Estimation of Certain Power in Single-phase Network and DC Circuits

Constantin Daniel Oancea
University Politehnica of Bucharest/Department of Electrical Engineering, Romania, daniel.oancea@upb.ro

Abstract - **Requirement of electrical energy is in order to operate various domestic and industrial devices like appliances, equipment and machinery. Electrical energy per time unit represents electrical power transferred. Well knowing the electrical power is very important because of economical aspects. Electrical power, an important element of an energy system, which considers elements like harmonics and phase between voltage and current, influence the cost and elements component of electrical power grid. This paper presents the evaluation of mark-space method of electrical power system. This method represents a mathematical approach, simulation, and analog implementation solving the power measurement suitable for single-phase network and DC circuits. The schematics developed consist of well-known electronic circuits like the pulse width modulation (PWM), filtering with operational amplifier, and absolute value of signal. This approach avoids multiplying operations that aid error generating. The main purposes are to track the performance of such a circuit in different operating situations and present the schematic. Thus, influence of phase between voltage and current, influence of waveform shape regarding result using analyzed method, and characteristic linearity are example of tests perform. The results confirmed the theoretical principle being analyzed using specific simulation programs like LTSpice or Scilab, but not restricted to them.**

Cuvinte cheie: *putere aparenta, marcaj-spaţiu, simulare, modularea în lăţime a impulsurilor.*

Keywords: *apparent power, mark-space, simulation, PWM.*

I. Introduction

Knowing power generation or consumption with better accuracy is a permanent challenge.

For example, as defined, in circuits for measuring active power, it is necessary to multiply the instantaneous value of voltage and current, followed by the average of the product by using of a low-pass filter [1], [2]. Input blocks in the system provide proportional signals in phase with voltage (*u1*) and current (associated voltage, *u2*). At the output of the multiplier, the signal is proportional to the product $u1 \cdot u2$ and the time-averaging of this product produces a signal proportional to the active power at the output of the low-pass filter. The element that determines the properties and performance of the device is the multiplication circuit. This device may be built in various forms, with digital or analog nonlinear electronic circuits. The existence of the multiplication component brings with it negative aspects [2].

To avoid the multiplication circuit, a mark-space method can be used. The etymology comes from telecommunications, and represents the chop of a signal with a variable duty cycle.

Developing a model of mark-space method involves mathematical operations that avoid multiplication. This paper investigates the usability of the mark-space method. First will describes mark-space method principle, followed by some simulation results. Finally one experiment was developed to reveal viability of the method.

II. Method Principle

"Mark - Space" can be an alternative method of measuring electrical power in single - phase AC circuits. The idea is quite simple and involves several steps: 1. Retrieve the current absolute value; 2. Estimate the root mean square; 3. A rectangular signal is generated with a duty cycle that depends on the value of the previously calculated root mean square (PWM); 4. Create the voltage absolute value; 5. Perform a logic AND operation between the PWM signal and the voltage module; 6. Use a low-pass filter to extract electrical power.

The improvement of the method consists in the elimination of the multiplication circuits, which could cause specific problems. The case considered further is an ideal one in which the harmonics of the current were not taken into account. It is known that the voltage has an approximate sinusoidal shape but the current presents harmonics due to commutations and disturbances phenomena in the power grid.

Signals are also illustrated at different points in Fig.1 to track their transformations along the processing sequences. The component blocks are: absolute value, integrator, pulse width modulation (PWM), electronic switch and low-pass filter (LPF). Each one of these blocks components can be developed with well know analog circuits.

The central element of the mathematical algorithm is a pulse width modulation (PWM) generator, as illustrated in Fig. 2. The pulse width modulation generator gives the performance of the entire mark-space circuit.

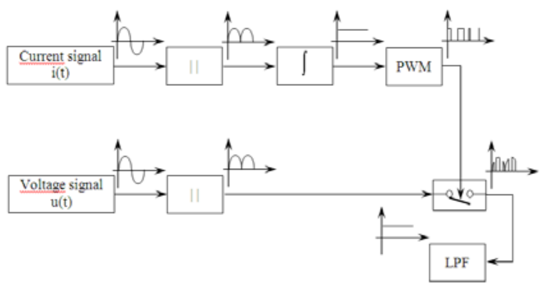

Fig. 1. Successive transformation of signals.

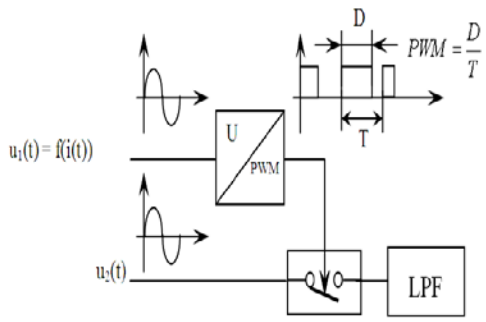

Fig. 2. Mathematical algorithm description.

Mean value of output voltage depends explicitly by input voltage and by duty cycle of another voltage depending by input current, (1).

$$V_m = \frac{1}{T}\int_0^T u(t)dt = \frac{1}{T}\sum_{k=0}^{n}\int_{t_k}^{t_k+D} u(t)dt = \frac{1}{T}\sum_{k=0}^{n} u_k\cdot\left(t_k+D-t_k\right) = \frac{D}{T}\sum_{k=0}^{n}u_k \qquad (1)$$

The voltage to pulse width modulation (*U/PWM*) conversion block can be based on integrated analog circuits, which can be analogous, not including requirement of multiplication operations.

III. Simulation Results

The principle of functioning is based on the integration of impulses, directly proportional to the actual value of one of the voltages. This method can be implemented both analogue by analogue elements as well numerical circuits, processing the signals directly using the computer, [3], [4], [5], [6], [7]. Here was analyzed the analog circumstances.

Principally, the operating diagram consists in few blocks: current and voltage sensors, voltage to pulse width modulation converter, electronic switch and an integrator.

Simulations were made in Scilab software. For example, to reveal successive transformation of signal, the source of PWM signal generation and operation between PWM signal and voltage are described below.

```
// normalizing RMS of current => PWM percent
fu=((40*Ai*100)/40000)*100;  // current range 0..10 A
// PWM signal construction
fc=squarewave(t,fu);
//  operation between PWM command and voltage
sout=zeros(1,length(t));
for j=1:length(t)
   sout(j)=abs(u(j))*fc(j);
end;
```

Simulation results, according with presented details of signal transformation, are on 3 (A) and 7 (A) load, Fig. 3, Fig. 4. Signals represented are in orders: current, PWM signal with duty cycle depending by current, voltage, absolute voltage chopped by PWM signal and the output signal (apparent power) from integration of previously signal.

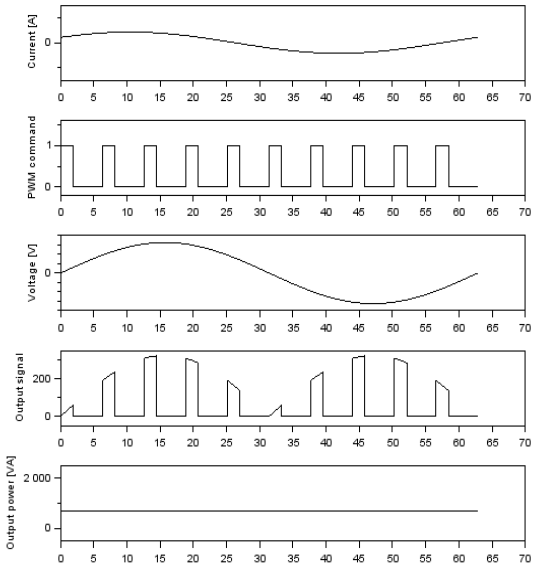

Fig. 3. Scilab simulation, 3(A) load.

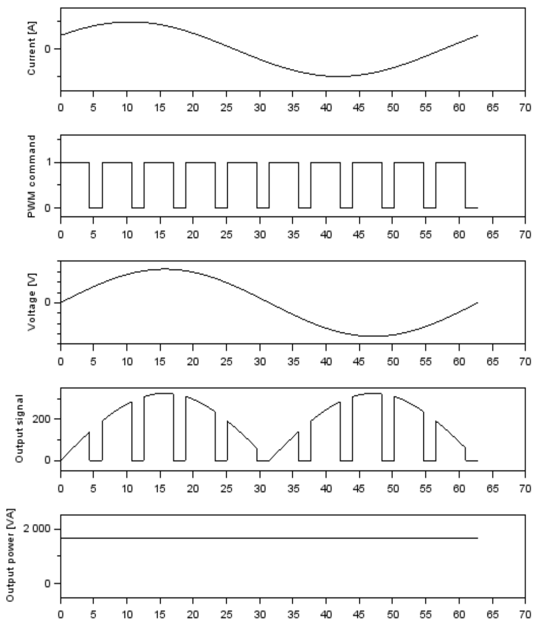

Fig. 4. Scilab simulation, 7(A) load.

The method does not explicitly take into account the phase present in the circuit, performing the equivalent multiplication operation. Even in simulation, the results not depend by phase between voltage and current. For inductive or capacitive circuits, the measured power is the apparent power (V·A), $S = U \cdot I$.

In DC circuits, the result has more clarified, representing the power transferred in/from circuit. In this case some elements are not mandatory, like absolute value of voltage or root mean square value of current. Designing the circuit is more simplified.

The output characteristic has a good linearity, Fig. 5. Load range was considered from 0 to 10 A. Non linearity error was less than 0.3 % in the domain taken into consideration, Fig. 6.

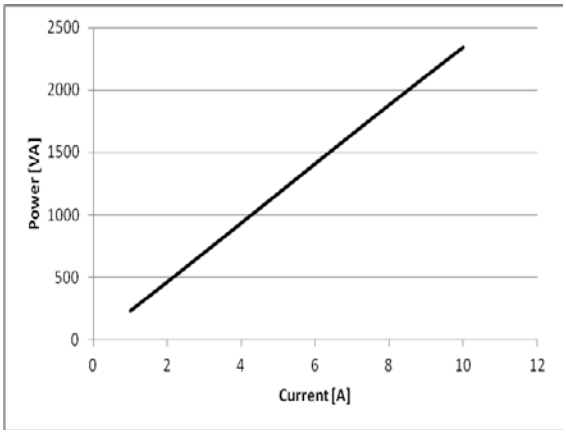

Fig. 5. Output characteristic of Scilab simulation.

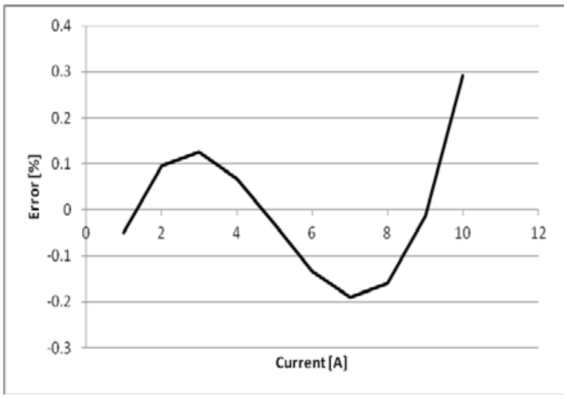

Fig. 6. Scilab simulation error.

For averaging pulses was used trapeze integration method [8], [9], [10]. Also, the influence of sampling frequency over computed results was analyzed (Table I). If sampling rule is met, errors are appropriate.

Representing errors of sampling seems to be a systematic error which can be easily reduced, Fig. 7. Simulations have different loads, 3, 5 and 7 (A).

For presented principle of operation was developed analog schematic using trivial components like operational amplifier and transistors. Schematics and simulation was made in LTSpice IV. Entire schematics include pulse width modulation (PWM) generator, absolute value circuit and output circuit, Fig. 8, [11].

TABLE I.
APPARENT POWER VS. SAMPLING FREQUENCY

Sampling frequency (Hz)	Load		
	3 (A)	5 (A)	7 (A)
100	684.20	1146.09	1617.09
500	683.65	1149.95	1616.48
3500	683.59	1150.04	1616.49
5500	683.60	1150.05	1616.50
7500	683.61	1150.06	1616.52

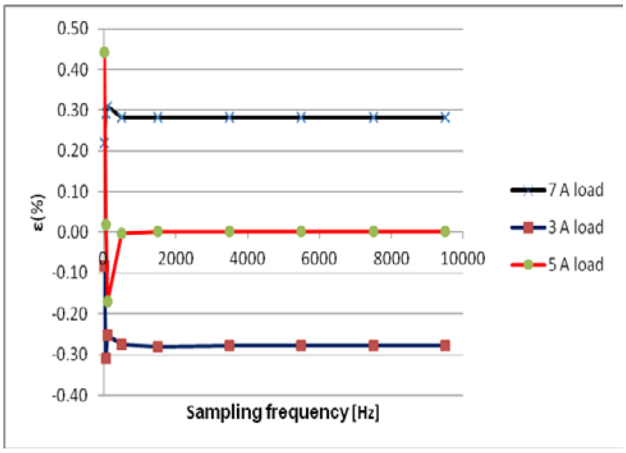

Fig. 7. Sampling error evaluation.

The PWM circuit has notorious LM555 integrated circuit as essential element. Here is in classical design to generate a duty cycle from 50% to 100%. To extend to full range (0% to 100%) should use other schematic. At this time is only to demonstrate the principle of functionality and have to avoid complex design of schematics. For the input signals, two voltage sources V2 were used to obtain the appropriate signal for a voltage to be modulated in width, V4 for the modulation of the V2 signal, processed in advance with an absolute value circuit. The voltage given by V3 is proportional to the value of the current absorbed by the load. Thus, the pulses from the LM555 circuit output have the frequency of those given by the V2 source and are modulated in the width of the V4 source.

The integrated circuit 555 is in monostable connection triggered by the input impulses and modulated by the control signal.

The other operational amplifiers have the task to change waveform, and the low pass filter has a high enough response time to transform the pulses into continuous voltage.

The PWM circuit has as inputs the value of current and sampling signal (suggest twenty times than power line frequency). Charging timing capacitor, C1, of PWM section is through current mirror. Charging capacitor via a constant current source, consisting by current mirror by transistors Q1 and Q2, ensure much stability. Relation 2 is the calculation formula for the capacitor charging current. All components are commonly used in electronic equipments and don't request special technical specifications.

$$I_C = \frac{V_{CC} - V_{BE}}{R_4 + R_3} = \frac{12 - 0.7}{22000} = 0.51 \ [\mathrm{mA}] \qquad (2)$$

Central section of schematics contains voltage and absolute value of input voltage source. Classical schematic of signal detection is presented, not an accurate circuit but full-wave rectifier. The output circuit is, here, an inverted low-pass filter. Signal pass through low-pass filter via electronic switches, modulated by PWM signal. Electronic switches are a generic symbol and can be field effect transistor, metal-oxide-semiconductor transistors or transmission gates.

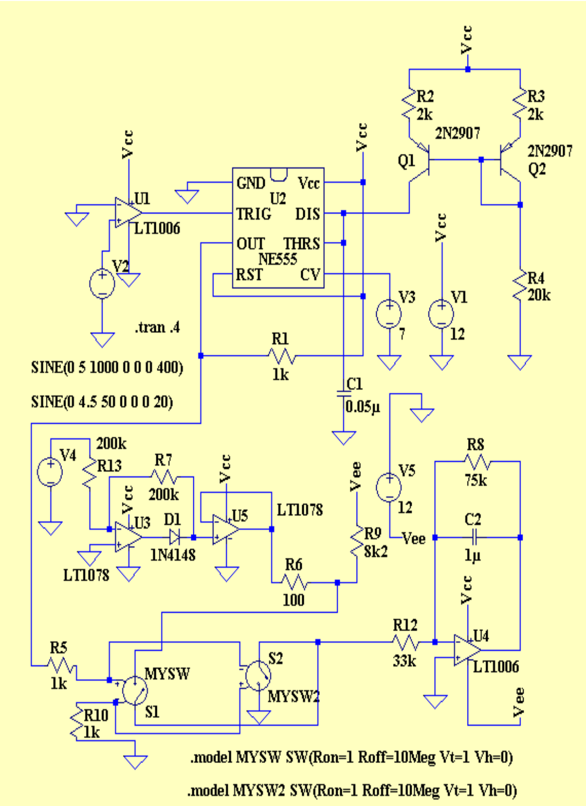

Fig. 8. Variant circuit diagram used for simulation, LTSpice.

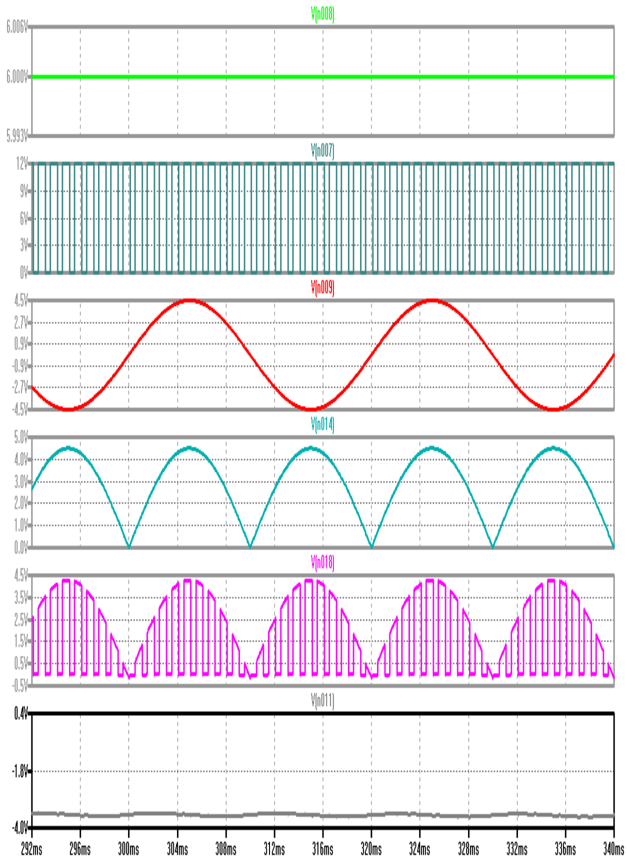

Fig. 9. An example of the signals obtained by simulation, LTSpice.

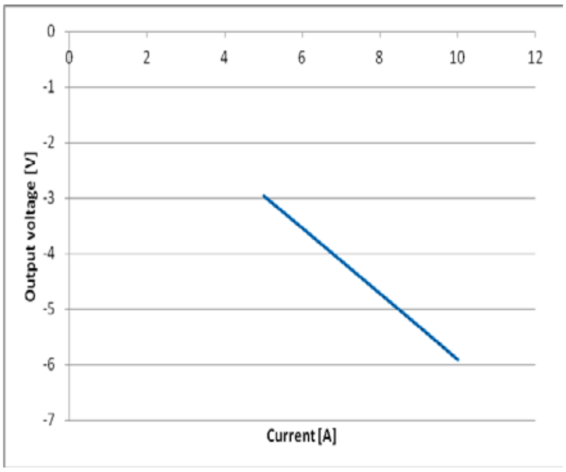

Fig. 10. Output characteristic of LTSpice simulation.

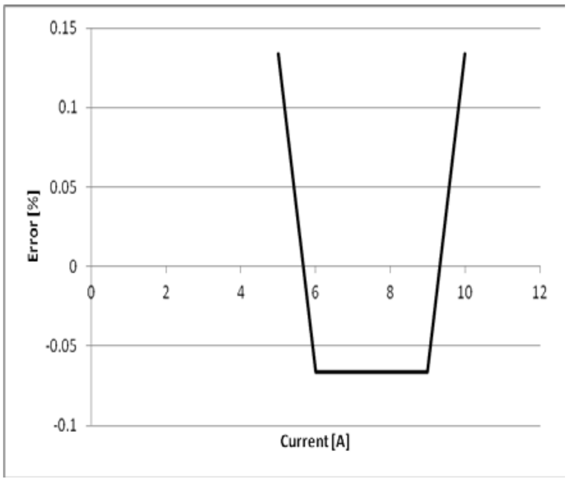

Fig. 11. Non-linearity error.

Signals considered have successive transformations, according with presented method, from input current (top) to output voltage (bottom), Fig. 9. All signals can be scaled, by adjusting schematics components value, to reveal a final value, similar to multiplication between current and voltage. Following the simulations, a linear characteristic, current dependent power has been obtained, Fig 10. It is noted that the data was obtained by positioning the cursor on the waveform. Thus, there are possible implications for reading accuracy, depending on the cursor resolution. On considered interval of input current, 5-10 (A), linearity is affected by not as much of 0.15%, mainly from PWM signal generator, Fig. 11.

The working manner of the other component blocks assures minimal influence on the whole process. After the two different simulations, the results obtained theoretically confirm the principle of the method.

IV. Experimental Results

Because of good results of method simulation, was developed an experimental device, Fig. 12. In order to obtain and experimental results, an implementation of the proposed scheme was achieved through a breadboard.

As operational amplifier, well known 741 types, were used.

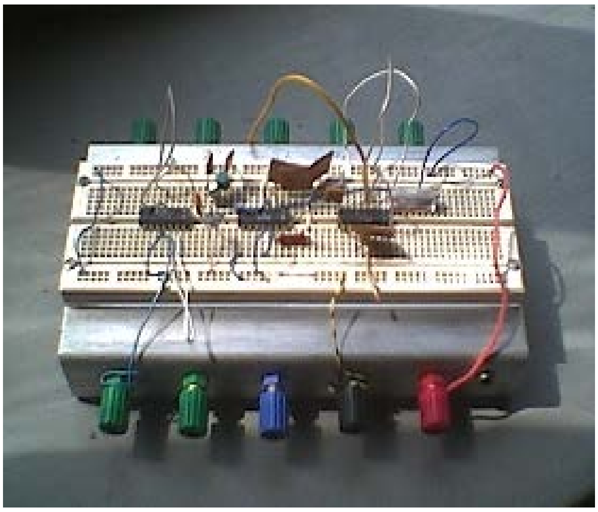

Fig. 12. Aspect of experimental device.

Proportionality of pulse width modulation generator, made by 555 sort integrated circuit, is near simulation results, Fig. 13. Along measured results, the graphical representation was approximated by linear interpolation. These both representations ensure possibilities to compute signal error, Fig. 14. Experimental error was less than 3% on entire considered input range. Error is highest in the middle of the interval and reduces towards the ends of the range.

To avoid another conversion block from current to voltage was used directly a power supply to replace current to voltage conversion block.

Practical realization does not require special requirements, but if use high quality components and a good power supply it is possible to achieve better experimental results.

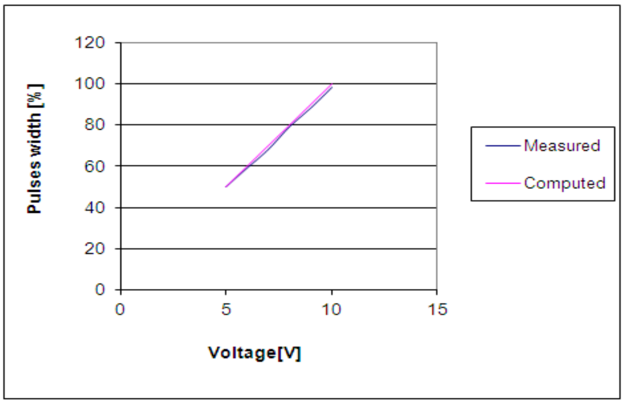

Fig. 13. Output characteristic (PWM) of experimental device.

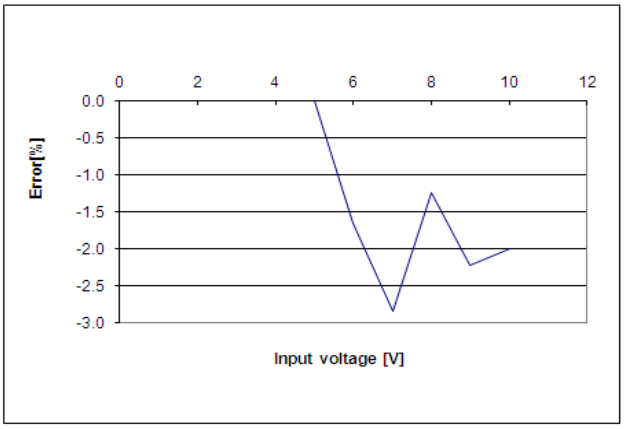

Fig. 14. Error of experimental device.

V. Conclusions

The requirements of electrical energy parameters are dictated by various domestic and industrial devices like appliances, equipment, and machinery. Electrical energy per time unit represents electrical power transferred. Knowing the electrical power with accuracy is very important because of the economic aspects involved.

The analyzed mark-space method is an alternative to classical methods which use multiplication operations.

For the analysis and design of the scheme, two programming environments were used, (Scilab from Scilab Enterprises and LTSpice from Linear Technologies, each one with its own specificity: the former validates the mathematical principle of method and the later validates the schematics).

It is known that the voltage waveform is very close to the sinusoidal shape but the current enclose harmonics due to the switching phenomena. Due to the fact that the pulsed voltages are filtered the value obtained does not depend, theoretically, on its waveform shape. The method is suitable for measuring deformed power, not being influenced by the waveform. This can be a future research development of mark-space method.

Additionally, the phase between voltage and current does not affect the result. Knowledge of phase shift is required when there is an inductive or capacitive load and it is necessary to know the active and/or reactive power consumed by it.

The topic addressed, is to follow a different approach to measuring power. Practically, the multiplication between a voltage and another voltage, proportional to the measured current value, was changed to filtering/integration of PWM signal.

PWM signal was synthesis from PWM pulses (duty cycle proportional with root mean square of current) and absolute value of voltage. Based on the mathematical formula of the principle, the simulation of operation reveal anticipated the results.

The linearity of output characteristic is remarkable, with less than 0.15% error, on considered inputs range. From a practical point of view, there are no problems specific to analog multiplication cells related to component parameters. Controlling the proportionality of the input and output signals of each block ensures good results.

It is possible to approach the practical realization of the method by changing the use the adjusting elements schematic, in order to obtain an output value as close as possible to the expected one and establishing range limits of the linear characteristic.

The input of the transducers that transform voltage and current levels into electronics-friendly values has not been analyzed in this paper, but it is important for practical results. It is preferable to use transducers with a high degree of integration, to ensure lower errors and lower power consumption. An example can be LEM transducer type with a good linearity and galvanic isolation but with disadvantages from the costs point of view.

For future development, the functionality of analyzed circuit can be extended by implementation in conjunction with virtual instrumentation.

In conclusion, some supplementary functions such as real time representation, or energy meter, give more flexibility and versatility to the method implementation.

ACKNOWLEDGMENT

This work was developed in laboratory of University Politehnica of Bucharest, Faculty of Electrical Engineering, Department of Measurement, Electrical Apparatus and Power Converters.

Thanks to Viviana Vladutescu Associate Professor of Electrical Engineering, Department of Electrical & Telecommunications Engineering Technology, New York City College of Technology, of the City University of New York, for logistic support.

Source of research funding in this article: Personal funding.

Received on May 16, 2017

Editorial Approval on November 29, 2017

REFERENCES

[1] Schneider Electric, “Electrical installation guide. According to IEC International Standards”, http://theguide.schneider-electric.com, Les Deux Ponts Printing, 2007, France.

[2] C. Ionescu Golovanov, *Electrical Measurements in Power Systems*, Romanian Academy Printing, AGIR Printing, 2009, Bucharest.

[3] E. Danila, D.D. Lucache, G. Livint, “Models and modeling the supercapacitors for a defined application”, Annals of the University of Craiova, No. 35, 2011; ISSN 1842-4805, pg.200-205.

[4] C. Chavez, J. A. Houdek, “Dynamic harmonic mitigation and power factor correction”, IEE. Electrical Power Quality, 9–11 Oct. 2007.

[5] S. L. Campbell, J.-P. Chancelier, R. Nikoukhah, *Modeling and Simulation in Scilab/Scicos*, Springer Edition, 2006.

[6] D. Floricau, M. Dumitrescu, I. Popa, S. Ivanov, “Basic topologies of direct PWM AC choppers”, Annals of the University of Craiova, Electrical Engineering Series, no. 30, 2006.

[7] I. L. Alboteanu, S. Ivanov, G. Manolea, “Modelling and simulation of a stand-alone photovoltaic system”, 8th WSEAS International Conference on POWER SYSTEMS (PS 2008), Santander, Cantabria, Spain, September 23-25, 2008.

[8] S. Ivanov, A. Campeanu, A. Bitoleanu, M. Popescu, “MATLAB-SIMULINK® library for AC drives simulation”, International Conference on Simulation (1998), 1998 p. 195 – 200, York, UK.

[9] P. Mathieu, P. Roux, *Scilab : I. Fundamentals*, Scilab Enterprises, 2016.

[10] C. Bunks, J. Chancelier, F. Delebecque, C. Gomez, M. Goursat, R. Nikoukhah, and S. Steer, *Engineering and Scientific Computing with Scilab*, Birkhauser, 1999.

[11] J. H. Mikkelsen, *LTspice – An Introduction*, *Technical report*, Institute of Electronic Systems, Aalborg University, Aalborg, 2005.