SIMULATION IN ELECTRICAL ENGINEERING WITH PSPICE

Philippe MISSIRLIU
Lycee Newton Enrea
1 Pl. Jules Verne
92110 Clichy
France

Why simulate ?

The use of a simulation software is an essential requirement to teach electrical engineering :

- Simulation technique gives results, when lacking sophisticated hardware for tasks such as: current measurement in a transistor, calculation of Fourier transform of a signal, measurement of magnetic fluxes.
- Students being what they are (and not what they should), many phenomena are too complex for a theoretical presentation. Simulation allows the presentation of findings, when time is short for explanation.
- Simulation is increasingly used in the industry, familiarising students with its use is therefore mandatory.

Why Pspice ?

Amongst the many software available on the market, Pspice has many advantages:

- It is based on the industrial standard Spice and therefore gives access to many libraries of models, developed by manufacturers.
- Pspice is widely used in industry.
- It allows the mixing of digital and analog parts without any problems.
- Learning the software is quite easy.

The professional version of Pspice is not within the range of budgets allotted to our high school. Fortunately an evaluation version is available for absolutely free, either to be downloaded from Internet or as a CDROM ordered from the distributor. An evaluation version has limitations of course. But here we are limited only in the number of nodes of the scheme to be simulated. All the many other features are available.

We have used Pspice in our department, in student projects, for three years. To that end, we developed a number of easy steps, along with a library of optimised parts. We have thereby managed to cover all the aspects of an electromechanical system: power electronics , mechanics, digital and analog electronics, regulation, all with one software, moreover one free software!

The aim of this paper is to illustrate the above stated, by way of a frequently used topic in EE: the speed control of a DC motor fed by a 3-phase controlled rectifier.

Switches

The evaluation version of Pspice offers a series of traditional switches such as: bipolar transistors, field-effect transistors, IGBT (starting from version 8), SCRs, triacs. However, an attempt at simulating a simple 3phase controlled rectifier, will produce two problems:

- The SCR available in the evaluation version has a VRRM of 50V and a max current of 1Amp.
- Using more than three SCRs results in the message : « EVALUATION VERSION analog Node Limit (64 Nodes) Exceeded! »

The solution consists in designing a "home made " SCR. This part does not completely simulate a real SCR, as there is no provision for a triggering circuit. Phenomena such as the hold current, or dV/dt triggering, are not simulated. Nonetheless the model proved satisfactory for our use.

Figure 1 shows a 3phase controlled rectifier using this part. It will be noticed that there is no apparent triggering circuit for the SCRs. The simulation results are presented in figure 2. Pspice has powerful means of visual presentation of results. One can visualise dimensions as instantaneous, average, rms values, and calculate an harmonic analysis.

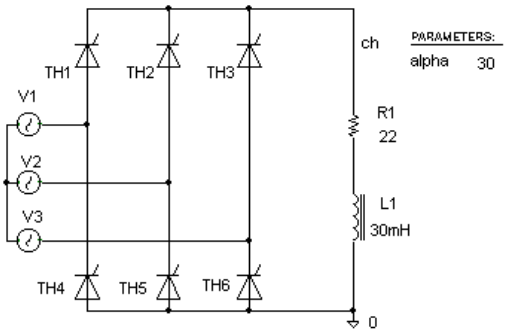

Figure 1

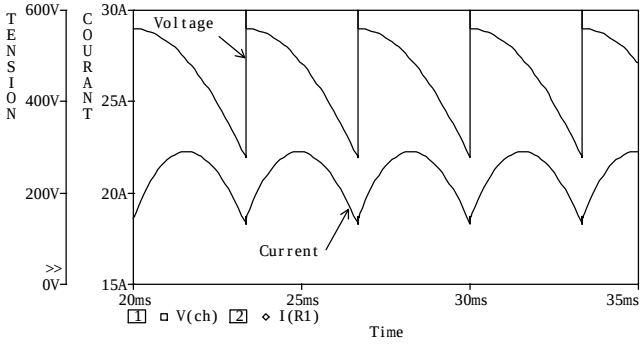

Figure 2

There are also more sophisticated functions available. In this case, one could contemplate visualising the variations of the average value of output voltage, as a function of the triggering delay.

Simulation of mechanical systems.

A software usable in electrical engineering must be able to simulate mechanical dimensions. This does not necessarily constitute an initial feature. We did however accomplish this by using a classical analogy between mechanical and electric dimensions:

Speed	Voltage
Torque	Current
Rotational inertia	Capacitance of a capacitor
Viscous friction	Reciprocal of a resistor
Elastic connection	Reciprocal of an inductance

By using this analogy, we developed a model of a DC MOTOR along with a model of a mechanical load. The parameters of these models may be modified. Figure 3 shows the modeled DC MOTOR fed from the 3phase rectifiers above mentioned. The simulation results are seen in figure 4.

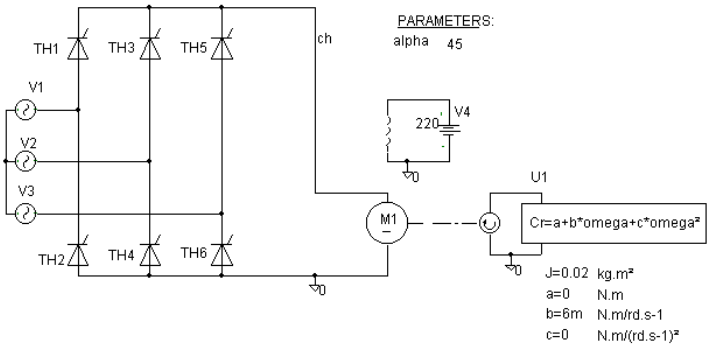

Figure 3

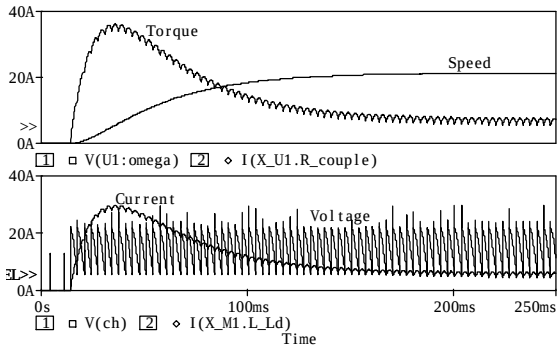

Figure 4

Regulation

Pspice allows the use of blocks defined by a mathematical function, such as Laplace transforms. We used this possibility to model the speed control of a DC motor. In order to avoid prolonged calculating time, the 3phase rectifier of the preceding paragraph, had to be replaced by a simpler model. For a start a simple gain. Eventually,the simulation of the influence of the rectifier may

be added, by introducing a delay. The classical structure with two intertwining loops with PI correctors is presented in figure 5

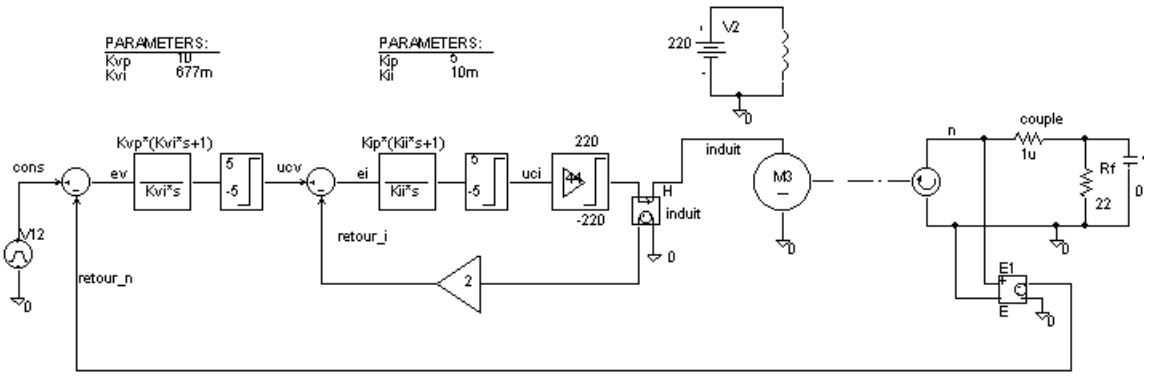

Figure 5

Figure 6 illustrates the response of the system. Pspice obviously allows the study of harmonics and the tracing of classical diagrams ie: Bode, Nyquist, and Black. One may easily wiew the effect of a modification of a parameter on the system's behavior. This makes Pspice a great teaching tool.

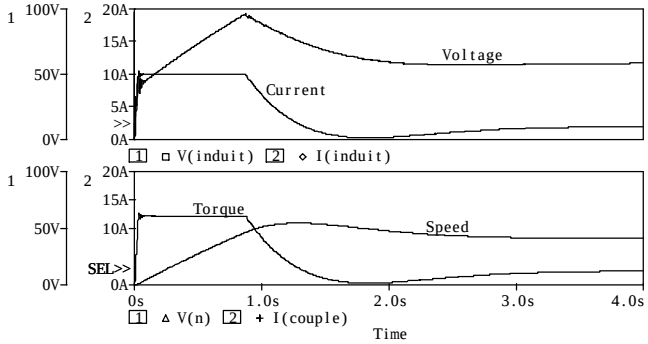

Figure 6

Control circuits

Simulation of a control circuit does not call for specific adjustments. The software is designed for this type of application. Digital and analog parts may be combined. The example presented is a project developed at the ENREA: a singlephase 50Hz inverter with PWM control, see schematics in figure 7. The aim is to suppress the first two harmonics.

The control sequence of the switches is introduced by way of an EPROM in which the first 1024 addresses are scanned every 20ms. Simulation results are presented in figure 8. Harmonics analysis of the data is presented in figure 9. The first harmonic of the tension measured is of the 7-th order. Note that the file used to program the virtual EPROM is identical with the one used for the real EPROM.

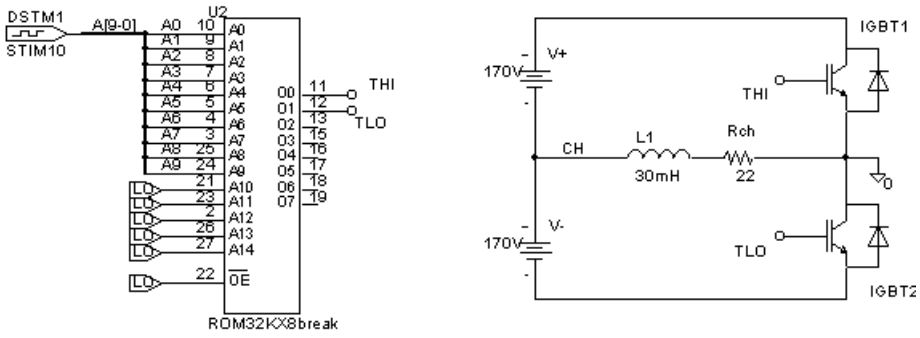

Figure 7

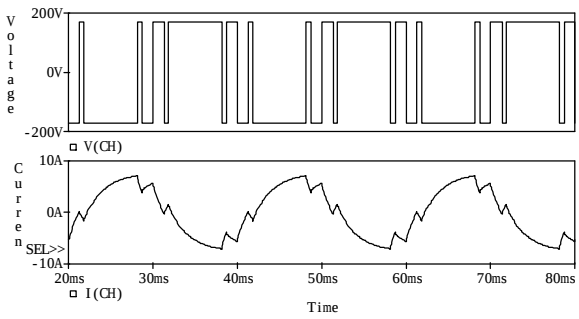

Figure 8

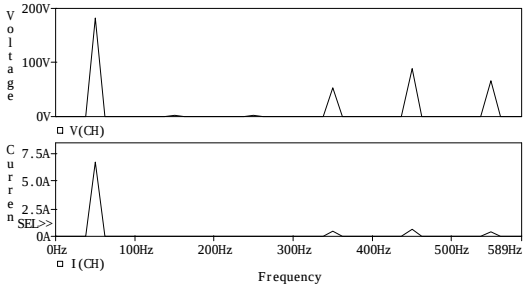

Figure 9

« Home made » parts

Spice is the de facto standard in the industry. One resulting advantage being the frequent offer of Spice compatible models by manufacturers. As an example: we use an AD633 analog multiplier manufactured by Analog Devices at the ENREA. We downloaded a Spice model of the part from their Internet site. We just created a corresponding symbol and added it to our own "home made" library. Another means of creating a part model is the ABM behavioural modelisation, using blocks defined by mathematical functions. The technique was used to create a model of the AD736 RMS converter of Analog Devices, of which no model was available. We used the synopsis of the part and the specifications mentioned in the manufacturer literature.

Conclusion

We have been using the demo version of Pspice at Lycee Newton ENREA for three year to our complete satisfaction. People interested may download the library we have created, from the attached Internet site, together whith the simulations contained in this paper.