

## PSPICE SIMULATION OF POWER ELECTRONICS CIRCUIT AND VECTOR CONTROL OF AC DRIVES

**Adrian Şchiop**

*University of Oradea,  
Armatei Române, 5, 410087, Oradea, România, aschiop@uoradea.ro*

**Abstract:** This paper shows how power electronics circuits, electric motors and drives, can be simulated with modern simulation programs. The focus will be on PSpice, which is one of the most widely used general-purpose simulation programs. A simulation example is presented, and the results are compared with those obtained with Power System Simulation Tool based on Simulink.

**Keywords:** power electronics circuits, electric motors, electric drives, PSpice

### 1. INTRODUCTION

Historically, simulation of transient phenomena related to power systems has been carried on using the electromagnetic transients program (EMTP) (Mohan *et al.*, 1994) or one of its variants, such as the alternative transient program (ATP) or electromagnetic transients for dc (EMTDC), which are all based on the trapezoidal integration rule and the nodal approach. These software packages use fixed-step algorithms, which yield excellent results for power systems free of any power electronics devices. However, the fixed-step algorithms do not adapt well to the presence of discontinuities, which are caused by the switching devices.

Spice is a general-purpose circuit simulation program, which was developed at the University of California, Berkeley (Bauer and Duijsen, 2001). The Spice circuit simulation program has become an industry standard. The major advantage of using Spice in power electronics is that, with the same software, a particular circuit can be designed and analyzed at different system and subsystem levels, i.e., at the levels of the power switch, the converter circuit, and converter systems, including feedback control. However, for higher levels of simulation, simplified models for the switch and the converter must be implemented, in order to minimize convergence problems and reduce the run times.

There are several commercial version of Spice that operates on personal computer under several popular operating systems. One commercial version of Spice is called PSpice. It contains models for basic circuit elements (R; L; C, independent and controlled sources, transformer, transmission line), switches, and most common semiconductor devices: diodes, bipolar junction transistors (BJT's), junction field-effect transistors (JFET's), MESFET's, and MOSFET's. PSpice is mainly applied to simulate electronic and electrical circuits for different analyses, including dc, ac, transient, zero pole, distortion, sensitivity, and noise. SPICE uses the nodal approach with a variable-time-step integration algorithm so that it can correctly simulate switching power electronic circuits. Using the analog behavioral modeling (ABM) blocks facilitates the simulation of control systems in PSpice A/D (a commercial version of Spice by MicroSim). However, there are no specific models for electrical machines, circuit breakers, surge arresters, etc. To simulate a power system, the user has to build the needed models using SPICE primitives and basic elements.

Sometimes, users of PSpice claim that the convergence problems are so severe that its use for simulations of power electronics circuits is just not possible or worth the effort. However, this is absolutely not true and with the proper techniques of

gate signal generation, we can simulate just about any given circuit with little or no convergence problems. In addition, if convergence problems are avoided, simulations run much faster and larger numbers of individual transitions can be studied. This is achieved by generating gate signals that are slightly less steep than in real circuits using analog behavioral elements. This gives a lot of insight into the cycle by cycle as well as the system level behavior of power electronics circuit. In this fashion, the function of an existing, as well as the expected performance of a new, proposed circuit, can be studied. An excellent application for these cycle-by-cycle simulations is the development and verification of control strategies for the power semiconductors. The aim of this paper is to present the capabilities of PSpice in simulating power electronics circuits, induction machines and induction motor drives. We present PSpice simulations of voltage source inverters with two levels, induction machine fed by three-phase voltage source inverter and of vector control of induction machine when exact motor parameters are known. These examples represent pure system level simulations, which could have also been done using programs like Matlab/Simulink.

## II SIMULATION OF THREE PHASE VOLTAGE SOURCE INVERTER WITH PSpice

To illustrate the capabilities of the PSpice simulation program, we present an example that shows a complete three-phase inverter bridge using six power MOSFETs. This circuit is shown in Fig. 1. Note that freewheeling diodes are an integral part of every power MOSFET and are not shown separately.

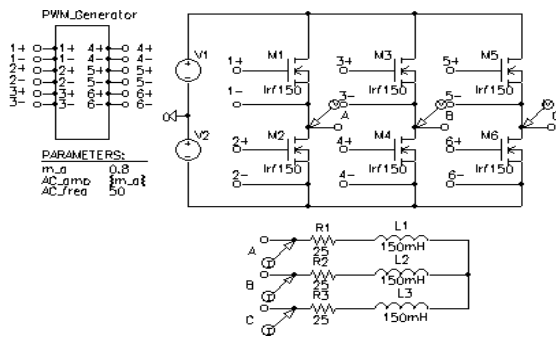


Fig.1 Circuit for a three-phase inverter.

The inverter drives a three-phase load, which could represent an induction motor for a singular operating point. The load is connected to the inverter output terminals with so-called connection bubbles. Due to the number of elements involved, the circuit for the gate drive signal generation is contained in a hierarchical block named PWM\_Generator presented in Fig.2.

The interface ports named "1+", "1-", "2+"..."6-", provide the connection between the subcircuit and the ports of the hierarchical block above. Here the

connection is to the ports on the PWM\_Generator block. The interface ports are created by simply drawing a wire up to the boundary of the block.

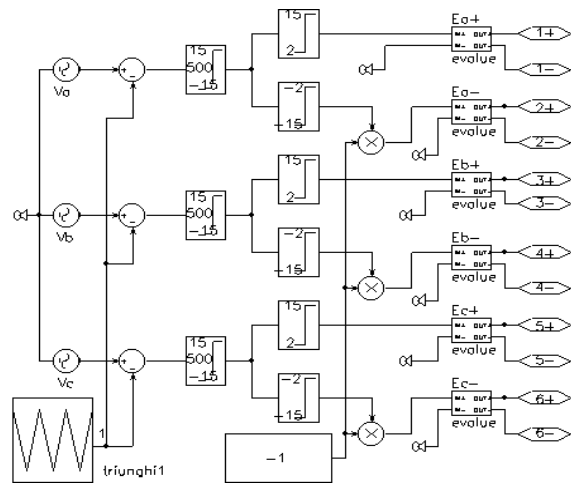


Fig.2 PWM generation sub-circuit for a three-phase inverter.

The interface ports named "1+", "1-", "2+"..."6-", provide the connection between the subcircuit and the ports of the hierarchical block above. Here the connection is to the ports on the PWM\_Generator block. The interface ports are created by simply drawing a wire up to the boundary of the block. The name of the port is initially generic, "Px", where x is a running number, but can be easily edited by double clicking on the generic name. After drawing a block and creating all the ports, double clicking inside the box will open up a schematic page for the subcircuit, which has all the appropriately named interface ports already in it. Additional details on hierarchical techniques can be found in (Andrei, 1994). Careful inspection of the implementation of the soft-limiter element provided in PSpice shows, that it uses a scaled hyperbolic tangent function. It can easily be seen that the result of the soft limiter is an output signal with smooth transitions, which is crucial to avoid convergence problems in PSpice. The soft limiter used here has an upper and limit of  $\pm 15V$  and a gain of 500.

PWM Generator compares a triangular carrier with three sinusoidal reference signals, one for each phase. The triangular carrier signal is symmetrical with respect to the time axis. The values cover the range from - 1.0 to 1.0. For linear modulation, the amplitude range of the reference signals is limited to the amplitude of the triangular carrier, e.g. 1V. The ratio of the reference wave amplitude and the carrier amplitude is called amplitude modulation ratio "m\_a". In the circuit shown in Fig. 1 "m\_a" has a value of 0.8. This value is defined by a parameter symbol and represents a global parameter, which is visible throughout all levels of the hierarchy. In Fig.2 the control functions for the "E\_x+", "E\_x-" sources, where x denotes the phase, are chosen such that the activation voltage levels are  $\pm 2V$ . If the output voltage of the soft limiter is between -2 V and +2V,

no MOSFET is activated, and shoot-through, meaning a short-circuit between the positive and negative bus, is avoided. Fig. 3 shows the simulation results for the three-phase inverter. The time scale is slightly stretched to show the details of the PWM signals better. The graph represents the line-to-line voltage VAB and load currents for all three phases. Due to the inductors contained in the load, the current cannot instantaneously change and follow the PWM signal. Therefore the load current is an almost pure sinusoid with very little ripple. This is representative of the line currents in induction motors.

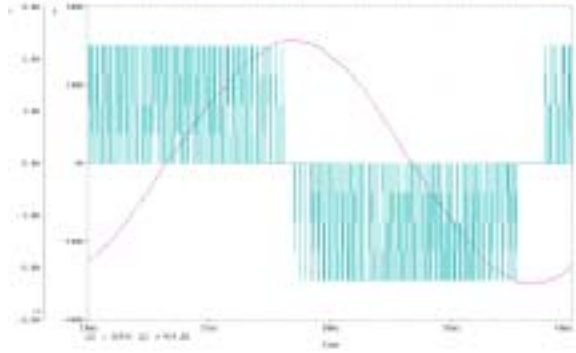


Fig.3 Output waveforms of the three-phase inverter with MOSFET.

### III. SIMULATION OF INDUCTION MACHINE FED BY THREE-PHASE INVERTER

The start-up of an induction motor, fed by the three-phase inverter shown in Fig. 1, is shown. For this purpose, an induction motor replaces the simple passive load in Fig. 1. The induction motor symbol represents the electromechanical model of an induction motor. The model is suitable for studies of electrical and mechanical transients as well as steady state conditions. The induction motor model has been derived for a two-phase equivalent motor. Attached to the motor is a bi-directional two-phase to three-phase converter module. This module is voltage and current invariant. This means that the voltage and current levels in the two-phase and the three-phase machine are equal. Consequently, the power in the two-phase machine is only 2/3 of the power in the three-phase circuit. In Fig. 4 the motor is represented by a custom symbol called "Motor 1". A simple hierarchical block could have been used for the motor, but a custom symbol has been created to achieve a more realistic and pleasing graphical representation. The symbol can be easily created with the symbol editor, which is build into the regular schematic editor.

The d-q model of induction machine is presented in Fig.5. The upper portion of this subcircuit represents the electrical model.

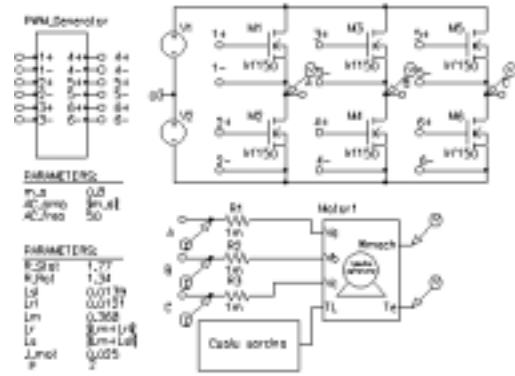


Fig.4 Induction motor start up with three phase inverter circuit.

The task of the electrical model is to calculate the stator and rotor currents, where the stator voltages and the mechanical speed of the machine are input parameters.

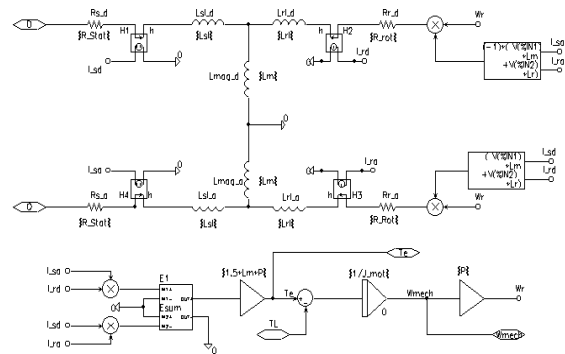


Fig.5 Subcircuit for d-q induction motor model.

The equation system for the electrical model is given by equation (1). The theory for this equation system is derived in (Bose,2002). The equation system and the model are formulated for the stationary reference frame.

$$\begin{bmatrix} V_d \\ V_q \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} R_{stat} + pL_s & 0 & pL_m & 0 \\ 0 & R_{stat} + pL_s & 0 & pL_m \\ pL_m & \omega_e L_m & R_{rot} + pL_r & \omega_e L_r \\ -\omega_e L_m & pL_m & -\omega_e L_r & R_{rot} + pL_r \end{bmatrix} \begin{bmatrix} I_{sd} \\ I_{sq} \\ I_{rd} \\ I_{rq} \end{bmatrix} \quad (1)$$

$$L_s = L_m + L_{s1}; L_r = L_m + L_{r1}; p = d/dt \quad (2)$$

The bottom of Fig. 5 represents the mechanical model. This circuit calculates the internally generated electro-magnetic torque using the stator and rotor currents as input values. The equation for the torque is given by (3) (Bose,2002):

$$Te = (3/2)p(I_{sq}I_{rd} - I_{sd}i_{rq}) \quad (3)$$

Using the generated torque, the load torque and the moment of inertia, the angular acceleration can be calculated. Integration of the angular acceleration yields the rotor speed, which is used in the electrical

model. Since typical induction machines are three-phase machines, it is often desirable to have a machine model with a three-phase input. Therefore a bi-directional two-phase to three-phase converter module, which can be attached to the motor, has been developed. A subcircuit for this module is shown in Fig. 6. This circuit is truly bi-directional, meaning that the circuit can be fed with voltage or current sources from either side. An interesting detail of the subcircuit is the three-phase switch on the input. This switch is necessary to ensure a stable initialization of the simulator in case the machine is fed with a controlled current source. The switch provides an initial shunt resistor from the three-phase input to ground. Soon after the simulation has started, the switch opens and leaves only a negligible shunt conductance to ground.

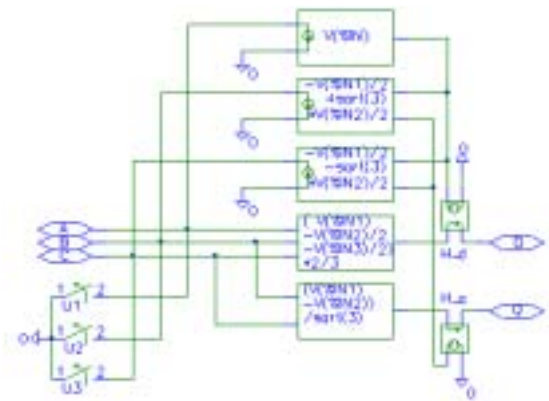


Fig. 6 Subcircuit for ABC-DQ transformation.

Fig. 7 shows the results for the start-up of the induction motor for the circuit of Fig.4. The motor's parameter is presented in Fig.4. The top trace in Fig. 7 shows the developed electromagnetic torque. The scale for this graph is 1V = 1 Nm. The graph below shows the mechanical angular velocity with a scale of 1 V = 1rad/s. Below the graph for the rotor speed, all three input currents are shown. Input voltage is the PWM waveform shown in the bottom graph.

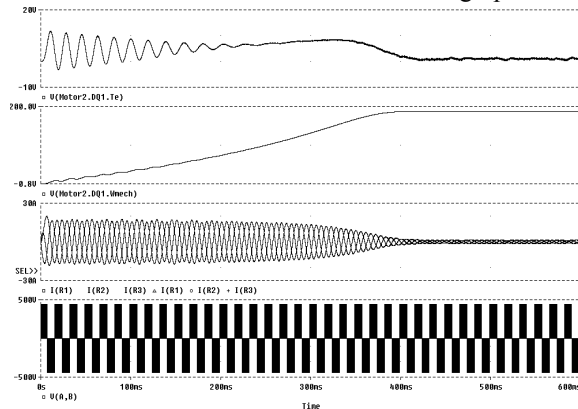


Fig.7 Induction machine with three-phase inverter.

#### IV SIMULATION OF AC INDUCTION MACHINES USING VECTOR CONTROL

This example will demonstrate the use of PSpice for simulations of ac induction machines using Field Oriented Control. The basic idea of field oriented control is to inject currents into the stator of an induction machine such that the magnetic flux level and the production of electromagnetic torque can be independently controlled and the dynamics of the machine resembles that of a separately excited dc machine without armature reaction. Using induction motor model for the synchronous reference frame, the D input of the induction machine would correspond to the field current that produces the flux, and Q input to the current component that produces electromagnetic torque by a linear relation (constant kT). Induction motor must be supplied with ac voltages and currents with a frequency determined mostly by the rotor speed to a small extent by the commanded torque. We still supply dc values representing the commanded flux and torque but we transform these dc values to appropriate ac values. We will assume that we can measure the actual rotor speed with a sensor. If we add the slip speed, that we determine mathematically from the torque command, to the measured rotor speed, we obtain the synchronous speed for the given operating point. With this synchronous speed we can transform the dc flux and torque command values from the synchronous reference frame to the stationary reference frame. We accomplish this by using a rotational transformation according to the matrix equation (4) (Bose,2002).  $\theta$  angle can be interpreted as the momentary rotational displacement angle between two Cartesian co-ordinate systems; one containing the input values and the other one the output values. This angle is obtained by integration of the angular velocity which the co-ordinate systems are rotating.

$$\begin{bmatrix} V_{D\_out} \\ V_{Q\_out} \end{bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) \\ \sin(\theta) & \cos(\theta) \end{bmatrix} \begin{bmatrix} V_{D\_in} \\ V_{Q\_in} \end{bmatrix} \quad (4)$$

Figure 8 shows the top level of a simulation example that implements vector control for induction machine with a stationary reference frame. The objective of the speed loop is to keep the speed at its initial value of 182 rad/sec, in spite of the load torque disturbance at  $t=1.5s$ . It was supposing that induction motor work in steady state with load torque  $TL=12.64$  Nm and at time  $t=1.5s$   $TL$  suddenly goes to one half of initial value. We will design the speed loop with a bandwidth of 25 rad/sec and a phase margin of 60 degrees (Şchiop, 2003).

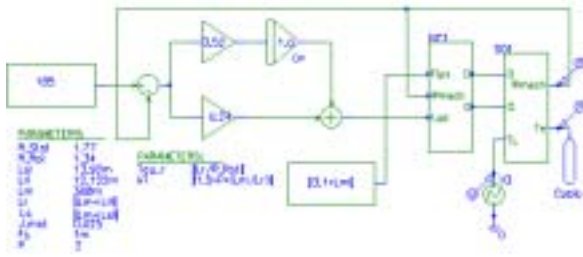


Fig.8 Indirect vector control.

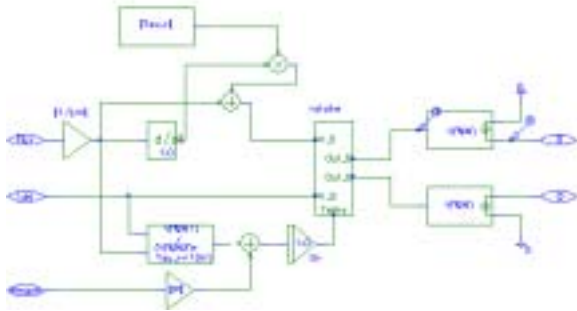


Fig.9 Sub-circuit for indirect vector control.

PSpice simulation results are presented in Fig.10 for electromagnetic torque, load torque, and speed variation. For the sake of comparison between the PSpice and other available software, we have considered Matlab/Simulink. The same converter is simulated under the same conditions using both types of software. It is to be mentioned that there is very good agreement between the results obtained by the PSpice and Matlab/Simulink.

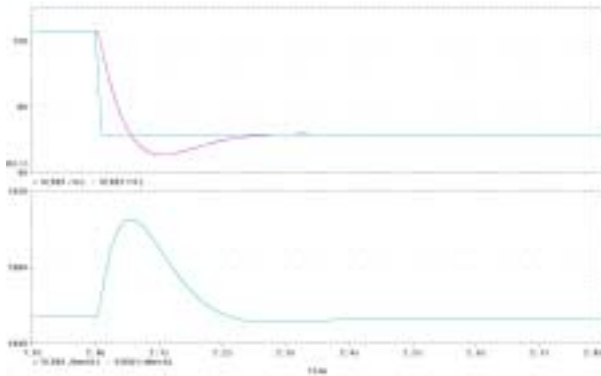


Fig.10 Spice simulation results.

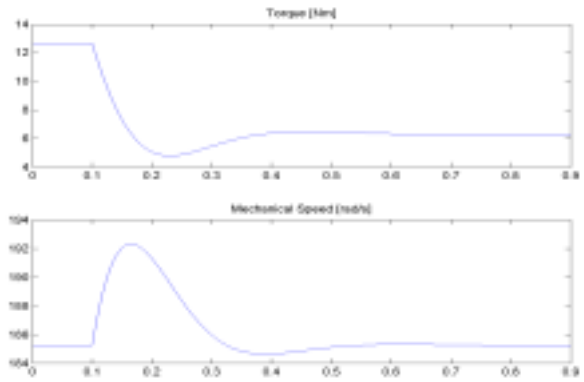


Fig. 11 Matlab/Simulink results.

## V CONCLUSIONS

This paper shows how power electronics circuits, electric motors and drives, can be simulated with PSpice software. A simulation example is presented, and the results are compared with those obtained with Matlab/Simulink.

## REFERENCES

- Mohan, N., W. Robbins, T. Undeland, R. Nilssen, Mo, O. Simulation of Power Electronic and Motion Control System- *An Overview Proceedings of the IEEE*, vol. 82, No.8, August 1994, pp. 1287- 1302.
- Bauer, P., P. Duijsen Modeling and Simulation for Power Electronics and Electrical Drives Mercury-Smekal, Kosice, 2001.
- Andrei Ş, *PSpice-Analiza asistată de calculator a circuitelor electronice* Ed. ICPE, Bucureşti, 1996.
- Bose B. K. *Modern Power Electronics and AC Drives* Prentice Hall PTR, Upper Saddle River, 2002.
- Şchiop A (2003) Analysis and Design of Speed Controller for Vector Controlled Induction Motor Drives *EMES 2003*, Oradea, pp.122-128.