NUMERICAL MODEL FOR THE REACTIVE POWER COMPENSATOR BASED ON SWITCHING CAPACITOR

D. OLARU[1]

In etapa actuală, criteriile de apreciere a calităţii distribuţiei energiei electrice au devenit mai stricte. Un aspect important este problema îmbunătăţirii metodelor de compensare a consumului de putere reactivă. Comutaţia statică a capacităţii de compensare oferă o soluţie acceptabilă ca performanţe şi costuri. Dimensionarea şi comanda unei scheme de acest tip necesită utilizarea unui model numeric care să permită simularea funcţionării în diferite condiţii. Lucrarea prezintă o procedură de lucru, reprezentabilă în limbajul MathCad. Aceasta oferă corespondenţa directă cu ecuaţiile de bază, având un caracter mai general. Astfel există posibilitatea unei abordării riguroase şi a alegerii ipotezelor de lucru. Rezultatele numerice prezentate atestă validitatea raţionamentului şi aplicabilităţii sale.

In last years, the evaluation criteria for the power delivery quality became more difficult to be satisfied. An important aspect is the improvement of the methods used for the reactive power compensation. The static switching of the compensating capacitor gives a good solution with reasonable expenses and performances. The design and the control of such configuration request a numerical model which enable simulation of the operation for different given conditions. The paper presents a working method, which enables a MathCAD language representation. This gives a direct link to the main equations, having a more general character. Thus, a rigorous approach and different hypothesis may be done. The numerical results yield the validity of the procedure and its applicability.

Keywords: reactive power compensation, switched capacitor, numerical method.

1. Introduction

The reactive power compensation problem for a power delivery network is an old one. But, in the last years, power electronic equipment gives new possibilities for the implementation solutions. A global solution is achieved when a UPFC (unified power flow controller) equipment is used. But, this solution is still expensive and difficult to realize, in present. A simpler and cheaper solution use the switching capacitor scheme (Fig.1).

[1] Lecturer, Ph.D., Dept. of Measures, Switchgear and Static Converters, University “Politehnica” of Bucharest, Romania, e-mail: danolaru@apel.apar.pub.ro

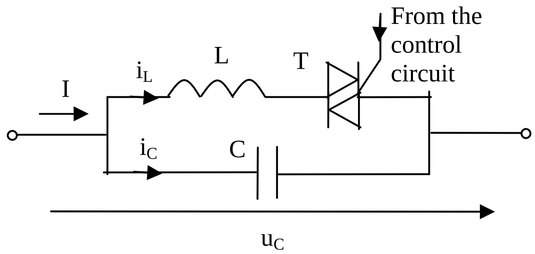

Fig.1. The basic electric schema.

Though there are very few components implied, the optimization of such structure is not easy to realize, because its non ideal working mode. The UFPC equipment has large possibilities to control the shape of the current wave. Because its simplicity and the presence of triac device, the switching capacitor scheme operation is based on a free evolution of parameters, after the conduction state is achieved. This leads to a more disturbed shape of the desired current wave. However, because the intrinsec integration proprieties of the delivery network, which act as a low pass filter, this equipment is really useful. Nevertheless its optimization isn't very easy. An experimental study of operation is expensive and difficult. A good solution is a numerical simulation, by computer programs, as Simulink-MATLAB or SPICE, for example. Each of these approaches has some disadvantages. The Simulink-MATLAB program is a signal flow simulator. The switching working mode scheme creates difficult implementation problems, because the energy accumulated in the reactive devices, which must be taken under consideration when switching maneuvers are done. The SPICE program is an electric or electronic circuit simulator. Thus, the input circuit scheme must contain only electric or electronic devices. Every else operating necessities must be simulated by the use of such devices. So, some processes like the command procedures, which in the reality are digital implemented, make serious difficulties.

For both the programs, as for the majority of technical oriented simulation programs, the main problem is the error controlling problem and the modeling of the nonlinear devices. The most representative, for this point of view, is the SPICE program. This uses a first order approximation procedure for the differential equation involved by the reactive devices. When the inductances are present, accuracy or convergence problems can arise. In any case, the intervention of the user in order to improve the quality of the simulation is impossible.

For such reasons, the utilization of a mathematical oriented program is for many points of view more benefic. Such a program is the MathCAD software. By this, the recursive form of the differential equations may be edit in a direct manner. By this way, the user can control the entire mathematical model. Thus the extensions and the improvements are easy to be made. Moreover, the clear and the concise edited form of the equations are benefic when error or debugging problems arise.

The paper presents a MathCAD implemented mathematical model, proving good results for the switching capacitor compensator for different parameters values.

2. The physical approach

The presented scheme must achieve a variable controlled capacitive compensation. In the steady state (the triac device is in the open state) the circuit acts as a pure capacitor. Its value must insure the maximal compensation capability for the associate circuit. When the triac is commanded for the conduction state, the inductor current i_L is added to the capacitor current. Thus the capacitive character may be reduced. The choice of the moments when the gate pulses are active determines the weight of the inductive current and the shape of the total current i. The circuit operation is fixed by the inductor and capacitor values, on one hand, and by the synchronism of the triac gate control, on the other hand.

The following analysis of the circuit is focused on the current-voltage dependence, yielding the behavior of the reactive compensator. In Fig.2 it is showed the configuration principle for the reactive compensator, integrated in a scheme, containing the network Thevenin model, the line reactive model and the resistive-inductive load model. It is clear that the compensator acts as a capacitive current receiver. The compensator and the load circuit operate near the resonance regime, having a null reactive behavior. Because the resonance phenomena, this circuit, which is equivalent with a parallel RLC one, has a null reactive current consumption and all the electrical parameters are well defined.

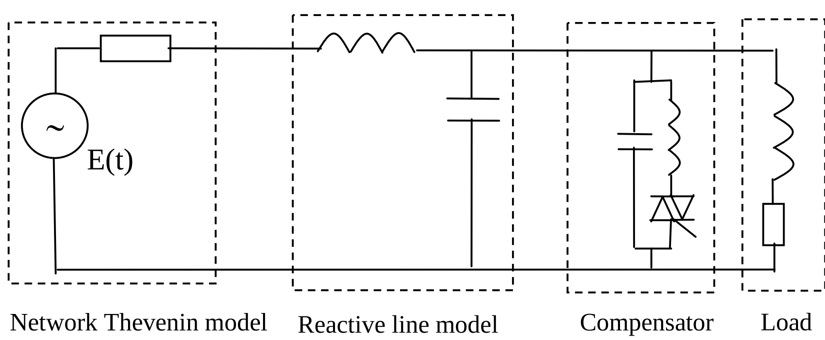

Fig.2. Network integration of the reactive compensator.

Because the transient operation regime, the circuit must be modeled by a nonlinear differential system of equations. In order to derive a first order ordinary differential system, the main variables must be chosen as the inductor current i_L and the capacitor voltage u_C (for the circuit showed in Fig.1). Thus, the voltage across circuit become an independent variable, and must be not imposed as given data. So, the external supply of the circuit must be a current source, driving the current *I(t)* through the circuit. In Fig.3 it is shown a simplified model used for the analysis of the reactive circuit operation.

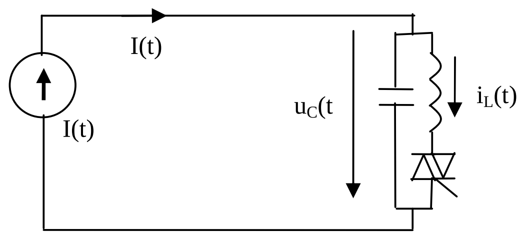

Fig.3. The physical model of the reactive compensator operation.

This is a LC parallel configuration supplied by an ideal current source. When the circuit is near the resonance, the current may reach high values on the compensator circuit loop. Thus, because the ideal current supply, the voltage on the capacitive impedance, (and on the compensator circuit, too) may reach also, high values. These considerations show that, during near resonance operation, we can expect increasing values for the voltage variation.

3. The mathematical model

For simplicity, at a first step, the triac device presence will be neglected. From the characteristic equations of the two reactive elements, where the Khirchhoff law was used for the inductor current substitution, the following basic equations can be written.

$$L\frac{d(I-i_C)}{dt}=u_C \qquad C\frac{du_C}{dt}=i_C \tag{1}$$

If the first order finite differences are used to approximate the differential, a discrete form of the system may be written:

$$L\frac{\Delta I-\Delta i_C}{\Delta t}=u_C \qquad C\frac{\Delta u_C}{\Delta t}=i_C \tag{2}$$

Taken under consideration two successive time steps t_k and t_{k+1}, an Euler type, discrete scheme may be obtained. The following designations are used:

$h = t_{k+1} - t_k$, the time step length;
$\Delta I(k)$ – the finite difference corresponding to the source current, for two time steps in the past, relatively to t_k moment: $\Delta I(k)=I(k)-I(k-1)$;
$i_C(k)$, $i_C(k+1)$, $u_C(k)$, $u_C(k+1)$ – the current and voltage discrete values.

$$L\,\frac{\Delta I(k)-\left(i_C(k+1)-i_C(k)\right)}{h}=u_C(k) \tag{3}$$

$$C\,\frac{u_C(k+1)-u_C(k)}{h}=i_C(k) \tag{4}$$

Separating the values corresponding to the following time moment, a computing explicit scheme is derived:

$$i_C(k+1):=\Delta I(k)+i_C(k)-\frac{h}{L}u_C(k) \tag{5}$$

$$u_C(k+1):=u_C(k)+\frac{h}{C}i_C(k) \tag{6}$$

In order to obtain a single vectorial recursive equation, the two unknown discrete functions $i_C(k)$ and $u_C(k)$ must be represented as the components of a single vectorial discrete function, denoted as $(v^{<k>})_0$ and $(v^{<k>})_1$, using the MathCAD style. Thus the first order differential system may be written as:

$$v^{<k+1>}:=\begin{bmatrix}\Delta I(k)+\left(v^{<k>}\right)_0-\frac{h}{L}\left(v^{<k>}\right)_1\\ \left(v^{<k>}\right)_1+\frac{h}{C}\left(v^{<k>}\right)_0\end{bmatrix} \tag{7}$$

Now the triac device will be taken under consideration, as a nonlinear behavior of the circuit. In this approach, the component parameters have constant values, but the circuit configuration will change, following the triac characteristic. Thus, the inductor current will have a non null value in two cases: a current command will be present at the gate or, the preceding triac current has a value above a given maintaining level, denoted by i_ε. To state the presence of the triac command current, a time depending, switch function is used, denoted by $K(t_k)$, corresponding to the t_k time moment.

In the MathCAD language notation, a conditional function may be used, having the syntax if (condition, expression A, expression B). This function will return the expression A value if the “condition” is satisfied and expression B value for the other case. Thus, the evolution of the circuit parameters will be given by a single recursive, vectorial, equation:

$$v^{<k+1>} := \begin{bmatrix} if\left[K(t_k)=1, \Delta I(k)+\left(v^{<k>}\right)_0-\frac{h}{L}\left(v^{<k>}\right)_1, if\left[\left(v^{<k>}\right)_0-I(t_k)\le i_\varepsilon, I(t_{k+1}), \Delta I(k)+\left(v^{<k>}\right)_0-\frac{h}{L}\left(v^{<k>}\right)_1\right]\right] \\ \left(v^{<k>}\right)_1+\frac{h}{C}\left(v^{<k>}\right)_0 \end{bmatrix} \tag{8}$$

4. The numerical simulation

The numerical simulation purpose is to study the reactive compensator behavior, for different kind of commands. The circuit operation is full determined by the triac gate phase dependent control. This is defined by the command delay, referred to the source current variation.

In the mathematical model, the switch function $K(t_k)$ is used for the control of this parameter. In order to obtain a similar "*sign(.)*" function, the predefined MathCAD "*Φ(.)*" function is used, where the point represents the argument. This returns "1" value, for a positive argument and "0", for the other cases. In order to obtain a phase sensitive function, we consider for the argument of the *Φ(.)* function the difference between a given constant value and a retarded *sin(.)* function. Thus, the phase delay of the retarded *sin(.)* will define the phase of the switch function $K(t_k)$. This represents a pulse function. The given constant value determines the duty cycle value of the pulse shape function *Φ(.)*.

$$K(t) := \Phi(|\sin(\omega t+\alpha)| - 0.95) \tag{9}$$

If we use this form, the evolution of *K(t)* function, versus the sinusoidal source current is showed in Fig.4:

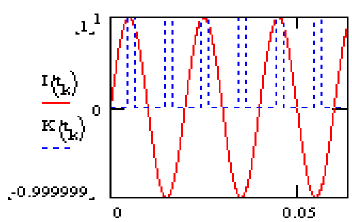

Fig.4. The pulse control function evolution (pointed).

The numerical simulation was done, in order to study the behavior of the circuit as a variable capacitive reactive compensator. In the real MathCAD language representation, there are used some different notations for the working variables, having the following correspondence:

$(u^{<k>})_1$ represents the voltage across the compensator circuit;

$I(t_k)$ represent the in current source instantaneous values;
$I(t_k)$- $(u^{<k>})_0$ represent the inductance current instantaneous values.

Two cases were taken under consideration corresponding for two values of α parameter, corresponding to a non-reactive behavior (Fig.5) and a pure capacitive behavior (Fig.6). The diagrams show that the compensator circuit can act as a non reactive or a capacitive impedance. This behavior can be observed after a transient regime, which takes place near the initial moment.

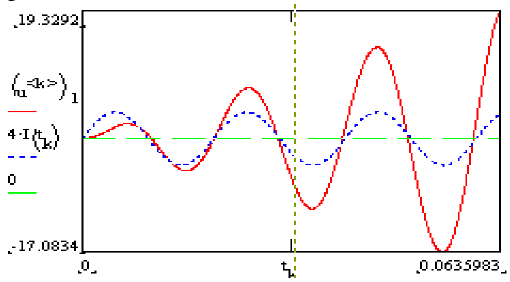

Fig.5. The phase delay characteristic for α=0 (resistive behavior).

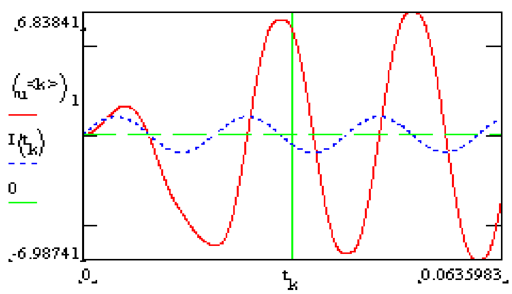

Fig.6. The phase delay characteristic for α=π/2 (resistive behavior).

In order to decide the good operation of the circuit, there are plotted the inductance current variation and the source current variation, as time functions (Fig.7). The triac device active window command was represented, too. It can be observed that the beginning variation front of this signal is synchronous with the inductance current occurrence.

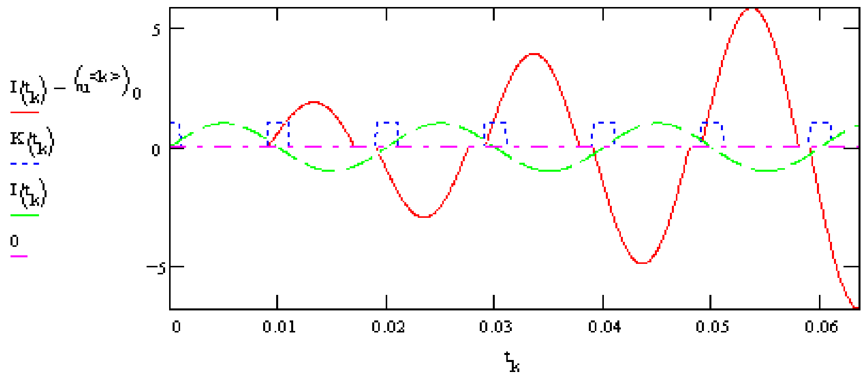

Fig.7. The inductance current, versus the command signal and supply current variation.

It is important to specify that all the voltage variations plotted after the numerical simulations have a non-stationary behavior, characterized by increasing amplitude. This is due to the resonance phenomenon, described above.

5. Conclusions

The paper is focused on the reactive power compensation problem. The presented solution is simple to be implemented and has the advantage of a direct tuning possibility of the capacitive effect. But, the control of the circuit must be stable and able to give the requested effect. In order to define a correct apriori command a prealable simulation must be done. The MathCAD program gives good results and has the most general representation, from the mathematical point of view. Nonlinear device behavior can be easy included in the differential equation model.

R E F E R E N C E S

[1] *B.K.Perkins, M.R.Iravani* – Dynamic modeling of a TCSC with application to SSR analysis. IEEE Power Eng.Soc.Winter Meeting. Paper no.PE 477-PWRS-0-01-1997,1997.

[2] *P.Mattavelli, G.C.Verghese, A.M.Stankovic* – Phasor dynamics of thyristor-controlled series capacitor systems. IEEE Trans.on Power Systems 12(3) Aug.1997.

[3] *H.A.Othman, L.Angquist* – Analytical modeling of tyristor-controlled series capacitors for SSR studies. IEEE Trans.on Power Systems 11(1) February 1996.