A Comparison & Performance of Simulation Tools
MATLAB/SIMULINK, PSIM & PSPICE for Power Electronics Circuits

Santosh S. Raghuwanshi  
EX Dept., RGPV, Bhopal (M.P.) India  
er_sant@rediffmail.com

Ankita Singh  
EX Dept., RGPV, Bhopal (M.P.) India

Yamini Mokhariwale  
EX Dept., RGPV, Bhopal (M.P.) India

Abstract—The software packages available for simulation of power electronic circuits are MATLAB, PSPICE and PSIM. In this paper the simulation of rectifier and inverter circuits were performed in the MATLAB, PSPICE and PSIM environment and the comparison of results were made. Rectifier and inverter are the power electronic circuits which were highly used in all power supply units. The paper explains the theoretical approach of these circuits and then the simulation results are given in order to show the effectiveness of the system in the simulation arena.

Keywords—MATLAB, PSIM, PASICE, DC-AC Converter, AC-DC Converter

I. INTRODUCTION
Power Electronics is interdisciplinary and is at the confluence of three fundamental technical areas - power, electronics and control, and is used in a wide variety of industries from computers, chemical plants to rolling mills. The importance of power electronics has grown over the years due to several factors. Computer simulation can greatly aid in the analysis, design and education of Power Electronics. A computer simulation (or "sim") is an attempt to model a real-life or hypothetical situation on a computer so that it can be studied to see how the system works. By changing variables, predictions may be made about the behavior of the system. In our work towards this we have ensured to bring out the different responses of current and voltage in the power electronic circuits. However, simulation of power electronics systems is made challenging by the following factors:

- Extreme non-linearity presented by switches
- Time constants within the system may differ by several orders of magnitude and
- A lack of models

Therefore, it is important that the objective of the computer analysis be evaluated carefully and an appropriate simulation package be chosen.

In view of the above considerations, a SPICE based simulation package, MATLAB and PSIM have been used for simulating the power electronic circuits like rectifiers, inverters, choppers and AC voltage controllers. They have had the detailed device models and have been able to represent the controller portion of the converter system by its functional features in as simplified a manner as possible. In this paper the simulation of rectifier and inverter circuits were taken into consideration.

A rectifier is an electrical device that converts alternating current (AC) to direct current (DC), a process known as rectification. Rectifiers have many uses including as components of power supplies and as detectors of radio signals. Rectifiers may be made of solid state diodes, vacuum tube diodes, mercury arc valves, and other components.

An inverter is an electrical or electro-mechanical device that converts direct current (DC) to alternating current (AC); the resulting AC can be at any required voltage and frequency with the use of appropriate transformers, switching, and control circuits. Static Inverters have no moving parts and are used in a wide range of applications, from small switching power supplies in computers, to large electric utility high-voltage direct current applications that transport bulk power. Inverters are commonly used to supply AC power from DC sources such as solar panels or batteries. The electrical inverter is a high-power electronic oscillator. It is so named because early mechanical AC to DC converters was made to work in reverse, and thus was "inverted", to convert DC to AC. The inverter performs the opposite function of a rectifier. The comparison of the software tools are explained in the following sections.
MATLAB is numeric computation software for engineering and scientific calculations. MATLAB is being used for circuit theory, filter design, random processes, control systems and communication theory. MATLAB matrix functions are shown to be versatile in doing analysis of data obtained from electronics experiments. The graphical features of MATLAB are especially useful for display of frequency response of amplifiers and illustrating the principles and concepts of semiconductor physics. The interactive programming and versatile graphics of MATLAB is especially effective in exploring some of the characteristics of devices and electronic circuits.

PSIM is simulation software specifically designed for power electronics and motor drives. With fast simulation and friendly user interface, PSIM provides a powerful simulation environment for power electronics, analog and digital control, magnetics, and motor drive system studies. Powersim develops and markets leading simulation and design tools for research and product development in power supplies, motor drives and power conversion and control systems.

SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis and was inspired by the need to accurately model devices used in integrated circuit design. It has now become the standard computer program for electrical and electronic simulation. The majority of commercial packages are based on SPICE2 version G6 from the University of California at Berkeley although development has now progressed to SPICE3. The increased utilization of PCs has led to the production of PSPICE, a widely available PC version distributed by the MicroSim Corporation whilst HSPICE from Meta-Software has been popular for workstations and is now also available for the PC. One of the reasons for the popularity of Pspice is the availability and the capability to share its evaluation version freely at no cost. This evaluation version is very powerful for power electronics simulations.

II. DC-AC CONVERSION

In this section the simulation of rectifier is explained with three different simulation packages and the corresponding waveforms are plotted.

A. PSIM Based Simulation Method

Fig. 1 shows SCR based full bridge inverter circuit diagram in PSIM. In this circuit Vs is 100v, load R-L, resistance is 1kohm and inductance is 1m henry. The SCR switching frequency is 1k hertz. The output wave form shown in fig.2 in this wave sinusoidal output voltage is 100v and output current 0.1amp.

B. MATLAB Based Simulation Method

Fig. 3 shows MOSFET based full bridge inverter circuit diagram in MATLAB. In this circuit Vs is 100v, load R-L, resistance is 1kohm and inductance is 1m henry. The SCR switching frequency is 1k hertz. The output wave form shown in fig.4 in this wave sinusoidal output voltage is 100v and output current 0.1amp.

C. PSPICE Based Simulation Method
PSPICE based simulation for full bridge inverter and the corresponding output waveforms are shown in Fig. 5-6 respectively.

![Fig.5 Inverter circuit in PSPICE](image)

Fig. 5 Inverter circuit in PSPICE

![Fig.6 Output voltage waveform](image)

Fig. 6 Output voltage waveform

III. AC-DC CONVERSION

In this section the simulation of rectifier is explained with three different simulation packages and the corresponding waveforms are plotted.

A. PSIM based Simulation Method

Fig.7 shows the power circuit of the fully controlled single-phase PWM converter in PSIM, which uses four transistors with anti parallel diodes in SCR bridge block to produce a controlled dc voltage Vo. Using a bipolar PWM switching strategy, this converter may have two conduction states:

- transistors T1 and T4 in the ON state and T2 and T3 in the OFF state; or
- transistors T2 and T3 in the ON state and T1 and T4 in the OFF state.

In this topology, the output voltage Vo must be higher than the peak value of the ac source voltage vs in order to ensure proper control of the input current. The input voltage value Vs= 200v, switching frequency is 1KHz. After simulation the output voltage and current wave form with resistive load is shown in fig. 8. In this wave harmonics are presence.

![Fig.7 Single-phase full bridge converter circuit in PSIM](image)

Fig.7 Single-phase full bridge converter circuit in PSIM

B. MATLAB Based Simulation Method

In MATLAB the simulation has been carried out for rectifier in Simulink block set. Here the full wave rectifier is used which conducts for both positive and negative half cycles respectively. Fig.9 shows the circuit diagram for full wave rectifier and the corresponding input and output waveforms are shown in Fig.10. The specifications of the circuit are: Input AC Voltage=48V, Load resistor=100Ω and employing 4 diodes for rectification. The output thus obtained is a waveform without using a filter circuit. By employing a filter circuit a pulsating DC waveform can be obtained. The circuit is uncontrolled full bridge rectifier. For a controlled rectifier thyristor switches can be used.

![Fig.9 Full wave rectifier circuit in MATLAB](image)

Fig.9 Full wave rectifier circuit in MATLAB

![Fig.10 Input and Output voltage waveforms](image)

Fig. 10 Input and Output voltage waveforms

C. PSPICE Based Simulation Method

PSPICE based simulation for full bridge rectifier is shown in Fig.11. The corresponding input and output voltage waveforms are shown in Figs. 12 and 13 respectively. The circuit specifications are input AC Voltage=20V, frequency=50Hz and load resistor=100Ω.
IV. CONCLUSIONS

A detailed analysis of simulation using the software tools like MATLAB, PSIM and PSPICE are given. Here the examples taken are rectifier and inverter circuits. The waveform which was obtained has to be analysed in each and every half cycle interval of time. The software packages provide the way for getting the sequences happening in each and every cycle. In addition the various parameters which can be measured are voltages and currents across the inputs, outputs and also across the switches. A comparison between the software’s discussed is listed below.

- MATLAB finds applications in all areas from control systems to robotics. Any controller can be designed and tested in the simulation arena for power electronics and power system based circuits. This also provides the provision of graphical user interfaces and m file programming to design the various intelligent controllers like neural networks, fuzzy logic control, genetic algorithms etc.,

Advantages of using PSPICE are:

- PSpice allows multiple plots to be viewed simultaneously, such as voltage, power, etc. Also, specific points, such as a voltage at a certain time, can be selected and marked on the output plot in PSpice,

- PSpice contains libraries full of specific components with manufacturer specifications. These components are included so the user may obtain realistic simulation results,

- Very simple to represent any electrical circuit, in particular power-electronic circuits and

- a wide library of commercial electric components are available.

Usage of PSIM increases due to:

- With PSIM’s interactive simulation capability, we can change parameter values and view voltages/currents in the middle of a simulation. It is like having a virtual test bench running on our computer,

- we can design and simulate digital power supplies using PSIM’s Digital Control Module. The digital control can be implemented in either block diagram or custom C code,

- PSIM has a built-in C compiler which allows us to enter our own C code into PSIM without compiling. This makes it very easy and flexible to implement our own function or control methods,

- We can use the Thermal Module to calculate semiconductor device losses (conduction losses and switching losses) based on the device information from manufacturers datasheet.

Today’s computer technology enables a new approach to this work which has not been considered feasible before. Simulation programs will run on inexpensive machines and be widely available. Circuits will be specified in a simple graphical format which is self documenting. Models will be available to meet today’s needs and yet be sufficiently versatile to be adapted to new devices as they appear. By means of a suitable choice of simulator elements, even the inexpert user will be able to customize his package to incorporate future device developments. In comparing the above mentioned packages for a wide variety of applications all the software’s provide its own unique property in obtaining and analyzing the results. Hence its upto the user to decide the software as either MATLAB or PSPICE or PSIM depending upon there area of work and applications. The other software packages such as spice and octave/scilab can also be compared with the existing packages.

REFERENCES


Kunrong Wang, Fred C. Lee, Jason Lai, Operation principle of bi-directional full-bridge DC-DC converter with unified soft-switching scheme and soft-switching capability, APEC 2000, pp.1 I-1 I-1 I


Power Electronics: Computer Simulation, Analysis and Education Using PSpice Schematics by Prof. NED MOHAN.

Power Electronics: converters, applications and design by MOHAN.UNDELAND.ROBBINS.