A Unified SPICE Compatible Model for Large and Small Signal Envelope Simulation of Linear Circuits Excited by Modulated Signals

Simon Lineykin and Sam Ben-Yaakov*

Power Electronics Laboratory
Department of Electrical and Computer Engineering
Ben-Gurion University of the Negev
P. O. Box 653,Beer-Sheva 84105, ISRAEL, Phone: +972-8-646-1561,Fax: +972-8-647-2949
Email: sby@ee.bgu.ac.il, Website: www.ee.bgu.ac.il/~pel

Abstract - **The envelope simulation method, developed earlier for large signal simulation (time domain, TRAN), is extended to include small signal envelope simulation (AC) and DC Sweep simulation (steady state for a range of carrier frequencies). The model is derived for AM, FM and PM modulation schemes and is demonstrated on a piezoelectric transformer circuit. The analytical derivations of the model were verified against full circuit simulations that include the high frequency carrier. Excellent agreement was found between the simulation results according to the new unified envelope model and the full simulation.**

I. INTRODUCTION

Various power electronics systems such as resonant converters, electronic ballasts for discharge lamps, piezoelectric transformers and others, are based on resonant networks that are often exposed to modulated signals. For example, the conventional method of setting the light output of a lamp powered by an electronic ballast, is to control the drive frequency of the ballast. When such systems operate in closed loop (Fig. 1) the feedback signal of interest is normally the envelope of the sensed signal. In this case, the error signal is translated into a frequency-modulated signal which, in turn, is affecting the envelop of the signal that is sensed at the output (Fig. 1). Consequently, analysis and simulation of the small signal transfer functions, needed for controller design and stability analysis, is rather complicated. As demonstrated earlier [1 - 4] envelope simulation by a SPICE compatible model could be used to simplify the extraction of the relevant small signal transfer functions and for testing the dynamic stability of frequency driven systems. However, in the previous approach only large signal (time domain) simulations could be carried out. The required small signal response had to be extracted from the envelops of the time domain simulation runs [4]. This, of course, is a tedious process since each frequency domain point requires a lengthy time domain simulation run.

The objective of this study was to develop a unified model that can be used to run both large signal (time domain, TRAN) envelope simulation as well as small signal (frequency domain, AC) simulation by applying the same model. That is, to develop a model that can be used as is, without the need for an analytical linearization effort, for TRAN, AC as well as DC (steady state sweep) analysis. Since the method hinges on the earlier SPICE compatible envelope simulation model, we present the essentials of that model by a way of an example: a piezoelectric transformer driven by a FM signal [5-7]. The details of the envelope simulation model are given in [3] and an example of its application in [4].

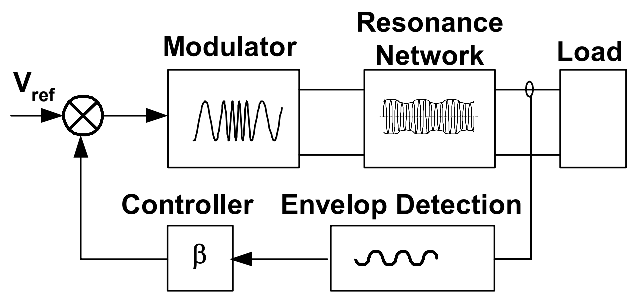

Fig. 1. A resonant converter under close-loop control.

II. LARGE SIGNAL ENVELOPE SIMULATION

Any analog modulated signal (AM, FM, and PM) can be described by the following general expression:

$$u(t) = U_1(t)\cos(\omega_c t) + U_2(t)\sin(\omega_c t) \tag{1}$$

where $U_1(t)$ and $U_2(t)$ are modulation signals and ω_c is the angular frequency of the carrier signal.

Expression (1) can also be written in complex form as:

$$u(t) = \mathrm{Re}[(U_1(t) - jU_2(t))e^{j\omega_c t}] \tag{2}$$

or as

$$u(t) = \left|\vec{U}(t)\right| \mathrm{Re}[e^{\arg(\vec{U}(t))} e^{j\omega_c t}] \tag{3}$$

where:

$$\vec{U}(t) = U_1(t) - jU_2(t) \tag{4}$$

and

$$\arg(\vec{U}(t)) = \tan^{-1}\left(-\frac{U_2(t)}{U_1(t)}\right) \tag{5}$$

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

$$\left|\vec{U}(t)\right| = \sqrt{U_1^2(t) + U_2^2(t)} \quad (6)$$

Expression (3) reveals that any modulated signal can be represented by a generalized phasor with time dependent magnitude and phase.

As demonstrated earlier [3,4], the SPICE compatible envelope simulation circuit can be developed by means of the following stages:

- Duplicating the circuit to create the real part and the imaginary parts.
- Replacing reactive elements (L, C) as shown in Fig. 2 into the real and imaginary sections of the circuit.
- Placing two excitation sources for real and imaginary parts ($U_1(t)$ and $U_2(t)$) but excluding the carrier.
- Adding a behavioral element for calculating the square root of the sum of squares of real and imaginary components of the output signals

The expressions for U_1 and U_2 for various modulation schemes are given below for the case of a single tone modulation with a modulating signal m(t) and a carrier c(t):

$$m(t) = A_m \sin(2\pi f_m t) \quad (7)$$

$$c(t) = A_c \cos(2\pi f_c t) \quad (8)$$

where: A_m, A_c – amplitudes of the modulating signal m(t) and carrier signal c(t) respectively, f_c – frequency of the carrier, f_m-frequency of the modulating signal.

The AM signal is described by:

$$u(t) = A_c(1 + k_a A_m \sin(2\pi f_m t))\cos(2\pi f_c t) \quad (9)$$

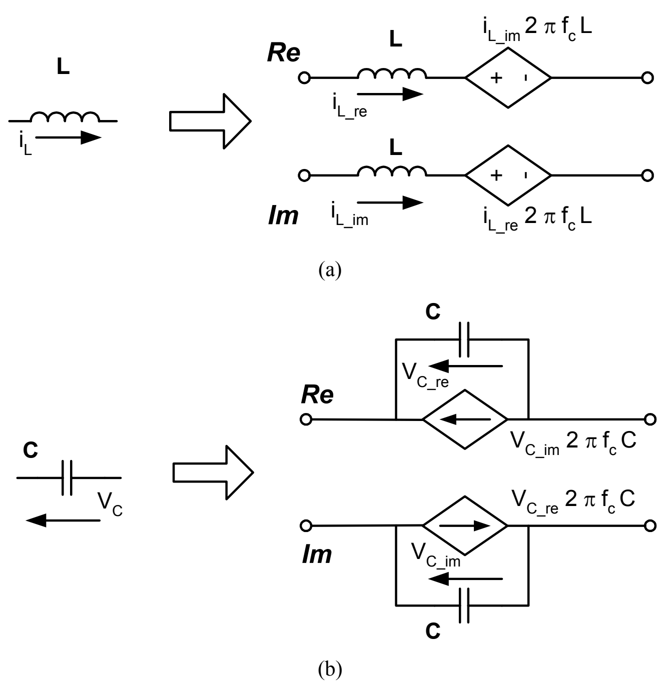

Fig. 2. Replacement of reactive elements by equivalent circuits for envelope simulation: (a)-Replacing an inductor by an inductor and dependent voltage source; (b)-Replacing a capacitor by a capacitor and dependent current source.

An FM signal for any modulating signal $V_m(t)$:

$$u(t) = A_c \cos\left(2\pi f_c t + 2\pi k_f \int V_m(t)dt\right) \quad (10)$$

and in the single tone case:

$$u(t) = A_c \cos\left(2\pi f_c t - \frac{k_f A_m}{f_m}\cos(2\pi f_m t)\right) \quad (11)$$

The PM signal is expressed as:

$$u(t) = A_c \cos(2\pi f_c t + k_p A_m \sin(2\pi f_m t)) \quad (12)$$

where 'k' is the modulation coefficient and the subscript indexes "a", "p" and "f" stand for AM, PM and FM respectively.

The decomposed signal sources (U_1, U_2) for amplitude modulation (AM) are:

$$U_1 = A_c + k_a A_m A_c \sin(2\pi f_m t) \quad (13)$$

$$U_2 = 0 \quad (14)$$

For frequency modulation (FM):

$$U_1 = A_c \cos(\beta \sin(2\pi f_m t)) \quad (15)$$

$$U_2 = A_c \sin(\beta \sin(2\pi f_m t)) \quad (16)$$

where: $\beta = \frac{A_m k_f}{f_m}$

For phase modulation (PM):

$$U_1 = A_c \cos(k_p A_m \sin(2\pi f_m t)) \quad (17)$$

$$U_2 = A_c \sin(k_p A_m \sin(2\pi f_m t)) \quad (18)$$

Based on the above, the piezoelectric transformer circuit of Fig. 3, driven by an FM modulated source can be represented by the SPICE circuit of Fig. 4 (shown for OrCAD V 9.2). The circuit is split into two sections representing the real and imaginary parts and includes two excitation ports 'inre' and 'inim' that are driven by the U_1 and U_2 sources of the FM case. The behavioral source 'abs_out' carries out the calculation of the square root of the sum of squares of real and imaginary components of the output voltage.

Typical simulation results that compare the traditional transient simulation of Fig 3, as is, to the results of envelope simulation by the model of Fig. 4 are depicted in Fig. 5. The two sets of results are identical and not merely 'similar' since the envelope simulation method is based on an exact analytical representation of the circuit.

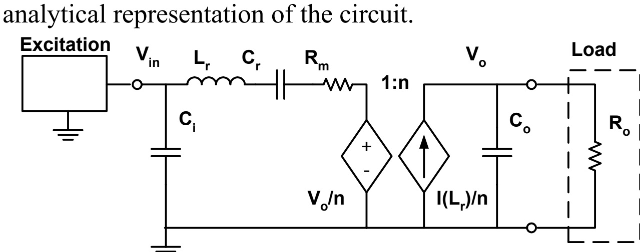

Fig. 3. An equivalent circuit of a piezoelectric transformer that is loaded by a resistor R_o and driven by a modulated signal (AM, FM, or PM).

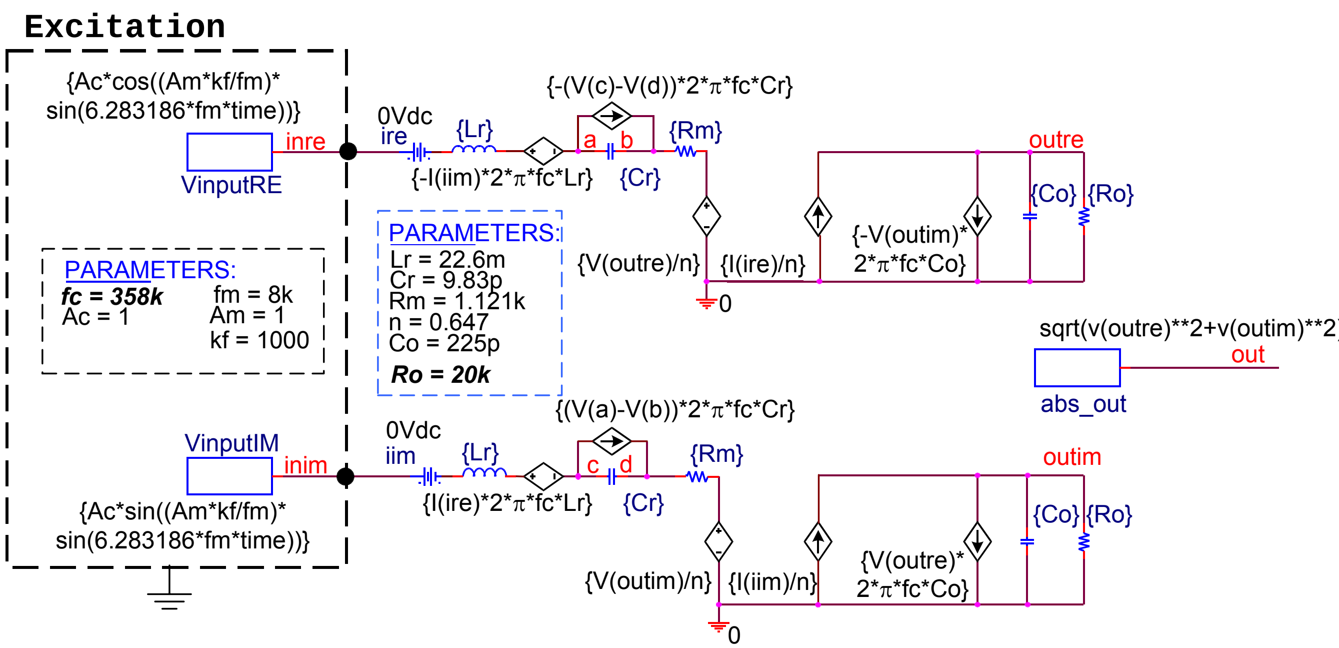

Fig. 4. Schematics of envelope simulation model of the piezoelectric transformer circuit (Fig. 3) excited by a frequency modulated signal (SPICE/OrCAD evaluation Version 9.2).

III. Unified Model for Envelope Simulation

The envelope simulation model will now be extended to include not only the case of large signal (time domain, TRAN) analysis, but also the small signal analysis case (frequency domain, AC) and the DC sweep case. The latter is a steady state analysis carried out for a range of carrier frequencies (f_c). Since the large signal SPICE-compatible circuit of phasor model is linear, the circuit itself is applicable as is for all three types of analysis. The difference will be in the excitation signals that is, the expression for the real part U_1 and the imaginary part U_2. The required excitation signals will be, in general, different for each modulation scheme and for each type of analysis (TRAN, DC and AC).

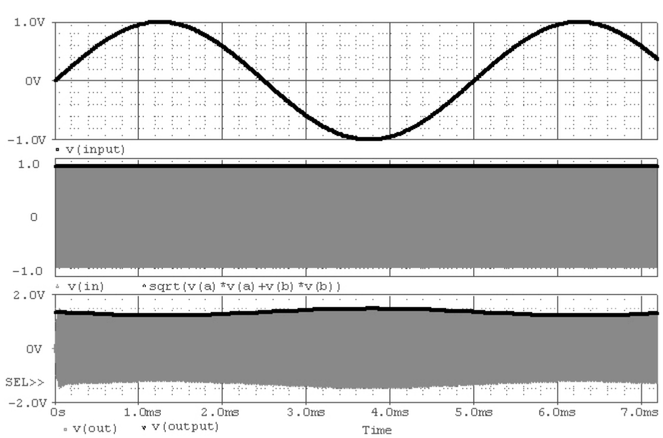

Fig. 5. Transient and envelope simulation results for FM modulation. Upper curve: modulating signal. Middle curves: frequency modulated input carrier signal (gray) and envelope of the input signal (black line). Lower plot: PT's output signal (gray curve) and its envelope (black curve) obtained by the envelope simulation model of Fig. 4.

TRAN analysis cases. The excitations of the time domain analyses follow equations (13), (14) for AM, (15), (16) for FM and (17), (18) for PM.

DC analysis cases. In this steady state analysis, the simulation is repeated for a number of carrier frequencies within a specified range. That is, the amplitude of the carrier frequency (A_c) is constant; the amplitude of the modulation frequency (A_m) is zero; and the 'DC' sweep is over the carrier frequency (f_c). Under these conditions, the excitation for all analysis types (AM, FM, PM) are found from (13) - (18) to be:

$$U_1 = A_c \tag{19}$$

$$U_2 = 0 \tag{20}$$

AC analysis case. Small signal analysis is carried out after linearization of the circuit and excitation sources. Since the circuit is linear, it will be left as is by the simulator when running the AC analysis. The excitation sources, however, need to be modified. This can be accomplished by: (a) reducing the large signal expressions to the small signal case (i.e. narrow band modulation) and (b) replacing the time dependent representation of the TRAN sources by phasors. This will be exemplified next by considering the case of FM modulation. For small signal $A_m \to 0$, and (15) and (16) reduce to:

$$U_1 = A_c \tag{21}$$

$$U_2 = \frac{A_c k_f}{f_m} A_m \sin(2\pi f_m t) \tag{22}$$

Hence, in AC analysis U_1 need to be represented by a DC source of magnitude A_c:

$$U_1 = A_c \tag{23}$$

and U_2 by:

$$U_2 = \frac{A_c k_f}{f_m}\tilde{A}_m \tag{24}$$

or:

$$U_2 = 2\pi A_c k_f \int \tilde{A}_m dt \tag{25}$$

where $\tilde{A}_m$ is a phasor of magnitude A_m.

Hence, for AC analysis the port 'inre' need to be fed by a DC source of magnitude A_c and the port 'inim' by an AC source of magnitude A_m followed by an integrator (a standard behavioral model) and multiplied by $A_c k_f$. A proposed implementation in OrCAD Version 9.2 is shown in Fig. 6.

A summary of the excitation sources and conditions for each type of modulation and analysis is given in Table I. For all modulation schemes, the variables that are swept in each analysis are as follows: 'time' for TRAN analysis, carrier frequent (f_c) for DC analysis, and 'frequency' for AC analysis. Each type of analysis calls for a unique real excitation (inre) and imaginary (inim) excitation while the circuit itself is left as is. The excitation sources used in each type of analysis should be consistent with the analysis. That is, in TRAN analysis, DC and time dependent sources are used; in AC analysis, DC and AC sources are used, while in DC analysis only DC sources are used.

IV. Simulation Results

The proposed unified envelope simulation method was tested by comparing the envelope simulation results obtained by the proposed model to the results of full simulation (that includes the carrier) of the piezoelectric circuit of Fig. 3. Typical simulation results for FM excitation are given in Figs. 7, 8.

The 'DC' simulation results of Fig. 7 were obtained by applying the source as shown in Table I and sweeping the parameter {f_c}, that is the carrier frequency, over the

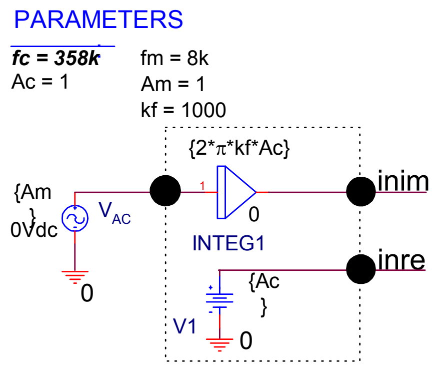

Fig. 6. PSpice implementation of the decomposed sources for AC-sweep envelope simulation for the case of FM excitation.

frequency range of 340kHz to 370kHz. The simulation was repeated for different loads by applying the 'parametric sweep' option. The simulation results of Fig. 7 represent the frequency dependence of the output of a piezoelectric transformer for different load values.

Fig. 8 compares the results of AC-envelope simulation, according to proposed method, to the results of conventional simulation of the original circuit as is. The AC small signal response was obtained from the full circuit TRAN simulation by running the simulation for many (time domain) FM modulated signals and measuring the resulting steady state envelopes [2-4]. Fig. 8 shows the results for the conventional

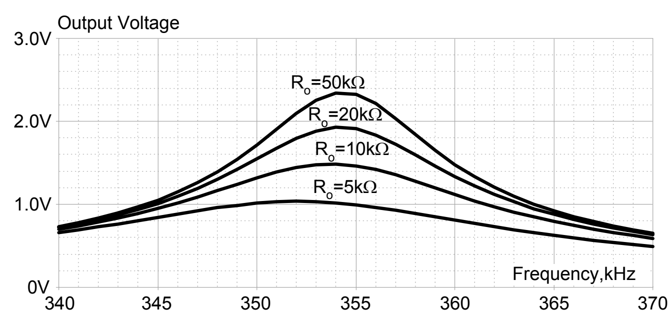

Fig. 7. Steady state output voltage as a function of excitation (carrier) frequency of the simulation model (Fig. 4) for different loads R_o obtained by applying DC-envelope analysis.

TABLE I
Real (inre) and Imaginary (inim) Signals Required for Carrying Out Small Signal, Large Signal and DC Simulation in the AM, FM and PM Cases.

Modulation type	Analysis	Carrier (A_c=const)	Sweep Variable	inre	inim
AM	Large signal	f_c=const	time	$A_c(1+k_a A_m \sin(2\pi f_m t))$	0
	DC	f_c=variable	f_c	A_c	0
	Small signal	f_c=const	frequency	$A_c(1+k_a)\tilde{A}_m$	0
FM	Large signal	f_c=const	time	$A_c\cos(2\pi k_f \int A_m \sin(2\pi f_m t)dt)$	$A_c\sin(2\pi k_f \int A_m \sin(2\pi f_m t)dt)$
	DC	f_c=variable	f_c	A_c	0
	Small signal	f_c=const	frequency	A_c	$2\pi k_f A_c \int \tilde{A}_m dt$
PM	Large signal	f_c=const	time	$A_c\cos(k_p A_m \sin(2\pi f_m t))$	$A_c\sin(k_p A_m \sin(2\pi f_m t))$
	DC	f_c=variable	f_c	A_c	0
	Small signal	f_c=const	frequency	A_c	$k_p A_c \tilde{A}_m$

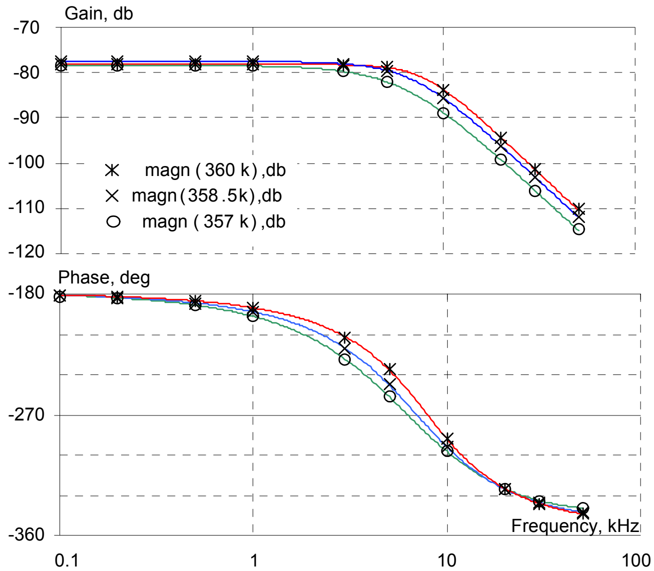

Fig. 8. Small-signal (AC) simulation results of the PT phasor model Fig. 4 excited by a FM signal (lines) compared to the results of multiple runs of transient analysis (symbols) for different carrier frequencies f_c.

simulation runs (phase-shift and amplitude) and the results of the AC-envelope simulation for the case of FM. Fig. 8 confirms that the results obtained by the two methods are identical. However while the results of the AC–envelope simulation were obtain is few seconds, running the set of transient simulations and extracting the results took hours.

V. Discussion and Conclusions

The major contribution of this work is the novel modeling approach that facilitates AC-envelope simulation. The two major advantages of the method are: (a) the use of the AC analysis capability of SPICE rather the tedious extraction of the small signal response from sets of TRAN analyses, and (b) the simplicity of the excitation sources that can be used to run, as is and without further analytical deviation (such as small signal perturbation), TRAN, DC and AC analysis.

As simulation tools are developed, Power Electronics will follow the trend of other areas in electronics in which a major part of the engineering design work is carried out by simulation. The proposed modeling method could be useful to the engineer and to the researcher since it provides the tool to explore systems that are very difficult to be examined analytically. In particular, the method can be advantageously used to extract the small signal responses needed for the design of the control loops in feedback systems. The simple and unified approach and the short simulation time are making the approach easy to use and user friendly.

References

[1] C. Rim and G. Cho, "Phasor Transformation and its Application to the DC/AC Analyses of Frequency Phase-Controlled Series Resonant Converters", *IEEE Transactions on Power Electronics*, vol. 5, no. 2, pp. 201 – 211, April 1990.

[2] S. Ben-Yaakov, S. Glozman, and R. Rabinovici, "Envelope Simulation by SPICE-Compatible Models of Electric Circuits Driven by Modulated Signals," *IEEE Transactions on Industrial Electronics*, vol. 47, no. 1, pp. 222 – 225, February 2000.

[3] S. Ben-Yaakov, S. Glozman, and R. Rabinovici, "Envelope Simulation by SPICE-Compatible Models of Linear Electric Circuits Driven by Modulated Signals," *IEEE Transactions on Industry Applications*, vol. 37, no. 2, pp. 527 - 533, March/April 2001.

[4] S. Glozman and S. Ben-Yaakov, "Dynamic Interaction Analysis of HF Ballast and Fluorescent Lamps Based on Envelope Simulation," *IEEE Transactions on Industry Applications*, vol. 37, no. 5, pp. 1531 – 1536.

[5] S. Lineykin and S. Ben-Yaakov, "Feedback Isolation by Piezoelectric Transformers: A Feasibility Study," *Power Conversion and Intelligent Motion, PCIM-2000*, pp. 175-181, June 2000.

[6] S. Ben-Yaakov and S. Lineykin, "Frequency tracking to maximum power of piezoelectric transformer HV converters under load variations," *PESC-2002*, pp. 657-662, June 2002.

[7] G. Ivensky, I. Zafrany, and S. Ben-Yaakov, "Generic operational characteristics of piezoelectric transformers," *IEEE Transactions on Power Electronics,* vol. 17, no. 6, pp. 1049-1057, November 2002.