

Michał LEWANDOWSKI

Instytut Elektrotechniki Teoretycznej i Przemysłowej. Zakład Elektrotechniki i Informatyki

## APPLICATION OF MATLAB SIMULATION PACKAGE FOR TRANSIENT ANALYSIS IN POWER SYSTEMS

**Summary.** The paper describes the results of a transient simulation analysis in power engineering. All tests were performed using *Matlab/Simulink* package with *SimPowerSystems* library, a dedicated tool for power circuits simulation. The paper presents the testing methodology, evaluation criteria, an example of a circuit analysis, proposal of an optimisation of the simulation procedure and a critical analysis of the software.

## ZASTOSOWANIE PAKIETU MATLAB DO ANALIZY STANÓW NIEUSTALONYCH W OBWODACH ELEKTROENERGETYCZNYCH

**Streszczenie.** W artykule opisano wyniki badań symulacyjnych dotyczących analizy stanów nieustalonych w układach elektroenergetycznych. Badania przeprowadzono z użyciem pakietu symulacyjnego *Matlab/Simulink* wraz ze specjalizowaną biblioteką *SimPowerSystems*. Przedstawiona została zarówno metodyka badań, kryteria oceny wyników symulacji, jak i propozycja strategii optymalizacji samego procesu symulacji oraz krytyczna analiza zastosowanego oprogramowania.

### 1. INTRODUCTION

The paper describes the results of an investigation and evaluation of *Matlab/Simulink* with *SimPowerSystems* blockset for dealing with transient simulation in power engineering. The paper is based upon the results and experience developed during the execution of the project entitled: "Numerical Simulation in Power Quality" [2], which was accomplished in year 2003 as a part of the European Union "Leonardo da Vinci" training programme. The purpose of this project was the investigation and evaluation of *Matlab/Simulink* with *SimPowerSystems* blockset as a tool for solving a typical industrial power quality problems, where numerical simulation is often needed. All of the simulated circuits were provided by the host organisation (*LABORELEC, Belgium, Power Quality Department*). *LABORELEC* was especially interested in potential simulation problems which may affect the simulation results or the simulation process itself. The software was analysed with special focus on: transient analysis results, inrush currents and voltages, computational speed, simulation accuracy and restrictions and ease of use. The project's final report contained the simulation results for each studied case, explanation of the observed phenomena, results of parameters recalculations to the form required by *Matlab* models, speed and accuracy discussions, revealed problems and proposals

of the solutions. The simulation results were also compared to the results obtained with the other software simulation package – ATP (Alternative Transients Program). However the ATP simulations were not an internal part of the mentioned project.

## 2. CIRCUIT FUNCTIONAL DIAGRAM

As a common base for all simulated circuits a part of a power substation was used. Devices common for all the simulations were shown in Fig.1. In each study case various devices were connected to the main bus.

The complete list of the provided study cases contains:

- Capacitor bank energizing with harmonic impedance calculation.
- Nonlinear transformer energizing (unloaded) with high harmonics calculation.
- Asynchronous motor starting with different loads.
- Reaction of a synchronous motor to a main bus fault.
- Reaction of an asynchronous motor to a voltage dip.
- Three phase diode bridge with resistive and resistive - inductive load.

Some of the listed cases were simulated in various configurations, with different initial conditions etc.

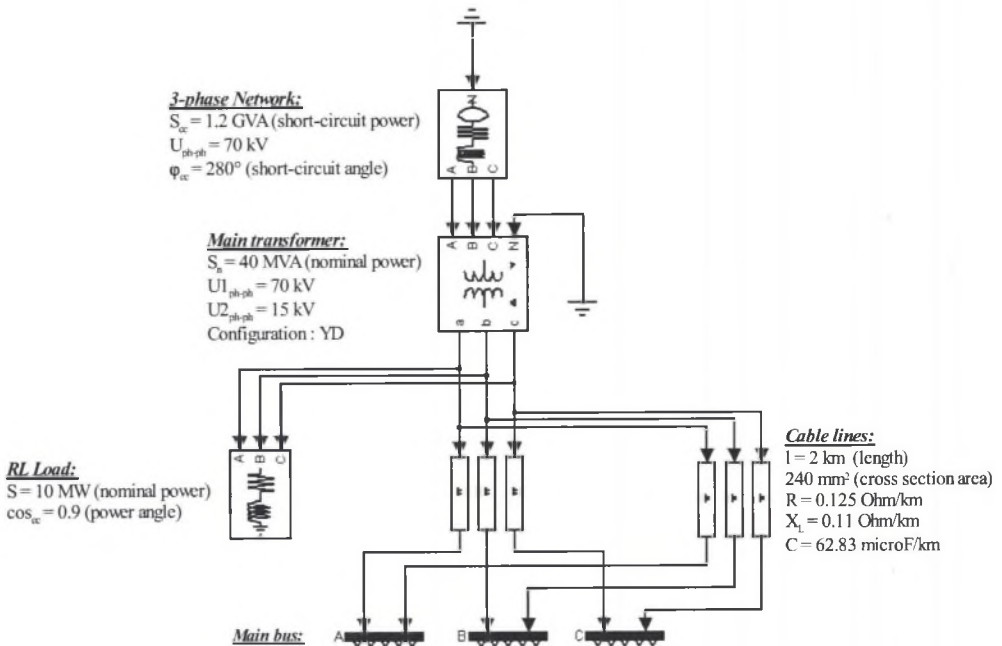


Fig. 1. Functional diagram of the circuit – common part  
 Rys.1. Schemat funkcjonalny badanego ukladu – część wspólna

## 3. TESTING ALGORITHM

Each selected study case were analysed using the algorithm shown in Fig. 2. First, the raw *Simulink* scheme of the selected case is created. In this case “raw” means that the scheme

contains only the common part from Fig.1 and the tested element (capacitor bank for example) without any additional *Simulink* virtual meters, interface blocks etc. In the next step all the parameters of the devices present in the raw scheme are set. Parameters recalculation is often necessary because *Simulink* models usually need different parameter sets than the one which are given. At this stage it is sometimes necessary to use a typical parameters from the publications like [1, 7]. It is desirable, when the provided parameter set is not complete enough to satisfy the conditions of a chosen *Simulink* model. When all the parameters are set properly, the scheme might be completed by adding the necessary virtual meters and additional testing blocks, required to measure the desired circuit variables during the simulation process.

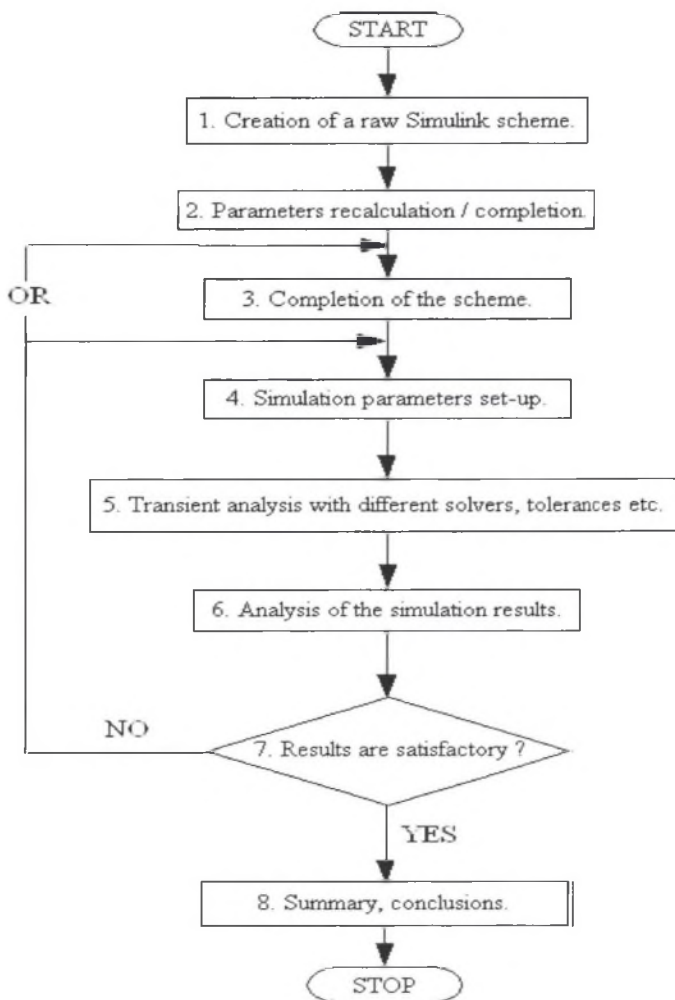


Fig. 2. Testing algorithm diagram

Rys. 2. Schemat blokowy algorytmu przeprowadzania badań

When the scheme is finished the simulation parameters should be set. At this stage one must decide which solver should be used (discrete, continuous, stiff or variable step), what tolerance should be chosen, how to set the initial conditions and how to set the simulation time. These decisions strongly depend on the analysed study case. As is described in [2, 3] the

mentioned decisions are critical for simulation accuracy and speed, so they should be taken very carefully. Information provided in description of the models might be very valuable for that. When the simulation parameters are set, one can perform the simulation. It is good to perform the simulation couple of times for different solvers and tolerances. It gives the opportunity to assess the accuracy and speed for different simulation parameters. For each analysed study case such a comparison was made to find the optimal parameters set [2]. In the next step the obtained simulation results are critically analysed to find the potential problems and assess the credibility. As a reference to the assessment, the data provided in the publications [4, 5, 6, 7] were used. When the simulation results are not satisfactory the simulation process is repeated with different parameters. Sometimes it might be even desirable to change the scheme to successfully accomplish the simulation [2]. When the simulation is finished successfully and the results are analysed, the summary and conclusions might be written.

The algorithm described above, unified for all the cases, gives the opportunity (after simulating all the study cases) to evaluate an optimal simulation strategy and easily extract the problems revealed in each case. Moreover, because similar procedure was performed for the same or similar cases using *ATP (Alternative Transients Program)* package, it was possible to compare *ATP* with *Matlab/Simulink/SimPowerSystems*.

#### 4. ANALYSIS EXAMPLE

This example presents a charging of a capacitor bank connected to the main bus. A complete (after stage 3 of the analysis algorithm) diagram of the circuit is presented in Fig. 3.

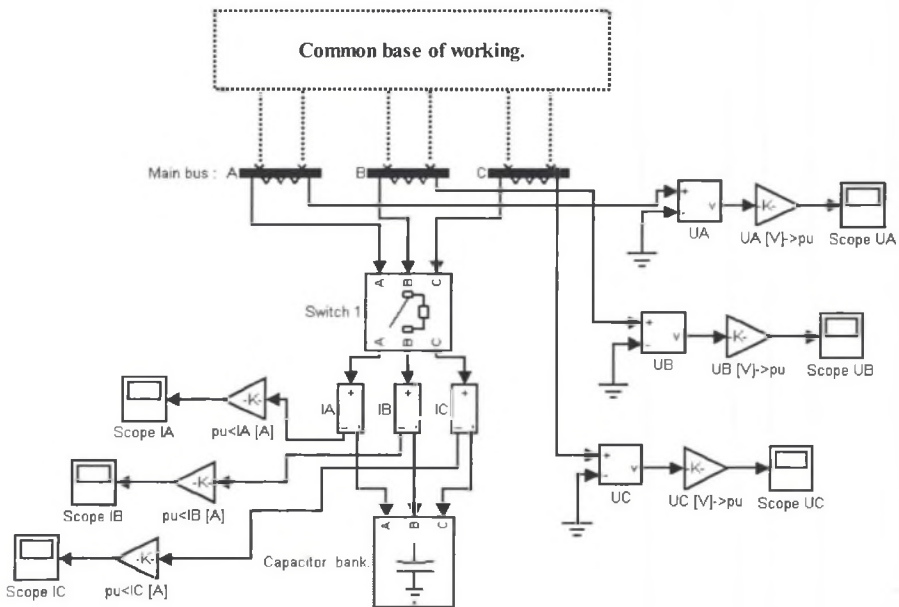


Fig. 3. Capacitor bank energizing – the scheme  
Rys. 3. Ładowanie baterii kondensatorów – schemat układu

The capacitor bank is connected to the main bus via 3-phase switch, which lets one to simulate the transient state of the system at any time point. During the transient simulation the measured system variables are: main bus voltages and capacitor bank currents. Parameters of

the devices are the same as in the Fig.1. A detailed parameters recalculation to the form needed by *Simulink* models can be found in [2]. The capacitor bank parameters are as follows:

$Q_c=5$  Mvar (reactive power),  
 $U_{ph-ph}=15$  kV (nominal phase to phase voltage).

When the devices parameters are set properly, the next step is the simulation parameters set-up. For this study case the parameters were set accordingly to table 1. Solver parameters which are not shown in the table were set to “auto”.

Table 1

Simulation parameters

Start time:	0 s
Stop time:	0.2 s
Solver type:	Variable-step ode23tb (stiff/TR-BDF2)
Relative tolerance:	1e-3, 1e-4, 1e-5, 1e-6 (speed of computation was measured for each of these values)

4.1. Transient simulation results

After setting up the parameters the simulation was performed. The switch-on time for the capacitor bank were set to 0.04 s. Main bus voltage waveforms are shown in Fig. 4. Values of all circuit variables are expressed in per-unit relative system with a nominal value as a base value.

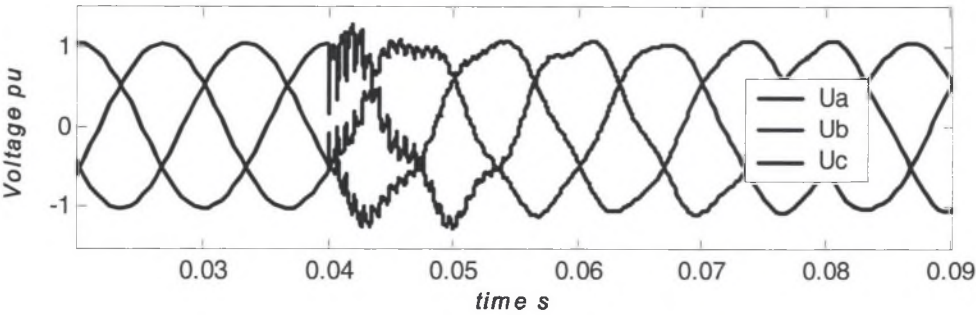
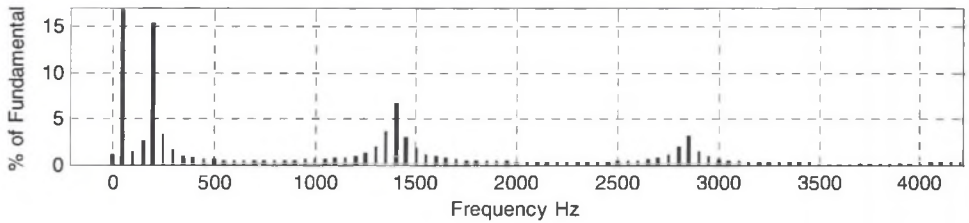


Fig. 4. Main bus voltages during the capacitor bank energizing  
Rys. 4. Napięcia na szynie głównej w trakcie ładowania baterii kondensatorów

The biggest peak value at switching-on time is the  $u_c(t)$  voltage (in Fig. 4 designated as  $U_C$ ), which can be observed in Fig. 4. Simulations made at the various switching times show, that the biggest overvoltage always appears at this voltage, which has its maximum value at switching-on time.

The voltage oscillations observed in the circuit after breaker switch-on are results of the  $C$  resonance of the capacitor bank and the  $LRC$  parameters of the main transformer, cable lines and the load. To identify the resonant frequencies FFT was applied to the  $u_c(t)$  voltage (Fig. 5).



Fig. 5. FFT of the bus voltage  $u_c(t)$ Rys. 5. Transformata Fouriera napięcia  $u_c(t)$ 

Using a Fourier analysis can be shown that the fundamental frequency is 50 Hz and the resonant frequencies of the system are present at approximately: 210 Hz, 1420 Hz, 2840 Hz. The resonant characteristic of the  $RLC$  circuit significantly affects not only the frequencies mentioned above, but also the bands located near these frequencies.

Currents of the capacitor bank were also simulated. These simulations revealed limitations in the model of the cable lines. After switching-on the capacitor bank, a very narrow spike with a large amplitude appeared (Fig. 6). The value of this spike is so high, that the current's oscillations are not visible on the graph. A zoom function was used to show these oscillations (Fig. 7).

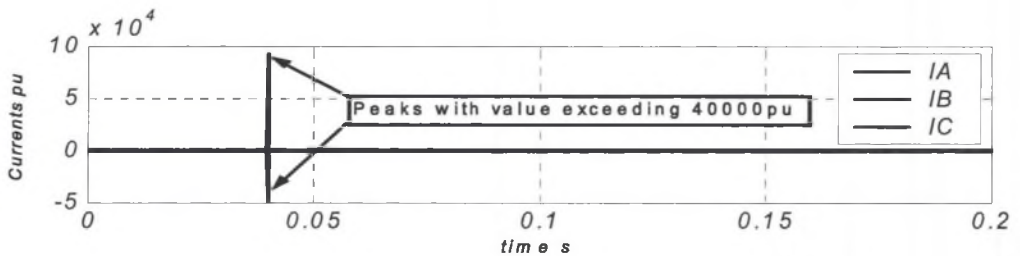


Fig. 6. Currents of the capacitor bank (short spike at 0.04 s)

Rys. 6. Prądy ładowania baterii kondensatorów (krótki impuls dla czasu 0.04 s)

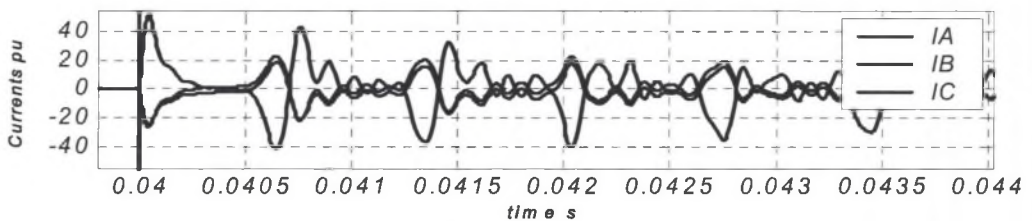


Fig. 7. Currents of the capacitor bank – zoomed

Rys. 7. Prądy baterii kondensatorów – powiększenie

The short spike with a very high value is caused by the “ $PI$  section line” model [2]. The “ $PI$  section line” model has a pure capacitance at the beginning and end of the line. The end connected to capacitor bank is shown in Fig. 8.

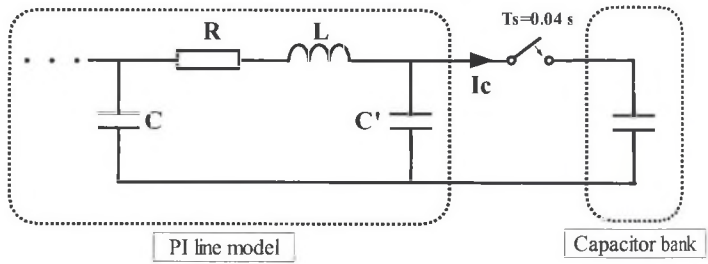


Fig. 8. PI section line model and the capacitor bank charging.  
Rys.8. Sekcyjny model linii typu PI a ładowanie baterii kondensatorów

At the switching-on time, the  $C'$  is already charged and the capacitor bank is uncharged. When the switch is closed, most of the energy stored in  $C'$  flows as current  $i_c(t)$  (in Fig. 8 designated as  $I_c$ ) to the capacitor bank without any restrictions, this causes the short spike on the currents plot. That spike is theoretically unlimited in value ( $i \propto \frac{du}{dt}$ ) but in the simulation it depends on the chosen integration method. The solution to this problem might be putting an RL branch at the beginning and end of the cable line model.

4.2. Computation Speed

In this case the computation speed was measured for different numbers of “PI” sections of the cable lines, with varying accuracy, to show how these parameters influences the speed. The variable step stiff solver “ode23tb” (solver based on a implicit Runge-Kutta formula) was used. Measured simulation time is shown in Fig.9.

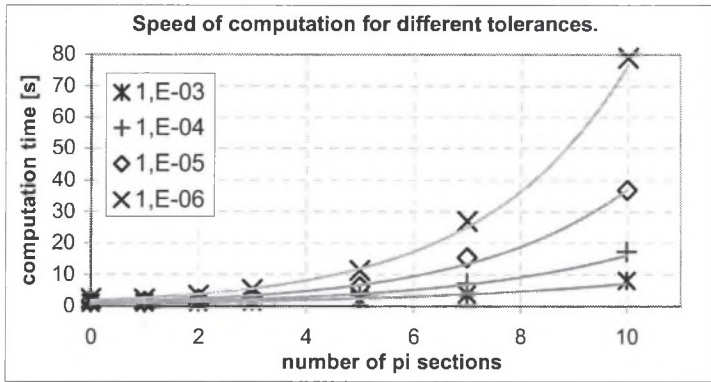


Fig. 9. Speed of computation for different accuracy and PI section line elements  
Rys. 9. Czas obliczeń dla różnej dokładności i różnej liczby sekcji typu PI

Fig. 9 [2] shows that for each tolerance considered, the computation time strongly depends on the number of “PI” sections.

### 4.3. Simulation Accuracy

To find out how the relative tolerance influences the accuracy of the simulation results, the simulation of one of the main bus voltages with different relative tolerance values was performed. Results of that simulation are shown in the Fig.10 [2].

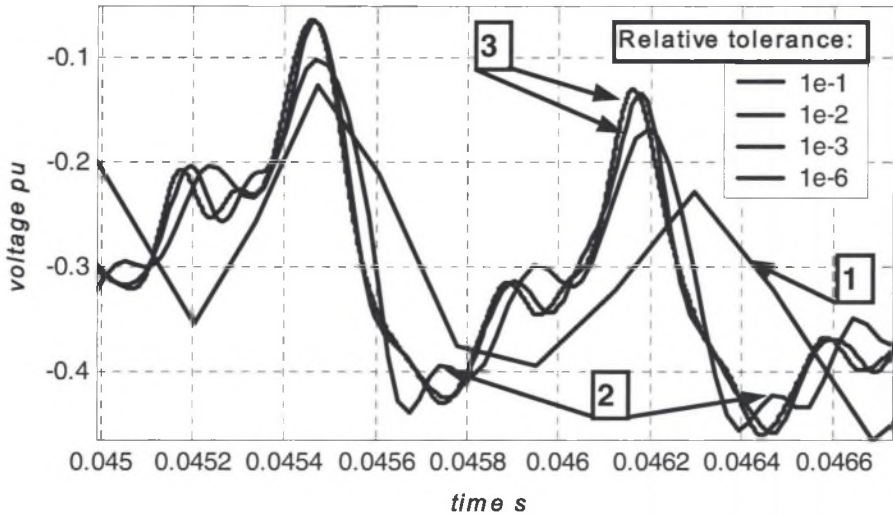


Fig. 10. Main bus voltage  $u_C(t)$  simulated with different solver tolerances (precisions)

Rys. 10. Napięcie szyny głównej  $u_C(t)$  dla symulacji z różną dokładnością całkowania

In Fig. 10 the following can be seen:

1. A tolerance of  $1e-1$  is too big, because some of the voltage oscillations are not visible. Additionally, the peak value is not computed correctly.
2. A tolerance of  $1e-2$  is also too big, because it indicates some voltage oscillations that are not present in the more precise results.
3. Starting from a tolerance value of  $1e-3$ , the results start to converge into a single curve.

The conclusion is that, for this particular circuit, the maximum relative tolerance value that can be used to produce valid results is:  $1e-3$ . It seems to be a good compromise between accuracy and speed of computation.

### 5. PROPOSAL OF OPTIMISATION STRATEGY

After simulating all the study cases listed in paragraph 2, a proposal of an optimisation strategy of the simulation process was drawn up [2]. This strategy is shown in Fig. 11. The optimisation algorithm shown in Fig.11 is not a strict and unique method and doesn't have to be true for each circuit. However, it can be useful as a reference point when one is performing simulations of a circuit similar to those which were presented in this paper.



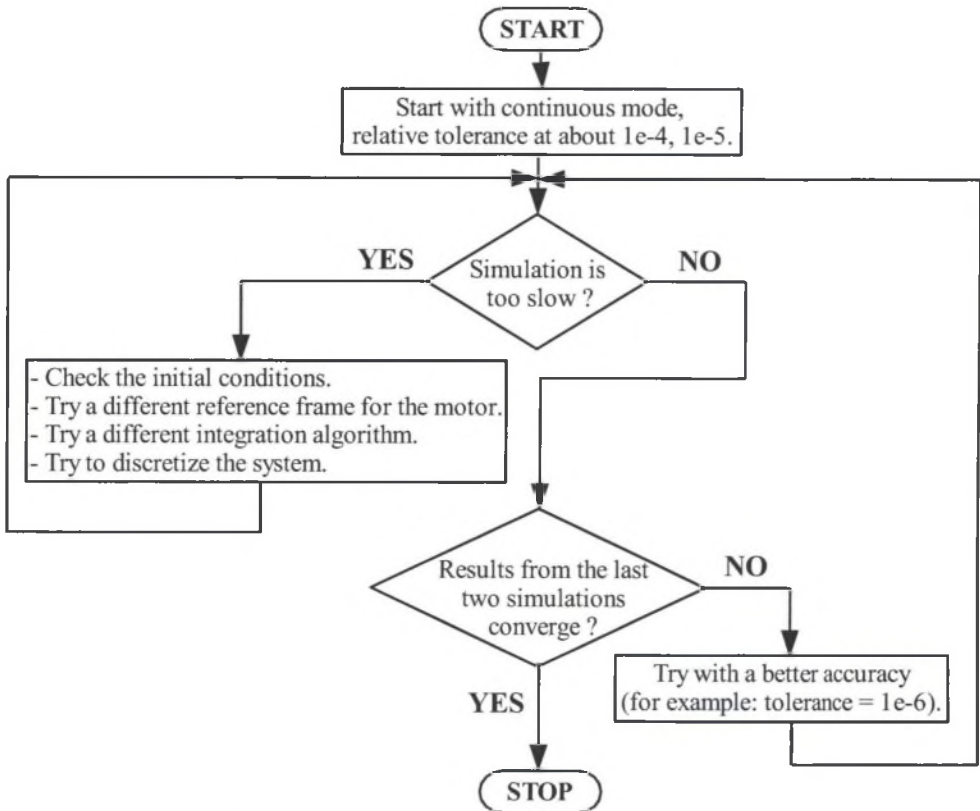


Fig. 11. Diagram of the optimisation strategy  
Rys. 11. Diagram strategii optymalizacji

The algorithm shown in Fig. 11 suggest to start the simulation with a relative tolerance of  $1e-4$  in continuous mode. When the simulation is too slow there are some guidelines which may help to solve the problem. The simulations are repeated until results from the last two simulations are converging.

## 6. CONCLUSIONS

*Matlab/Simulink/SimPowerSystems* environment is a useful tool for transient analysis in the power engineering domain. Its ease of use, number of options, well organized interface and openness, make it a powerful ally of an engineer and scientist.

Because of its generality, *Matlab/Simulink* is very flexible tool. The number of options, which can be controlled by the user is impressive. There is a choice between various integration algorithms, discrete or variable step solvers and many of the simulation parameters can be controlled manually or automatically. The subsystem creation option provides an easy way to build complex circuits by putting the sub-circuits into a *Subsystem* blocks. Another important thing is the easiness of data import and export. After performing the simulation the

data can be saved in *Matlab*'s internal file format or exported into a text file for further examination. Because the simulation results are stored in *Matlab*'s workspace, one can use the whole set of command line functions to analyse, transform or plot the data. *SimPowerSystems* includes a special set of blocks (interface blocks), which help to connect the standard *Simulink* library blocks to the *SimPowerSystems* circuit diagrams, which are greatly increasing flexibility of the *SimPowerSystems* toolbox.

It is also hard to overestimate the documentation provided with *SimPowerSystems*. Each model is well described so it is possible to solve simulation problems caused by the internal model structure (for example: problem with the described *PI* line model).

In contrast to the flexibility and ease of use, the speed of computation seems to be one of main weaknesses of *Matlab* (in comparison to *ATP*). Sometimes it is possible to speed up the simulation by changing the integration algorithm or discretizing the system, but it may take a negative effect on the accuracy of the results. On the other hand, using the described optimisation strategy, it is possible to reduce the simulation time to an acceptable level (minutes).

## REFERENCES

1. Chatelain J.: *Machines électriques* Georgi St. Saphorin, Lausanne 1983.
2. Lewandowski M.: *Numerical Simulation in Power Quality* Laborelec – Internal Project Report, Belgium 2003.
3. Mrozek B., Mrozek Z.: *MATLAB 5.x SIMULINK 2.x, Poradnik użytkownika* PLJ, Warszawa 1998.
4. Power Quality Working Group WG2.: *Guide to Quality of Electrical Supply for Industrial Installations Part 1: General Introduction to Electromagnetic Compatibility EMC: Types of Disturbances and Relevant Standards* International Union for Electroheat, Paris 1994.
5. Power Quality Working Group WG2.: *Guide to Quality of Electrical Supply for Industrial Installations Part 2: Voltage Dips and Short Interruptions*. International Union for Electroheat, Paris 1994.
6. Power Quality Working Group WG2.: *Guide to Quality of Electrical Supply for Industrial Installations Part 3: Harmonics*. International Union for Electroheat, Paris 1995.
7. Solignac G.: *Guide de l'ingénierie électrique* Georgi St. Saphorin, Lausanne 1999.

Recenzent: Dr hab. inż. Zbigniew Lubośny, prof. Pol. Gdańskiej

Wpłynęło do Redakcji dnia 13 kwietnia 2004 r.