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

Simon Lineykin and Shmuel Ben-Yaakov, *Member, IEEE*

***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 amplitude modulation (AM), frequency modulation (FM), and phase modulation (PM) modulation schemes and is demonstrated on a piezoelectric transformer circuit. The model is based on the equivalent circuit approach and can be run on any modern electronic circuit simulator. An excellent agreement was found between the simulation results according to the new unified envelope model, full simulation, and experimental results.**

***Index Terms*—Envelope simulation, piezoelectric transformers, resonant inverters, simulation program with integrated circuit emphasis (SPICE).**

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 outputs of lamps powered by electronic ballasts is to control the drive frequency of the ballasts. 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 envelope of the signal that is sensed at the output (Fig. 1). Consequently, the analysis and simulation of the small-signal transfer functions, needed for controller design and stability analysis, is rather complicated. Earlier studies attempted to tackle the problem of small-signal analysis of carrier-driven system by one of the following approaches: 1) signal perturbation and linearization of the state-space equations [1]–[4] or 2) signal perturbation and linearization of the system's equations after phasor transformation [5]–[12]. The common denominator of all these methods is the need for deriving analytically the small-signal expressions before calculation or simulation can be carried out. This normally requires rather tedious manual work, which has to be repeated for each system. Furthermore, the resulting expressions are exceedingly complex to the point that they might be too involved for the nonanalytical expert such as the common design engineer.

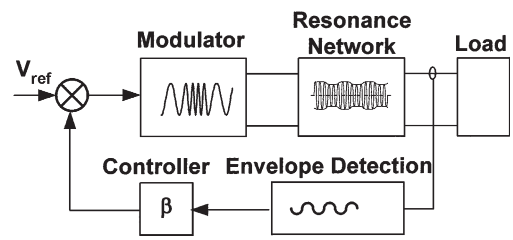

Fig. 1. Resonant converter under closed-loop control.

As demonstrated earlier [6]–[9], envelope simulation by a simulation program with integrated circuit emphasis (SPICE) compatible model could be used to simplify the extraction of the large-signal response of the carrier-driven linear power electronics systems. The method can also be used to simulate nonlinear systems by applying equivalent linear networks to emulate the behavior of the nonlinear parts of the system (such as the rectifier in a resonant dc–dc converter [13]–[17]). Obviously, linearization is required in any small-signal analysis method (e.g., [1] and [3]) that attempts to handle nonlinear systems.

The earlier SPICE compatible envelope-simulation method was confined to large-signal (time domain) simulation. Small-signal responses can be extracted from the envelopes of the time-domain-simulation runs by repeating the simulation for a range of modulating signal [8]. This, of course, is a tedious process since each frequency domain point requires a lengthy time-domain-simulation run.

The objective of this study is 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 perturbation and 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 modulated signal [9], [18]–[20]. The details of the basic envelope-simulation model are given in [7] and an example of its application in [8].

Manuscript received July 30, 2003; revised October 19, 2005. Abstract published on the Internet March 18, 2006. An earlier version of this paper was presented at the 2003 IEEE Power Electronics Specialist Conference (PESC'03) Acapulco, Mexico, June 15–19. This work was supported by The Israel Science Foundation under Grant 113/02 and by the Paul Ivanier Center for Robotics and Production Management.

The authors are with the Power Electronics Laboratory, Department of Electrical and Computer Engineering, Ben-Gurion University of the Negev, Beer-Sheva 84105, Israel (e-mail: sby@ee.bgu.ac.il).

Digital Object Identifier 10.1109/TIE.2006.874421

It should be noticed that the attribute “SPICE compatible” used here is meant to imply that the proposed model is based on the equivalent circuit approach and, as such, can be run on any electronic circuit simulator. Furthermore, by rewriting the state space equations of the equivalent circuits, one can run the simulation on MATLAB as was demonstrated earlier in connection with a behavioral model of pulsewidth-modulation (PWM) converters [21].

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.

Equation (1) can also be written in complex form as

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

or as

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

where

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

and

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

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

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

As demonstrated earlier in [7] and [8], the SPICE compatible envelope-simulation circuit can be developed by means of the following stages:

1) duplicating the circuit to create the real part and the imaginary part;
2) replacing reactive elements (*L*, *C*), as shown in Fig. 2, into the real and imaginary sections of the circuit;
3) placing two excitation sources for real and imaginary parts ($U_1(t)$ and $U_2(t)$) but excluding the carrier;
4) 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 $c(t)$

$$m(t) = A_m \sin(2\pi f_m t) \tag{7}$$

$$c(t) = A_c \cos(2\pi f_c t) \tag{8}$$

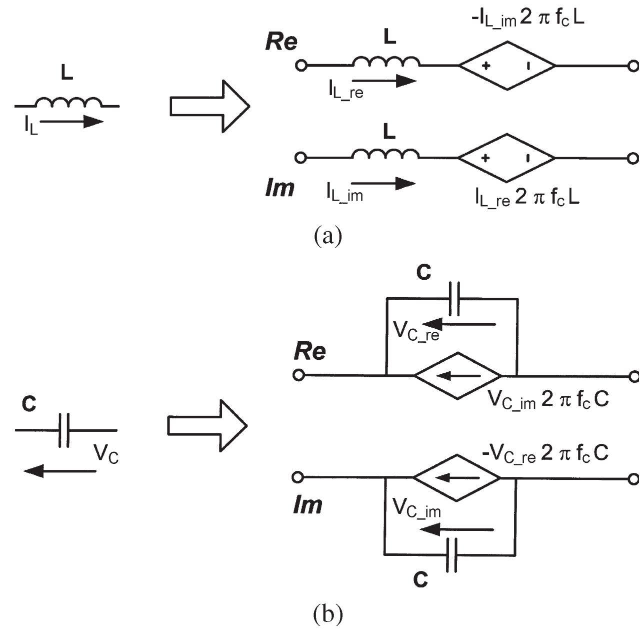

Fig. 2. Replacement of reactive elements by equivalent circuits for envelope simulation. (a) Replacing an inductor by an inductor and dependent voltage source. (b) Replacing a capacitor by a capacitor and dependent current source.

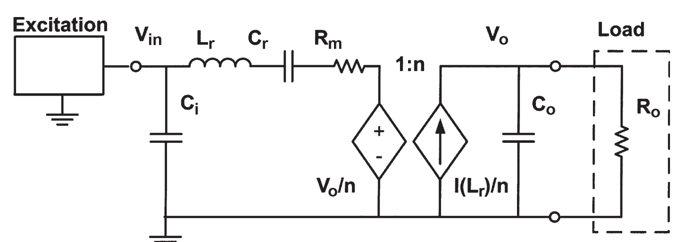

Fig. 3. Equivalent circuit of a piezoelectric transformer loaded by a resistor R_o and driven by a modulated signal (AM, FM, or PM).

where A_m and A_c are the amplitudes of the modulating signal $m(t)$ and carrier signal $c(t)$, respectively, f_c is the frequency of the carrier, and f_m is the frequency of the modulating signal.

The amplitude modulation (AM) signal is described by

$$\begin{aligned} u(t) &= (1 + k_a m(t))\, c(t) \\ &= A_c\,(1 + k_a A_m \sin(2\pi f_m t)) \cos(2\pi f_c t). \end{aligned} \tag{9}$$

A frequency modulation (FM) signal for any modulating signal $m(t)$

$$u(t) = A_c \cos\left(2\pi f_c t + 2\pi k_f \int m(t)dt\right) \tag{10}$$

and in the single-tone case

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

The phase modulation (PM) signal is expressed as

$$\begin{aligned} u(t) &= A_c \cos(2\pi f_c t + k_p m(t)) \\ &= A_c \cos(2\pi f_c t + k_p A_m \sin(2\pi f_m t)) \end{aligned} \tag{12}$$

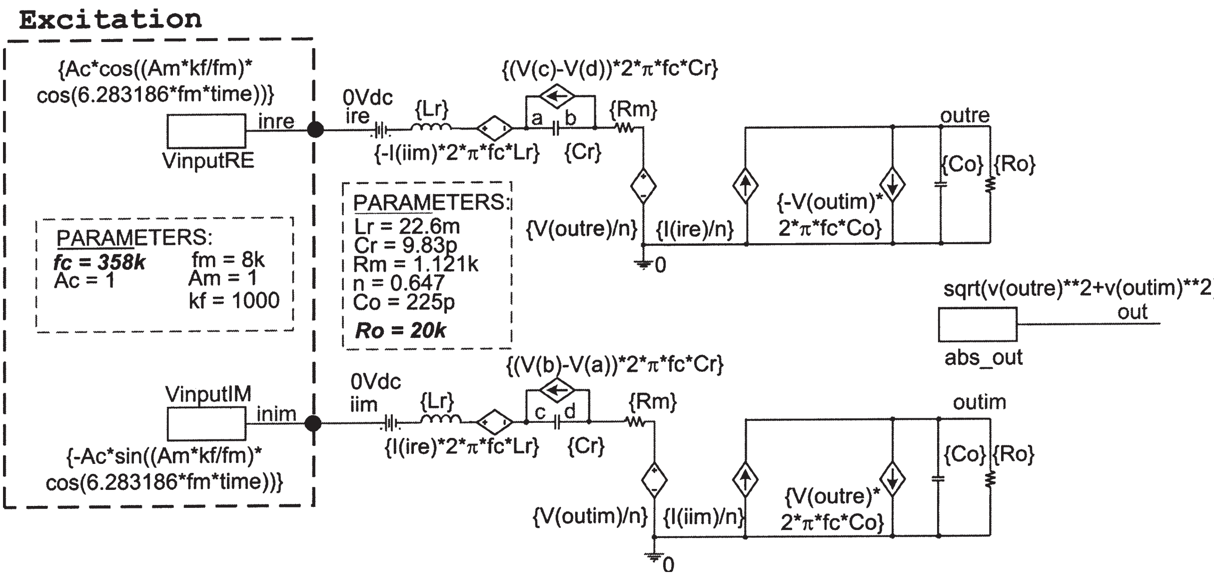

Fig. 4. Schematics of envelope-simulation model of the piezoelectric transformer circuit (Fig. 3) excited by a frequency-modulated signal (PSPICE/OrCAD evaluation Version 9.2).

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 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 FM

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

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

where $\beta = A_m k_f / f_m$.

For PM

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

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

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," which 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.

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 the phasor-transformed 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).

A. TRAN Analysis Cases

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

B. 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 excitations for all modulation types (AM, FM, PM) are found from (13)–(18) to be

$$\begin{cases} U_1 = A_c \\ U_2 = 0. \end{cases} \quad (19)$$

C. AC Analysis Case

Small-signal analysis is carried out after linearization of the circuit and excitation sources. Since the circuit is linear, it will

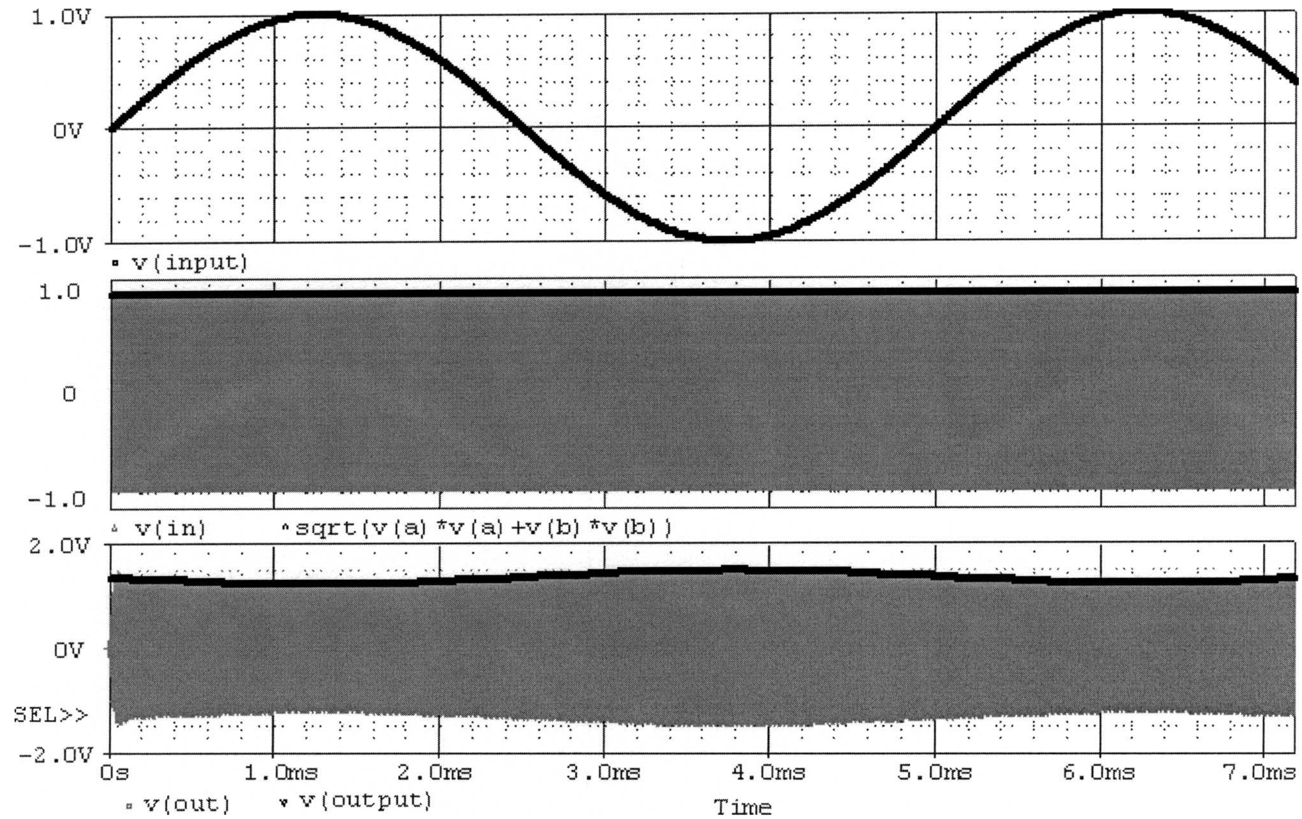

Fig. 5. Transient and envelope-simulation results for FM modulation. Upper curve: modulating signal; middle curves: frequency-modulated input carrier signal (gray) and envelope of the input signal (black line); and lower plot: PT's output signal (gray curve) and its envelope (black curve) obtained by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

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: 1) reducing the large-signal expressions to the small-signal case (i.e., narrowband modulation) and 2) 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 $\mathsf{A_m} \rightarrow 0$, and (15) and (16) reduce to

$$\begin{cases} U_1 = A_\mathrm{c} \\ U_2 = \frac{A_\mathrm{c} k_\mathrm{f}}{f_\mathrm{m}} A_\mathrm{m} \cos(2\pi f_\mathrm{m} t). \end{cases} \tag{20}$$

Hence, in ac analysis, U_1 needs to be represented by a DC source of magnitude A_c and U_2 by

$$U_2 = 2\pi A_\mathrm{c} k_\mathrm{f} \int \widetilde{A}_\mathrm{m} dt \tag{21}$$

where $\widetilde{A}_\mathrm{m}$ is a phasor of magnitude A_m.

Hence, for AC analysis, the port "inre" needs 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 $2\pi A_\mathrm{c} k_\mathrm{f}$. A proposed implementation in OrCAD Version 9.2 is shown in Fig. 6(a). Similar transformations produce the AC sources for other modulation types: AM [Fig. 6(b)] and PM [Fig. 6(c)].

A summary of the excitation sources and conditions for each type of modulation and analysis is given in Table I. For all

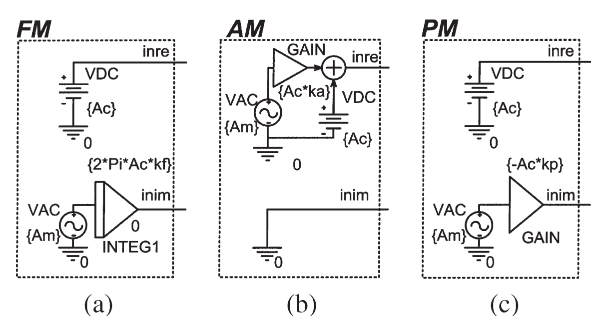

Fig. 6. PSPICE implementation of the decomposed sources for AC-sweep envelope simulation. Real and imaginary components of the source for small-signal envelope simulation for (a) FM, (b) AM, and (c) PM. A_c is the amplitude of the carrier wave, A_m is the amplitude of the modulating signal, k_f is the coefficient of frequency modulation, k_a is the coefficient of amplitude modulation, and k_p is the coefficient of phase modulation.

modulation schemes, the variables that are swept in each analysis are as follows: "time" for TRAN analysis, carrier frequency (f_c) for dc analysis, and "frequency" for AC analysis. Each type of analysis calls for a unique real excitation (inre) and imaginary excitation (inim), 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.

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***	***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	$-k_p A_c \tilde{A}_m$

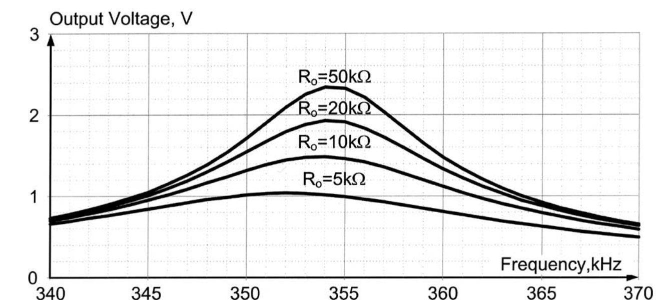

Fig. 7. Steady-state output voltage as a function of excitation (carrier) frequency of the simulation model (Fig. 4) for different loads R_o obtained by applying dc-envelope analysis.

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 (which includes the carrier) of the piezoelectric circuit of Fig. 3. The equivalent circuit of the experimental PT includes: $C_i = 200$ pF, $C_o = 225$ pF, $L_r = 22.6$ mH, $C_r = 9.83$ pF, $R_m = 1.121$ kΩ, and $N = 0.647$. Typical simulation results are given in Figs. 7 and 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 frequency range of 340–370 kHz. 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 voltage of a piezoelectric transformer for different load-resistance values.

Fig. 8 compares the results of ac-envelope simulation, according to the 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 [6]–[8]. Fig. 8 shows the results for the conventional 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 obtained in a few seconds, running the set of transient simulations and extracting the results took hours.

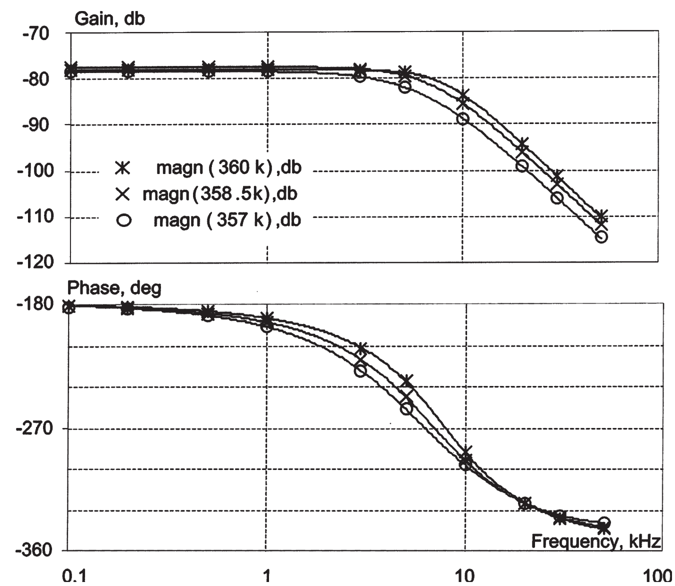

Fig. 8. Small-signal (ac) simulation results of the PT phasor model Fig. 4 excited by an FM signal (lines) compared to the results of multiple runs of transient analysis (symbols) for different carrier frequencies f_c.

V. EXPERIMENTAL

The small-signal transfer function of the PT under study was measured by applying the experimental setup of Fig. 9. The PT was driven by a modulated signal; the output was buffered (to control loading), rectified by a voltage doubler, and the rectified signal was buffered again.

Typical results of the experimental measurements (dashed lines) and simulations (heavy lines) are shown in Figs. 10 and 11. The small disagreement between the experimental data and the simulation results is probably due to the fact that the parameters of the PT model are in slight error. This is due to experimental limitations of practical parameter-extraction procedures and the nonlinearity of the PT. Notwithstanding the slight experimental errors, the good agreement between the experimental and simulation results supports the conjecture that small-signal envelope simulation is a viable tool to explore the transfer function of a PT under various modulating conditions. Further information on PT excitation by modulated signals and their envelope simulations can be found in [15] and [16].

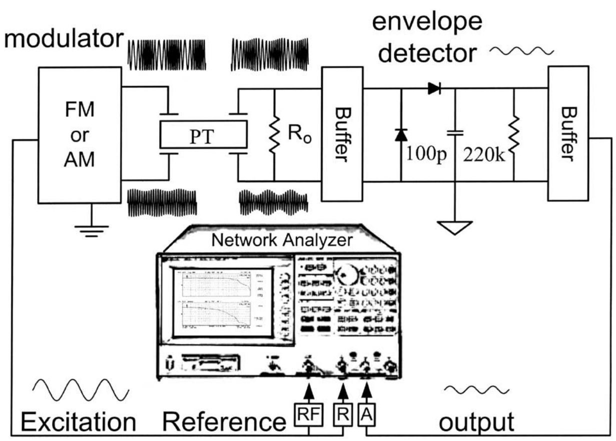

Fig. 9. Experimental setup. The network analyzer measures the ratio between the signals (A)—envelope of the output signal, and the reference terminal (R) which is connected to the excitation port (RF).

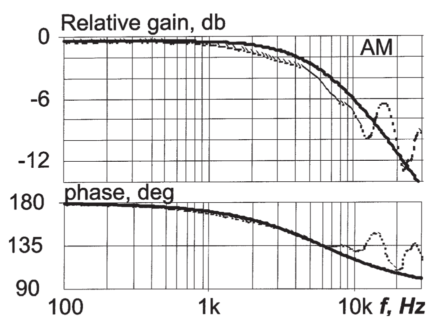

Fig. 10. Small-signal transfer function of the experimental PT under AM excitation. Solid line: small-signal envelope simulation; dashed line: experimental; and carrier frequency: 353 kHz.

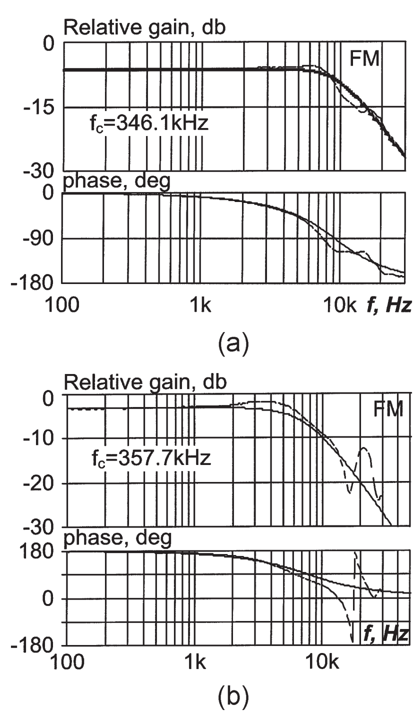

Fig. 11. Small-signal transfer function of the experimental PT under FM excitations. Solid line: small-signal envelope simulation; dashed line: experimental; carrier frequency. (a) $f_c = 346.1$ kHz. (b) $f_c = 357.7$ kHz.

VI. Discussion and Conclusion

The major contribution of this paper is the novel-modeling method that facilitates an SPICE compatible ac-envelope simulation by the equivalent circuit approach. The two major advantages of the method are: 1) the use of the AC-analysis capability of SPICE rather than the tedious extraction of the small-signal response from sets of TRAN analysis and 2) the simplicity of the excitation sources that can be used to run as is and without further analytical derivation (such as small-signal perturbation), TRAN, DC, and AC analyses. A good agreement was found between the simulation results according to the proposed method, full circuit simulation, and experimental results.

As simulation tools are developed, power electronics will follow the trend of other electronic areas, 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 method easy to use and user friendly.

References

[1] A. F. Witulski, A. F. Hernandez, and R. Erickson, "Small signal equivalent circuit modeling of resonant converters," *IEEE Trans. Power Electron.*, vol. 6, no. 1, pp. 11–27, Jan. 1991.

[2] V. Vorperian, "Approximate small-signal analysis of the series and the parallel resonant converters," *IEEE Trans. Power Electron.*, vol. 4, no. 1, pp. 15–24, Jan. 1989.

[3] J.-H. Cheng, A. F. Witulski, and J. U. Vollin, "A small-signal model utilizing amplitude modulation for the class-d converter at fixed frequency," *IEEE Trans. Power Electron.*, vol. 15, no. 6, pp. 1204–1211, Nov. 2000.

[4] E. X. Yang, F. C. Lee, and M. M. Jovanovic, "Small-signal modeling of series and parallel resonant converters," in *Proc. IEEE APEC*, 1992, pp. 786–792.

[5] C. Rim and G. Cho, "Phasor transformation and its application to the DC/AC analyses of frequency phase-controlled series resonant converters," *IEEE Trans. Power Electron.*, vol. 5, no. 2, pp. 201–211, Apr. 1990.

[6] S. Ben-Yaakov, S. Glozman, and R. Rabinovici, "Envelope simulation by SPICE-compatible models of electric circuits driven by modulated signals," *IEEE Trans. Ind. Electron.*, vol. 47, no. 1, pp. 222–225, Feb. 2000.

[7] ——, "Envelope simulation by SPICE-compatible models of linear electric circuits driven by modulated signals," *IEEE Trans. Ind. Appl.*, vol. 37, no. 2, pp. 527–533, Mar./Apr. 2001.

[8] S. Glozman and S. Ben-Yaakov, "Dynamic interaction analysis of HF ballast and fluorescent lamps based on envelope simulation," *IEEE Trans. Ind. Appl.*, vol. 37, no. 5, pp. 1531–1536, Sep./Oct. 2001.

[9] S. Lineykin and S. Ben-Yaakov, "A unified SPICE compatible model for large and small signal envelope simulation of linear circuits excited by modulated signals," in *Proc. IEEE PESC*, Acapulco, Mexico, Jun. 2003, pp. 1205–1209.

[10] Y. Yin, R. Zane, R. Erickson, and J. Glaser, "Dynamic analysis of frequency-controlled electronic ballasts," in *Conf. Rec. 37th IEEE-IAS Annu. Meeting*, Oct. 2002, pp. 685–691.

[11] G. Spiazzi and S. Buso, "Small-signal analysis of cold cathode fluorescent lamp ballasts," in *Proc. IEEE PESC*, Recife, Brazil, Jun. 2005, pp. 2783–2789.

[12] Y. Yin, R. Zane, R. Erickson, and J. Glaser, "Direct modeling of envelope dynamics in resonant inverters," in *Proc. IEEE PESC*, 2003, vol. 3, pp. 1313–1318.

[13] R. Steigerwald, “Analysis of a resonant dc–dc converter with capacitive output filter,” *IEEE Trans. Ind. Electron.*, vol. IE-32, no. 2, pp. 439–444, Nov. 1985.

[14] G. Ivensky, M. Shvartsas, and S. Ben-Yaakov, “Analysis and modeling of a voltage doubler rectifier fed by a piezoelectric transformer,” *IEEE Trans. Power Electron.*, vol. 19, no. 2, pp. 542–549, Mar. 2004.

[15] S. Lineykin and S. Ben-Yaakov, “Feedback isolation by piezoelectric transformers: Comparison of amplitude to frequency modulation,” *HAIT J. Sci. Eng.*, vol. 2, no. 5/6, pp. 830–847, Dec. 2005.

[16] ——, “Feedback isolation by piezoelectric transformers: Comparison of amplitude to frequency modulation,” in *Proc. IEEE PESC*, Aachen, Germany, Jun. 2004, pp. 1834–1840.

[17] J. A. Oliver, C. Fernandez, R. Prieto, and J. A. Cobos, “Circuit oriented model of rectifiers for large signal envelope simulation,” in *Proc. IEEE PESC*, Recife, Brazil, Jun. 2005, pp. 2771–2776.

[18] S. Lineykin and S. Ben-Yaakov, “Feedback isolation by piezoelectric transformers: A feasibility study,” in *Proc. PCIM*, Jun. 2000, pp. 175–181.

[19] S. Ben-Yaakov and S. Lineykin, “Frequency tracking to maximum power of piezoelectric transformer HV converters under load variations,” in *Proc. IEEE PESC*, Jun. 2002, pp. 657–662.

[20] G. Ivensky, I. Zafrany, and S. Ben-Yaakov, “Generic operational characteristics of piezoelectric transformers,” *IEEE Trans. Power Electron.*, vol. 17, no. 6, pp. 1049–1057, Nov. 2002.

[21] S. Ben-Yaakov and D. Adar (Edry), “Average models as tools for studying the dynamics of switch mode DC–DC converters,” in *Proc. IEEE PESC*, Taipei, Taiwan, R.O.C., 1994, pp. 1369–1376.

Simon Lineykin received the B.Sc. degree in mechanical engineering in 1997 and the M.S. degree in electrical engineering in 2000 from Ben-Gurion University of the Negev, Beer-Sheva, Israel, where he is currently working toward the Ph.D. degree in electrical engineering.

His research interests are modeling and emulation of the physical processes and active cooling systems using Peltier effect.

Shmuel (Sam) Ben-Yaakov (M’87) received the B.Sc. degree in electrical engineering from the Technion, Haifa, Israel, in 1961, and the M.S. and Ph.D. degrees in engineering from the University of California, Los Angeles (UCLA), in 1967 and 1970, respectively.

He is presently a Professor in the Department of Electrical and Computer Engineering, Ben-Gurion University of the Negev, Beer-Sheva, Israel, and heads the Power Electronics Group there. His current research interests include power electronics, circuits and systems, electronic instrumentation, and engineering education. He also serves as a Consultant to commercial companies in the areas of analog and power electronics.