}{\omega_r^2 L_{r\_int}} \quad (11)$$

Combining (8) and (11), yields the approximated initial resonant capacitor value:

$$C_{r\_int} = \frac{1}{\omega_r^2 L_{r\_int}} \quad (12)$$

Following the above procedure the estimated initial values were: C_{in_int}=110nF, L_{r_int}=115H, C_{r_int}=6nF, R_{m_int}=7KΩ.

V. Results and Discussion

The optimizer estimates for the problem under consideration were: C_{in}=64.36nF, L_r=106.7H, C_r=6.78nF, R_m=6.75KΩ. Fig. 3 shows the original admittance measurement and the admittance of the equivalent circuit. The main reason for the non-perfect fit is the fact the PRB exhibits non-linearity around the series resonance point (about 180Hz) when the current is high and the large deflection is beyond the linear portion of the material's elasticity.

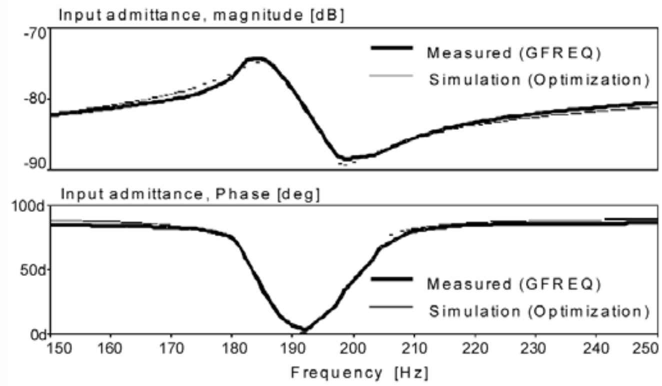


Fig. 3. Comparison between measured and model calculated admittances. Estimated parameters: C_{in}=64.36nF, L_r=106.7H, C_r=6.78nF, R_m=6.75KΩ.

Although demonstrated in a modeling application, E/GFREQ and the Optimization tool can be useful in many other Power Electronics applications such as optimizing the passive power elements components to meet given goals (size, efficiency, ripple, etc.), optimizing the feedback network of switch mode systems, and others.

Pspice simulation files of the PRB example discusses in this article (GFREQ and optimization) can be downloaded at: http://www.ee.bgu.ac.il/~pel/download.htm. The simulation project will run on ORCAD 9.2 or higher versions which include the PSpice optimization add-on package. The optimization routine will not run, however, on an evaluation version (9.2 Lite). The PRB simulation model itself will run on the evaluation version but with the fixed, already optimized, values.

Acknowledgment

This research was supported by THE ISRAEL SCIENCE FOUNDATION (grant No. 113/02) and by the Paul Ivanier Center for Robotics and Production management.

References

1) Cadence Design Systems, Inc., "PSpice A/D user's guide", 2nd edition, chapter 6, 2000.
2) Cadence Design Systems, Inc., "PSpice Optimizer user's guide", 2nd edition, 2000.
3) S. Ben-Yaakov and N. Krihely, "Modeling and driving piezoelectric resonant blade elements", IEEE Applied Power Electronics Conference, APEC-2004, 1733-1739, Anaheim, CA, 2004.
4) Piezo Systems, Inc., "Introduction to piezo transducers", http://www.piezo.com, 2004.

Biographies

Shmuel (Sam) Ben-Yaakov received the B.Sc. degree in Electrical Engineering from the Technion, Haifa, Israel, in 1961 and the MS and PhD degrees in Engineering from the University of California Los Angeles, in 1967 and 1970 respectively.

He is presently a Professor at the Department of Electrical and Computer Engineering, Ben-Gurion University of the Negev, Beer-Sheva, Israel, and heads the Power Electronics Group there. He served as the Chairman of that department during the period 1985 - 1989. His current research interests include power electronics, circuits and systems, electronic instrumentation and engineering education. Dr. Ben-Yaakov also serves as Chief Scientist of Green Power Technologies Ltd, Israel, and as a consultant to commercial companies on various subjects, including analog circuit design and Power Electronics.

Mor Mordechai Peretz was born in Beer-Sheva Israel in 1979. He received the B.Tech. degree in Electrical Engineering in 2003 from the Negev Academic College of Engineering, Beer-Sheva Israel. He is presently pursuing the M.Sc. studies at the Ben-Gurion University of the Negev.

His areas of interests include modeling and simulation, power factor correction and resonant power conversion systems.

10 Steps to Organizing a Successful Conference

F. Dong Tan

As your past vice president for meetings, one of the questions that was frequently asked of me is: how do I organize a meeting or a workshop that is sponsored by PELS? Here is a list of 10 steps that you need to follow to organize a successful conference or workshop.

1. Get a group of people together

To organize a successful conference or a workshop requires the dedication of a group of individual volunteers who share the same technical interests and dedication. It is generally a good