Information Technologies, Management and Society, 2014, Volume 7, No. 1, 48 - 54

MEASUREMENTS OF ELECTRICAL CHARGE AND ENERGY IN PSPICE SIMULATION ENVIRONMENT

A. FERIKOĞLU[A], Y. SARI[B], R. KÖKER[C] AND I. PEHLIVAN[D]

[a)]Department of Electrical and Electronics Engineering, Sakarya University, 54187, Serdivan, Sakarya, Turkey
e-mail: af@sakarya.edu.tr, fax: +902642956424, phone: +902642956449
[b)]Department of Electronics and Automation, Hendek Vocational High School, Sakarya University, 54300, Hendek, Sakarya, Turkey
e-mail: sari@sakarya.edu.tr, fax: +902646147788, phone: +902646147766
[c)]Department of Electrical and Electronics Engineering, Sakarya University, 54187, Serdivan, Sakarya, Turkey
e-mail: rkoker@sakarya.edu.tr, fax: +902642956424, phone: +902642955900
[d)]Department of Electrical and Electronics Engineering, Sakarya University, 54187, Serdivan, Sakarya, Turkey
e-mail: ipehlivan@sakarya.edu.tr, fax: +902642956461, phone: +902642955900

ABSTRACT

Modelling of electrical components and devices that exhibit their physical characteristics remain an attractive area of research. The models for components and devices together with operational blocks aid in the design as well as performance analysis and measurement of various circuits by allowing for detailed simulation of the circuit being tested. This paper describes application of Orcad PSpice on the measurement of such quantities as electrical charge and energy over a simulation example together with its theoretical background. The example is selected considering its educational value in that it illustrates a scientific definition of voltage and the basic law of energy conservation. Simulation results have shown that Orcad Pspice is highly easy and effective for conducting measurement simulations of electrical charge and energy as well as having a visual understanding about the underlying theory, which are valuable from the viewpoint of technical education.

Keywords: Simulation, Pspice, electrical charge.

1. INTRODUCTION

Computer simulation is a common technique for predicting the real world behaviour of a circuit. Although simulation software only reflects the capability of the model used in the back-plane and they cannot substitute the real time experimentation, they have proven educationally useful due to their easy to construct and visual properties. Orcad PSpice is a general purpose circuit simulator and one of the various versions of SPICE capable of handling analogue logical and mixed signal parts, circuits and systems [1, 2, 3]. It has been extensively used by engineers, Universities and semiconductor manufacturers as a CAD [4–15] tool because of its simplicity and effectiveness. Evaluation versions suited for educational purposes are available and can be found at the [16, 17, 18]. Many parameters of circuits and devices can be profitably simulated using these free educational versions of PSpice. In the simulation procedure different approaches may be followed, namely, electrical component level, subsystem functional block level and, higher system level comprising both of the previous ones.

This paper treats a designed and worked example of energy and charge measurement, which is fruitful from the educational standpoint. Somewhat detailed theoretical background is presented followed by an Orcad Pspice construction and analysis completed with graphics of the measured quantities, which exhibits a visual consistence with the theory. The design and simulation work are conducted in the PSpice

Orcad Capture medium, using a student version of v.9.2. Inside looks of the hierarchical blocks of models designed are also provided.

2. WORKED EXAMPLE

The example comprises the simulation of energy and charge measurement. This example also provides, as a by product, a visual verification for the standard definition of voltage. When a voltage source is involved the preferred term is *electromotor force (emf)*: the total energy supplied to the circuit per unit charge by the source. If a passive component such as a resistor is involved the common word is *potential difference (pd)*: the energy per unit charge converted to other energies by the component. Electrical voltage (emf or pd) is defined in units of joules per coulomb, or volts. The simulation work is performed over the exemplary circuit of Fig. 1

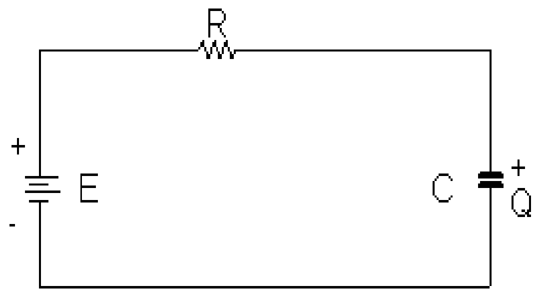

FIGURE 1. EXEMPLARY CIRCUIT OF ENERGY AND CHARGE MEASUREMENT

2.1. THEORETICAL FORMULATION

The straightforward analysis for the circuit of Fig.1 yields the instant values of capacitor voltage, current and energy functions, respectively, as,

$$V_C(t) = E\,(1-e^{-t/\tau}), \quad (1)$$

$$I_C(t) = \frac{E}{R}e^{-t/\tau}, \quad (2)$$

$$W_C(t) = \frac{1}{2}\mathrm{E}^2 C(1-2e^{-t/\tau}+e^{-2t/\tau}), \quad (3)$$

As time goes to infinity (3) becomes,

$$W_C(\infty) = \frac{1}{2}\mathrm{E}^2 C, \quad (4)$$

Energy functions for the resistor and the source, respectively, are,

$$W_R(t) = \frac{1}{2}\mathrm{E}^2 C(1-e^{-2t/\tau}), \quad (5)$$

$$W_S(t) = -\mathrm{E}^2 C(1-e^{-t/\tau}), \quad (6)$$

As time goes to infinity (5) and (6) becomes,

$$W_R(\infty) = \frac{1}{2}\mathrm{E}^2 C, \quad (7)$$

$$W_S(\infty) = -\mathrm{E}^2 C, \quad (8)$$

where the minus sign in (6) and (8) indicates that the source delivers energy to the capacitor and resistor, thus satisfying as expected the conservation of energy ,

$$W_C(t) + W_R(t) + W_S(t) = 0, \quad (9)$$

The capacitor charge is given by,

$$Q_C\ \ t = \int I_C(t) = CE(1-e^{-t/\tau}), \qquad (10)$$

Now, by definition given above the source voltage is

$$\frac{-d(W_S)}{d(Q_C)} = E, \qquad (11)$$

since E is a constant,

$$\frac{-W_S(t)}{Q_C(t)} = E, \qquad (12)$$

2.2. PSPICE SIMULATION

In order to simulate an experiment for testing the given theory, an energy meter and a charge meter are designed in the Pspice medium as what follows:

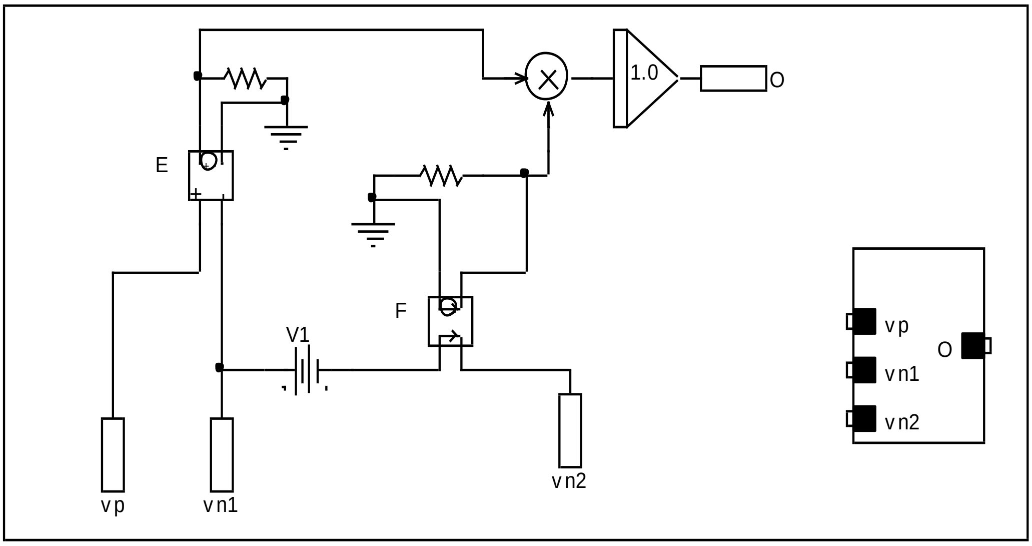

FIGURE 2. PSPICE ENERGY-METER (LEFT) AND ITS BLOCK REPRESENTATION (RIGHT) (WITH UNIT RESISTANCE AND UNIT GAIN OF VCVS AND CCCS)

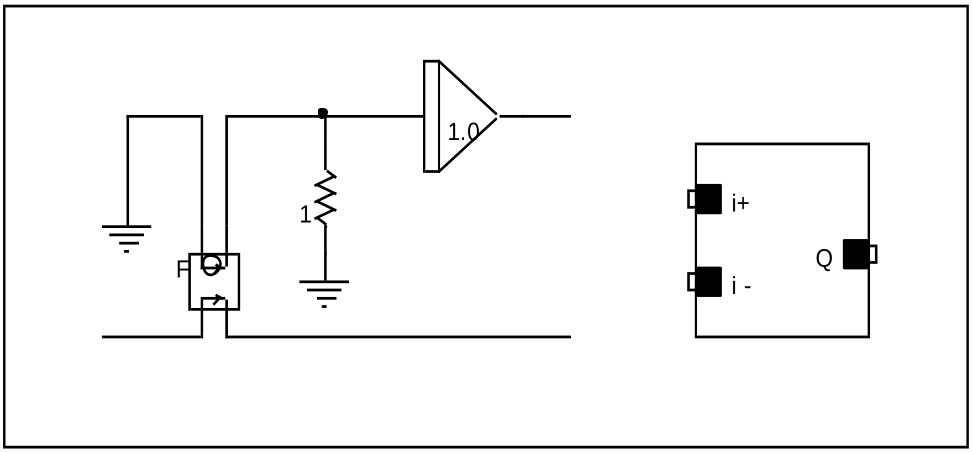

FIGURE 3. PSPICE CHARGE-METER (LEFT) AND ITS BLOCK REPRESENTATION (RIGHT) (WITH UNIT GAIN OF CCCS)

The hierarchical blocks of meters are wired via their connection terminals to the circuit components, where numerical values, for instance, of 5V, 2Ω, and 0.6 F are assigned to the voltage source, resistor, and capacitor, respectively, as below:

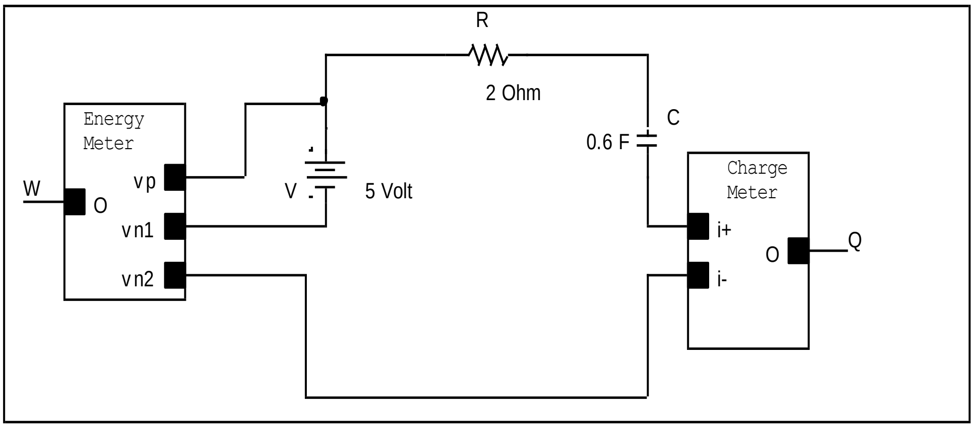

FIGURE 4. ENERGY AND CHARGE MEASUREMENT SIMULATION SETUP

The circuit Figure 4 is time analysed for 7 seconds, which is long enough to assume for the circuit to have reached its final values. The variable graphics thus obtained in Pspice Probe Window are provided below in Figure 5, together with numerical values from the output file in Table 1, which illustrates the definition of the voltage by (12).

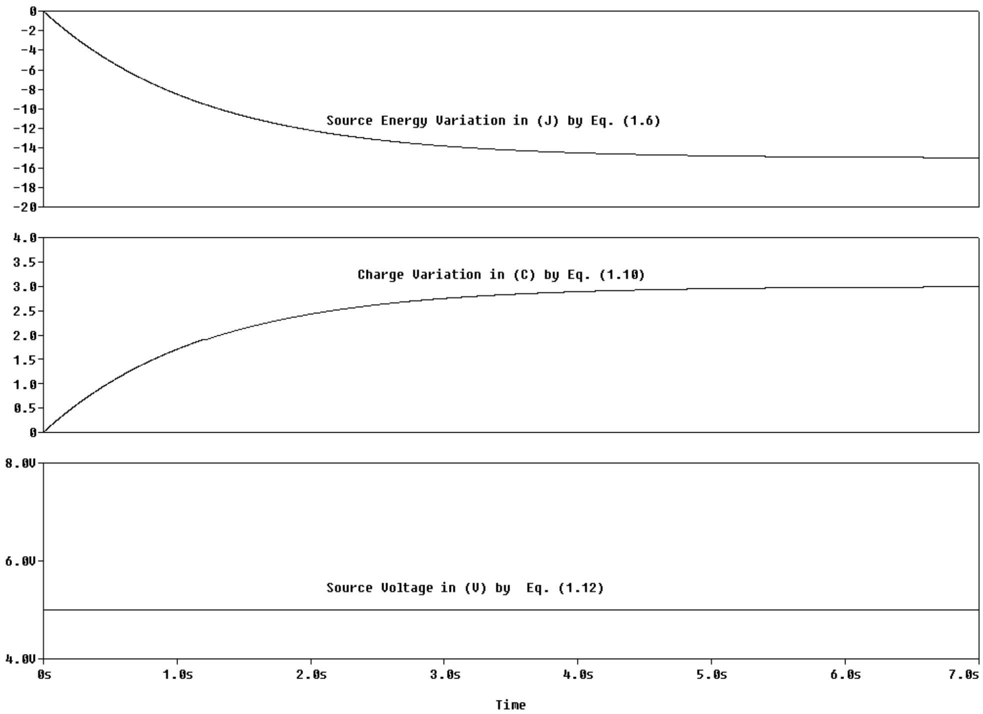

FIGURE 5. FOR THE CIRCUIT OF FIGURE 4., FROM TOP TO DOWN, SOURCE ENERGY FUNCTION GIVEN BY (6), CAPACITOR CHARGE FUNCTION GIVEN BY (10), AND SOURCE VOLTAGE GIVEN BY (11). NOTE: IN THE TOP TWO GRAPHICS VERTICAL AXIS VARIABLES ARE RENAMED IN ENERGY UNIT (J) AND CHARGE UNIT (Q), RESPECTIVELY, SINCE THE INTEGRATORS INSIDE THE METER BLOCKS WORK WITH VOLTS.

TABLE 1. INSTANT VALUES OF THE VARIABLES OF FIGURE 5 (OBTAINED FROM THE "OUTPUT FILE" IN PSPICE MEDIUM BY TAKING DATA EVERY 0.3S)

TIME (s)	Ws (J)	Q (C)	(-Ws/Q) (V)
0.00	-0.025	0.005	5.00
0.30	-3.34	0.67	5.00
0.60	-5.92	1.18	5.00
0.90	-7.93	1.59	5.00
1.20	-9.50	1.90	5.00
1.50	-10.72	2.14	5.00
1.80	-11.67	2.33	5.00
2.10	-12.41	2.48	5.00
2.40	-12.98	2.60	5.00
2.70	-13.43	2.69	5.00
3.00	-13.78	2.76	5.00

Finally, the validity of the energy conservation law given by (9) for the example is observed, from Table 2 and Figure 7 below, after the energy meters connected across the circuit components as in Figure 6.

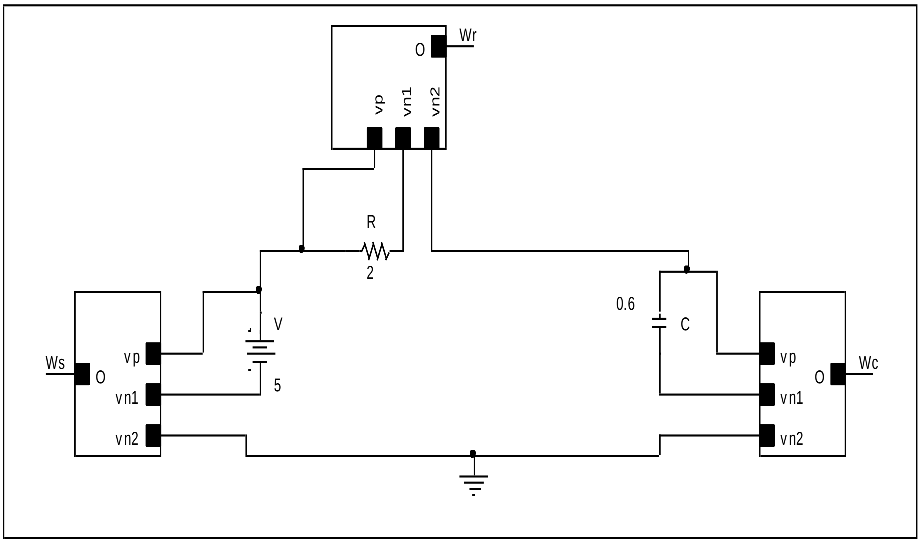

FIGURE 6. CONNECTION OF THE DESIGNED ENERGY METERS TO THE CIRCUIT COMPONENTS IN ORDER TO MEASURE THE ENERGIES OF THE COMPONENTS

TABLE 2. INSTANT VALUES OF THE VARIABLES OF FIGURE 7 (OBTAINED FROM THE "OUTPUT FILE" IN PSPICE MEDIUM BY TAKING DATA EVERY 0.3S), ILLUSTRATING NUMERICALLY THE LAW OF ENERGY CONSERVATION

TIME (s)	Wc (J)	Ws (J)	Wr (J)	Σ W (J)
0	0.00	-0.02	0.02	0.00
0.3	0.37	-3.34	2.97	0.00
0.6	1.16	-5.92	4.76	0.00
0.9	2.09	-7.93	5.84	0.00
1.2	3.00	-9.50	6.50	0.00
1.5	3.82	-10.72	6.90	0.00
1.8	4.53	-11.67	7.14	0.00
2.1	5.12	-12.41	7.29	0.00
2.4	5.61	-12.98	7.38	0.00
2.7	6.00	-13.43	7.43	0.00
3	6.32	-13.78	7.46	0.00

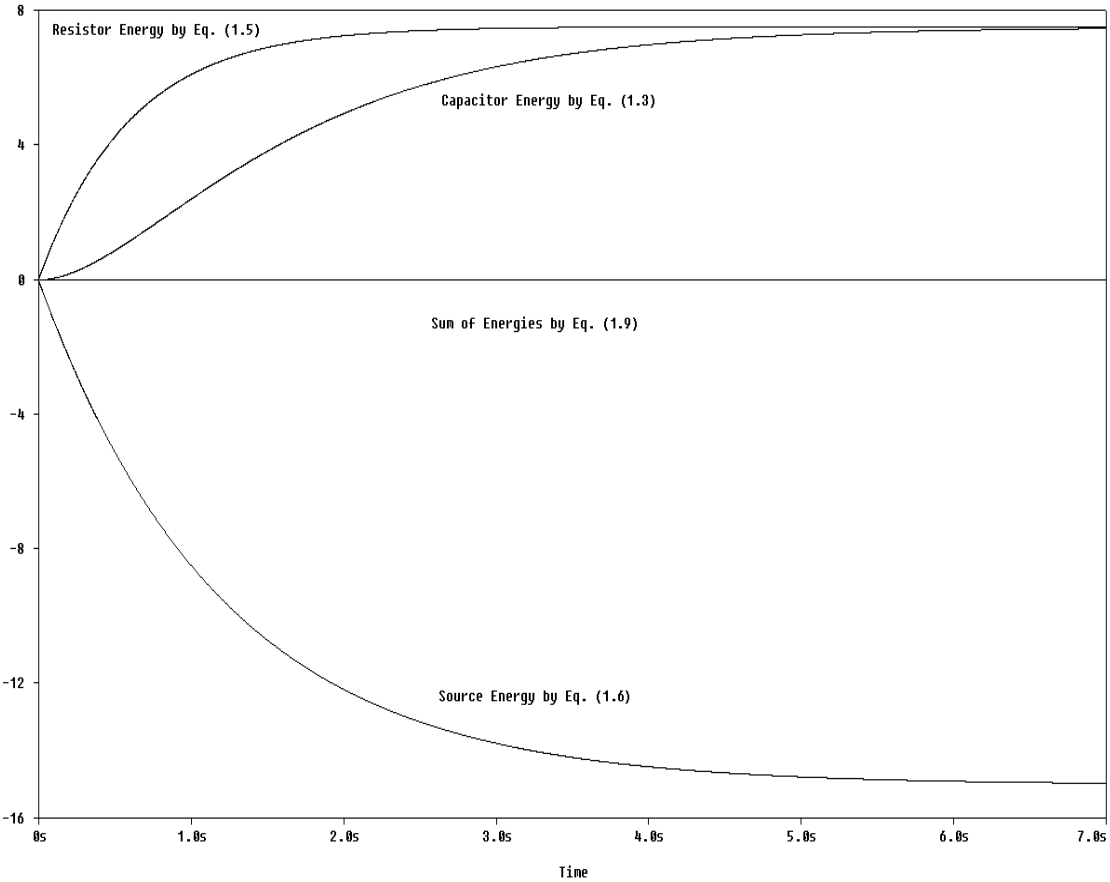

FIGURE 7. IN FIGURE 6, GRAPHICS OF THE ENERGY FUNCTIONS AND THEIR SUM, ILLUSTRATING THE LAW OF ENERGY CONVERSATION

3. CONCLUSIONS

In this study, a charge-meter and an energy meter have been designed and successfully tested in simulation medium, thus exhibiting the educational role of the powerful simulation software Orcad PSpice in Electrical Engineering. In the worked example, utilizing the new meters, the standard definition of voltage has been verified and the rule of conservation of energy has been observed.

In conclusion, the simulation results have proven to be illustrative and consistent to the theory underlying the analysis of electrical circuits and shown that Orcad PSpice, being highly visual, is very useful in the test and measurement of electrical circuits.

REFERENCES

[1] PSpice User's manual, OrCAD Corp. (Cadence Design Systems, Inc.)
[2] OrCAD Capture User's Guide, OrCAD Corp. (Cadence Design Systems, Inc.)
[3] Tobin P. The Role of PSpice in the Engineering Teaching Environment, International Conference on Engineering Education-ICEE 2007
[4] Sarı Y., Ferikoğlu A. (2011) The Role of PSpice in Analog and Mixed-Signal Simulation of Electrical Circuits // 6th International Advanced Technologies Symposium (IATS'11), 16-18 May 2011, Elazığ, Turkey
[5] Portnoy W.M. PSPICE as a Simulation Tool in Teaching Electrodynamics, fie-conference.org
[6] Pedra J., Sainz L., Corcoles F., Lopez R., Salichs M. (2002) PSPICE Computer Model of a Nonlinear Three-

Phase Three-Legged Transformer // Power Engineering Review, *IEEE*, 22(12). – P. 64.

[7] Koleva E., Kolev I., Balabanova I. (2009) Simulation of Optoelectronic Analog Circuits with PSPICE package // Electronics and Electrical Engineering. – Kaunas: Technologija, 8(96), P. 59–61.

[8] Marcinkevičius A., Jasonis V. (2003) The Aspects of Automated Design of Gigahertz Range Integrated Microchips // Electronics and Electrical Engineering. – Kaunas: Technologija, 2(44), 48-53.

[9] Aulas A., Dangelas V. (2001) High Power Amplifier IC for Mobile Phone // Electronics and Electrical Engineering. - Kaunas: Technologija, 5(34), 55-58.

[10] Pehlivan İ., Uyaroğlu Y. (2007) Rikitake Attractor and its Synchronization Application for Secure Communication Systems // Journal of Applied Sciences, 7(2), 232-236.

[11] Pehlivan İ., Uyaroğlu Y. (2007) Simplified Chaotic Diffusionless Lorenz Attractor and its Application to Secure Communication Systems // IET Communications, 1(5), 1015-1022.

[12] Uyaroglu Y, Pehlivan İ. (2010) Nonlinear Sprott94 Case A Chaotic Equation: Synchronization and Masking Communication Applications // Computers and Electrical Engineering, 36(6), 1093-1100.

[13] Pehlivan İ., Uyaroğlu Y., Yoğun M. (2010) Chaotic Oscillator Design and Realizations of the Rucklidge Attractor and its Synchronization and Masking Simulations // Scientific Research and Essays, 5(16), 2210-2219.

[14] Pehlivan İ., Uyaroglu Y. (2010) A New Chaotic attractor from General Lorenz System Family and its Electronic Experimental Implementation // Turkish Journal of Electrical Eng. Comput. Sci., 18(2), 171-184.

[15] Rashid M.H. (2006) Spice for Power Electronics and Electric Power. –CRC/Taylor & Francis, 552 p.

[16] www.ecircuitcenter.com/circuits.htm

[17] www.pspice.com/

[18] www.cadence.com/products/orcad/pages/default.aspx