World Academy of Science, Engineering and Technology
International Journal of Electrical and Computer Engineering
Vol:3, No:1, 2009

A Matlab / Simulink Based Tool for Power Electronic Circuits

Abdulatif A. M. Shaban

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf

Abstract**—**Transient simulation of power electronic circuits is of considerable interest to the designer. The switching nature of the devices used permits development of specialized algorithms which allow a considerable reduction in simulation time compared to general purpose simulation algorithms. This paper describes a method used to simulate a power electronic circuits using the SIMULINK toolbox within MATLAB software. Theoretical results are presented provides the basis of transient analysis of a power electronic circuits**.**

Keywords**—**Modelling, Simulation.

I. Introduction

POWER electronic circuit simulation is a subject whose importance can hardly be overemphasizes especially with new topologies and control strategies coming up quickly. General purpose simulators such as SPICE are inadequate in such areas since they are use micromodels of power and control circuit components. Such detailed analysis is too time consuming and unnecessary in the case of power electronic circuits where devices occupy only two distinct states [1]. Clearly these circuits are also of variable topology due to the switching action of devices, and hence one needs a special purpose power electronic simulation program which handles changes in topology in a unified manner without the user needing to provide a system description for every switch state combination.

State variable methods have been popular and have been used in many cases [1]. The facilities provided by the toolbox in SIMULINK/MATLAB are used to describe the role of computer simulations in the analysis and design of power electronics system. There are many benefits associated with simulation in the design process, which may be summaries as:

1. Simulations are well suited for educational purpose. It is an efficient way for a designer to learn how a circuit and associated control function which may lead to technical and / or cost improvements.
2. Simulations may give a comprehensive insight into system performance thereby giving a competitive edge to a company in their sphere of operation.
3. It is normally much cheaper to do a thorough analysis than to build the actual circuit in which component stresses are measured.
4. Destructive tests which could not be performed in the laboratory either because of safety or because of the costs involved can be simulated. Responses to faults and abnormal conditions can be thoroughly analyzed.
5. Simulated waveforms at different points of the circuit may be easily monitored without the hindrance of measurement noise.
6. It is possible to simplify parts of circuits in order to focus on a specific portion of the circuit. This may not be possible in a laboratory setup.

Abdulatif-A-M-Shaban is with the Alfateh University, Faculty of Janzoor Teachers, Tripoli-Libya (e-mail: a.shaban@alfateh.edu.ly).

II. Challenge in Computer Simulation

Simulation of power electronics and motion control systems poses many challenges to the simulation programmer and eventually to the use. The objective is always to make the simulation tool transparent and minimise the input data. The requirements for a simulation program may be summarised as providing:

1. User Friendly Interface: a simulation program must have an easy to use interface for the data entry and for the output data processing.
2. Mult-Level Modelling Capability: in the simulation of a motor drive, for example, the power electronic converters are described by the interconnection of circuit element models. On the other hand, the electrical machine and the load are best described by differential equations formulated in terms of state variables.
3. Accurate Models: for a detailed analysis a simulation program needs accurate models of all circuit elements. Even if accurate models are available it is difficult to know their parameter values. Parasitic inductances and capacitances are often difficult to estimate.
4. Robust Switching Operations: switching actions due to solid state switches (diodes, thyristors, and transistors) must be appropriately handled. Based on how the switches are modelled, their on/off transitions either represent an extreme nonlinearity or lead to a time varying structure of network [2].

The challenge listed above dictate that there is careful evaluation of the objective of the simulation. In general it is not desirable to simulate all aspects of the system in detail. The reason is that the simulation time may be very long and the output information content may be overwhelming, thus obscuring the phenomena of interest. In this respect the best

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf

simulation is the simplest possible which meets the immediate objective.

III. Mechanics of Simulation

At the core of each simulation there are differential and algebraic equations which describe the system. Simulation programs primarily differ in how these equations are solved. Other differences such as the program/user interface (pre-and post-processor, graphical capabilities) are also unique to a simulation program. There are two different types of simulation tools: 1) circuit oriented simulators and 2) equation solvers. In the circuit oriented programs the user need only supply the interconnection of the circuit element models. From this information, these programs themselves develop the system equations. This results in a very short setup time and it is easy to make changes in the circuit topology. The negative side of circuit oriented programs is that, there is little control over the simulation process which can lead to long simulation times or even worse to numerical convergence or oscillation problems, especially for analogue circuits causing the simulation to halt. An alternative to the use of circuit oriented simulators is to describe the circuit and the controller by means of differential and algebraic equations. It is possible to solve these equations using any one of the several higher level computer languages. In addition packages such as MATLAB contain sophisticated tool boxes for control analysis and design. MATLAB can easily perform array and matrix manipulations. Powerful plotting routines are built in SIMULINK is a powerful graphical user interface to MATLAB which allows dynamic system to be described in an easy block diagram form.

A. Treatment of Switches

Switches are the most widely used elements in power electronic simulations because all semiconductor devices can be modelled as switches. Switch models are the origin of most of the difficulties for the numerical routines used. It is therefore of great importance that the models of switches are carefully handled by the simulation program. However, the ideal switching yields a solutions, which although only qualitative, is sufficiently close to reality to permit the derivation of some important practical results [3,4]. As the thyristor is considered here as ideal switch then there is no voltage drop when conducting or delay in the switching process. The inclusion of the device voltage drop is not difficult; however, with a value of 1-2 volts depending on device characteristic and a supply voltage of 3000-4000 volts the error is insignificant.

The accuracy does depend on considering all aspects of parameters and in the end the simulation can only provide an indication of the results that would be obtained in practice.

With present day PC computing power and the availability of reliable circuit simulation software packages, power converter performances can be investigated readily and with confidence. In power electronics, because of the presence of semiconductor switches and other nonlinear devices, modelling in the PC environment can be difficulty and time consuming, frequently necessitating 'overnight' program execution. This problem can be pressing and to a certain extent can be avoided by shifting into 'switches' converter concept using MATLAB/SIMULINK, as will be demonstrated in the following section.

IV. MATLAB/Simulik for Power Electronic

The following section will looks at how the modelling and simulation of a power electronic converter can be carried out using MATLAB/SIMULINK software.

The blocks which are used to achieve the modelling as follow:

1. Repeating Sequence: this block will be used to generate a number of pulses in terms of time.
2. Sinewave: is used to generate a sinusoidal input with amplitude, frequency and phase.
3. Switch Function (Thy): is used to switch between the sinewave form and the firing pulse which is generated from the repeating sequence.
4. OR Logic: is used to turn the thyristor off when current reaches zero.

The power semiconductor devices can be operated as switches by applying control signals to the gate terminal of thyristors. The switching characteristic is shown in Fig. 1. The rise time of the anode current of a thyristor is define as the time taken for the thyristor voltage to fall from the 90% to the 10% level. The sum of the delay and rise times is the turn on time of the thyristor. Both rise time and the fall time determine the quality of a pulse waveform.

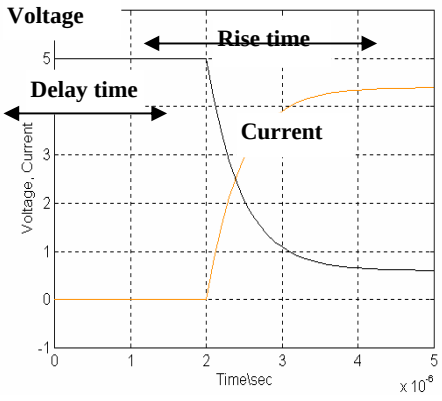


Fig. 1 Typical thyristor switching characteristic (turn on)

V. Simulation and Performance of Power Electronic Circuits

The simplest phase controlled rectifier circuit connected to inductive load is shown in Fig. 2. The presence of a commutating diode which prevents the load voltage reversing beyond the diode volt-drop value, resulting in the waveforms is shown in Fig. 3.

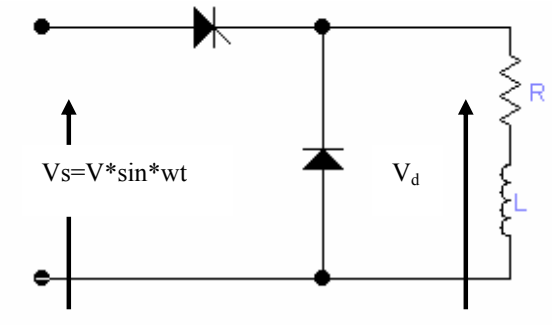


Fig. 2 Single phase half wave controlled rectifying circuit with commutating diode

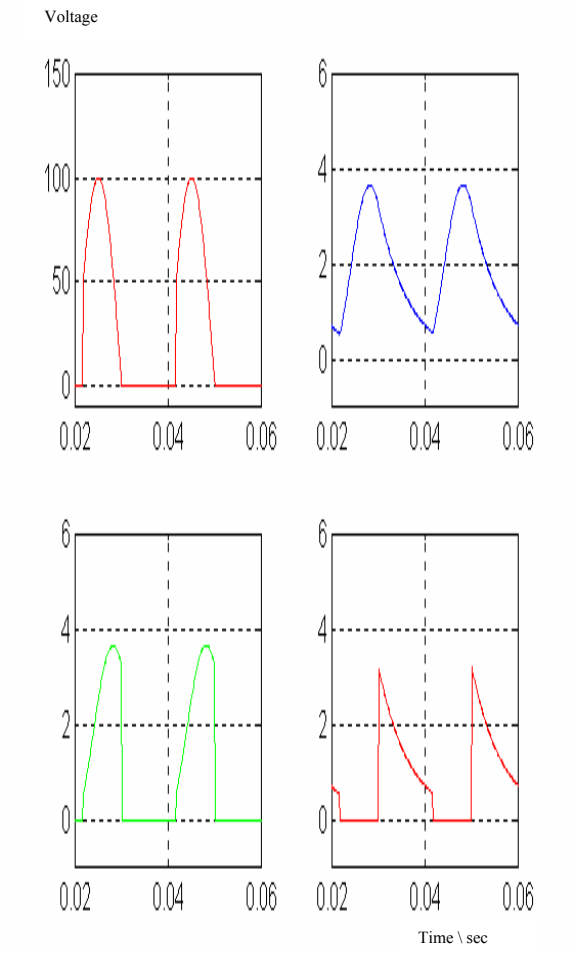


Fig. 3a Waveform for signal phase half wave controlled circuit with commutating diode

During the "thyristor on" period the load current is dictated by equation (1), but once the voltage reverses V_L is effectively zero and the load current follows an exponential decay.

$$V_L = R.i_L + L.\frac{di_L}{dt} \quad (1)$$

If the current level decays below the diode holding level then the load current is discontinuous as shown Fig. 3b. Fig. 3 shows a continuous load current condition where the decaying load current is still following when a thyristor is fired in next cycle. The load voltage waveform has a mean value for voltage given by:

$$V_{mean} = \frac{1}{2\pi}.\int_{\alpha}^{\pi} V_{max}.\sin\theta.d\theta = \frac{V_{max}}{2\pi}(1+\cos\alpha) \quad (2)$$

Inspection of the waveforms clearly shows that the greater the firing delay angle σ the lower is the mean load voltage, equation 2 confirmation that it falls to zero when σ=180°.

The thyristor voltage waveform V_t (Fig. 4) shows a positive voltage during the delay period and also both the peak forward and peak reverse voltage are equal to V_{max} of the supply.

Inspection of the waveforms in Fig. 2 and Fig. 3 clearly shows the two roles of the commutating diode, to prevent negative load voltage and to allow the thyristor to regain its blocking state at the voltage zero by transferring (or commutating) the load current away from the thyristor.

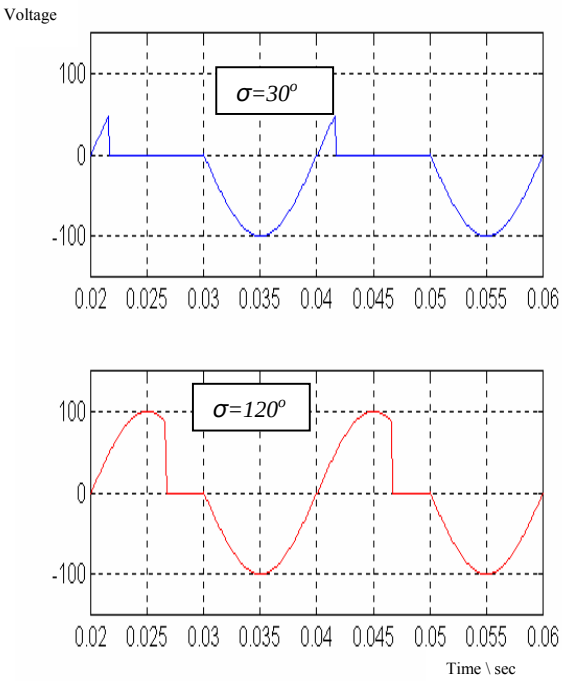


Fig. 3b Thyristor voltage waveforms for single phase half wave control circuit with commutating diode with σ=30°, σ=120°

A. Fully Controlled Single Phase Converter

The main elements of a fully controlled single phase converter are illustrated in Fig. 4, comprise four thyristors connected in a bridge formation [5]. The positive load terminal can be connected (via Thy1) to terminal A or (via Thy2) to terminal B of the input source and likewise the negative load terminal can be connected either to A or to B via Thy3 or Thy4 respectively.

The interest is to find the model operation applicability of the output voltage waveform on the d.c. side and in particular to discover how it can be controlled by varying the firing delay angle σ. The voltage waveform for a given σ will

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf

depend up on the nature of the load and to explore the basic mechanism of phase control the case where the load is a resistive is considered.

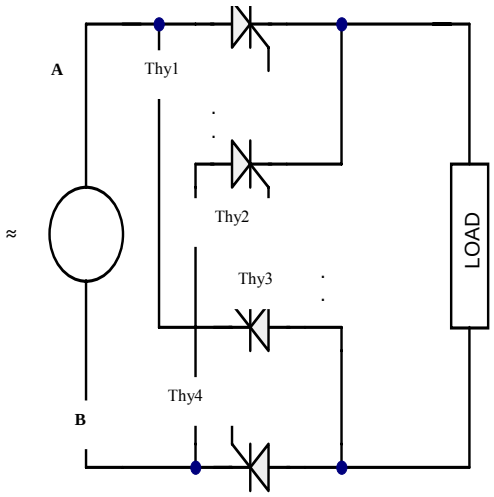


Fig. 4 Fully Controlled Single Phase Converter

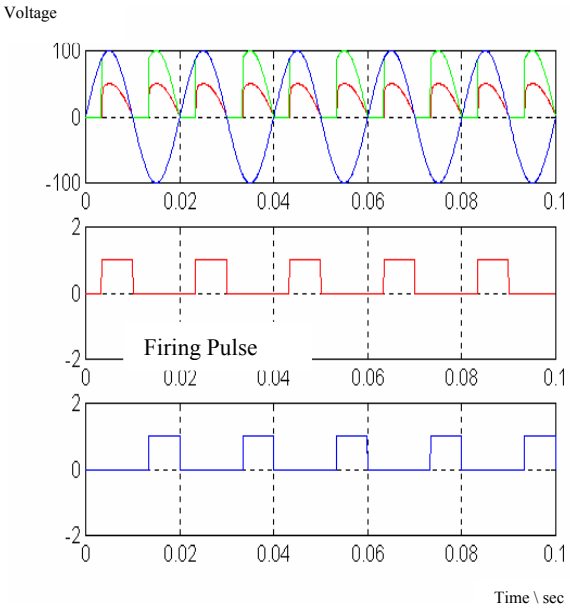


Fig. 5a Output voltage waveforms of signal phase fully controlled rectifier with resistive load for firing angle delays of 60°

Thyristors Thy1 and Thy4 are fired together when terminal A of the supply is positive while on the other half cycle when B is positive Thy2 and Thy3 are fired simultaneously. The output voltage waveforms are illustrated in Fig. 5. At every instant the load is either connected to the mains by the pair of switches Thy1 and Thy4 or it is connected by the pair of switches Thy2 and Thy3 or it is disconnected. It is much smoother than in the single pulse circuit, although again it is far from pure d.c.

The waveform shown in Fig. 5a corresponds to σ=60°, while Fig. 5b is for σ=120°. It is clear that the larger the delay angle the lower the output voltage.

The maximum output voltage (V_{do}) is obtained with σ=0, and given by:

$$V_{do} = \frac{2}{\pi}.\sqrt{2}.V_{rms} \quad (3)$$

Where V_{max} is the r.m.s voltage of the incoming mains. Obviously when σ is zero the output voltage is the same as it would be for an uncontrolled diode bridge rectifier, since the thyristors conduct for the whole of the half cycle for which they are forward biased. The variation of the mean d.c. voltage with σ is given by:

$$V_{dc} = \frac{1}{2}(1 + \cos \alpha).V_{do} \quad (4)$$

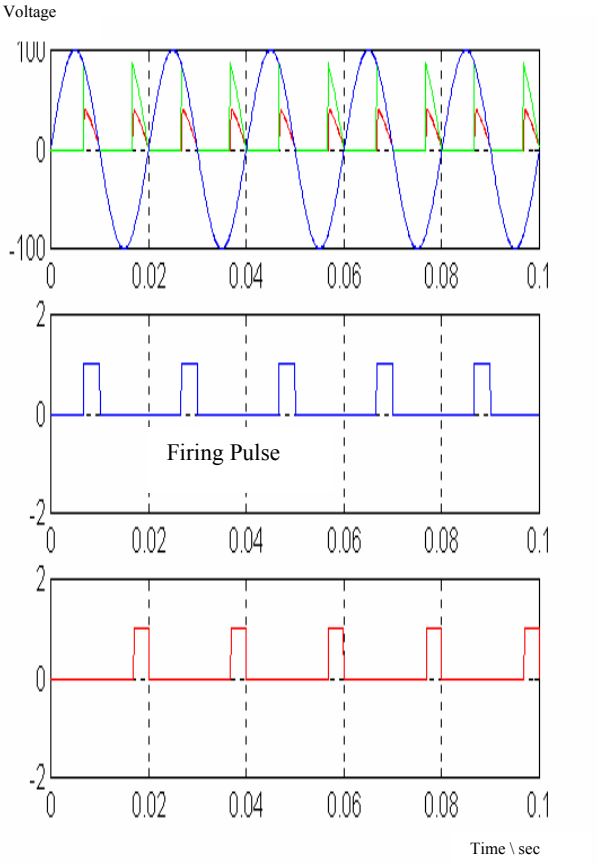


Fig 5b: Output voltage waveforms for resistive load for σ=120°

From which it can be seen that with a resistive load the d.c voltage can be varied from a maximum of V_{do} down to zero by varying σ from 0 to 180°.

B. Inductive Load

Motor loads are inductive and it is well known that the current cannot change instantaneously in an inductive load. Therefore the behaviour of the converter with an inductive load differs from that of a converter with a resistive load.

With a fixed σ, the mean output voltage with a resistive-inductive load is not the same as with a purely resistive load and therefore it is difficult to give a simple general formula for the mean output voltage in terms of σ.

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf

However, fortunately it transpires that the output voltage for a given σ does become independent of the load inductance once there is sufficient inductance to prevent the load current from falling to zero. This condition is known as continuous current and many motor circuits have sufficient self inductance to ensure that this condition is achieved. Under continuous current conditions the output voltage waveform only depends on the firing angle, and not on the actual value of the inductance present in the circuit. This makes matters straightforward and typical output voltage waveforms for this continuous current condition are illustrated in Fig. 6.

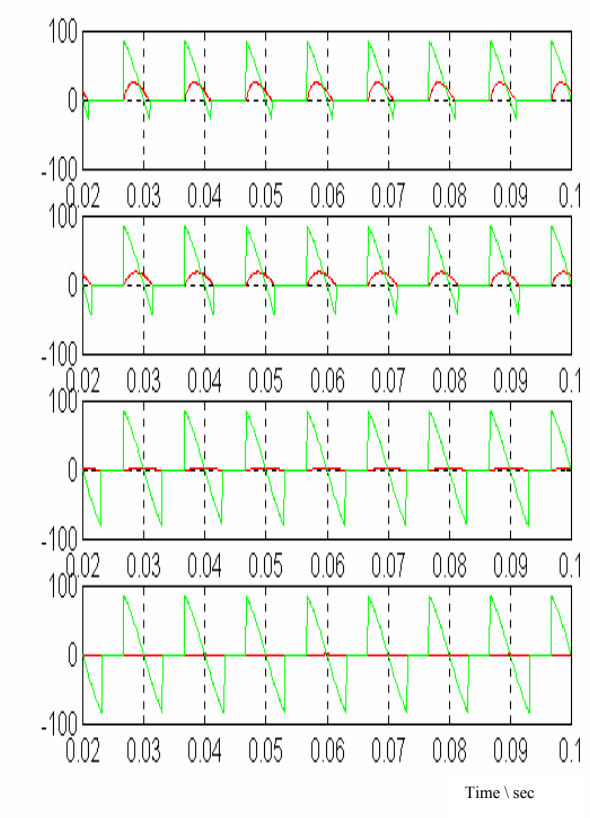
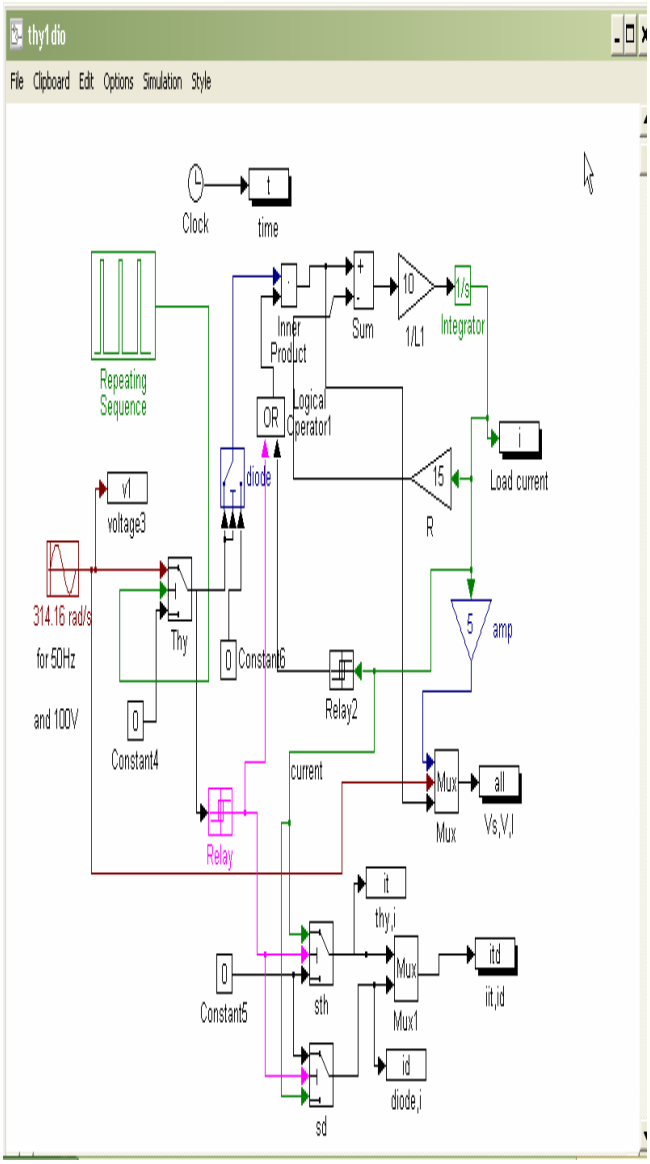


Fig. 6 Output voltage waveforms for fully controlled rectifier supplying an inductive load for σ=120°

VI. Conclusion

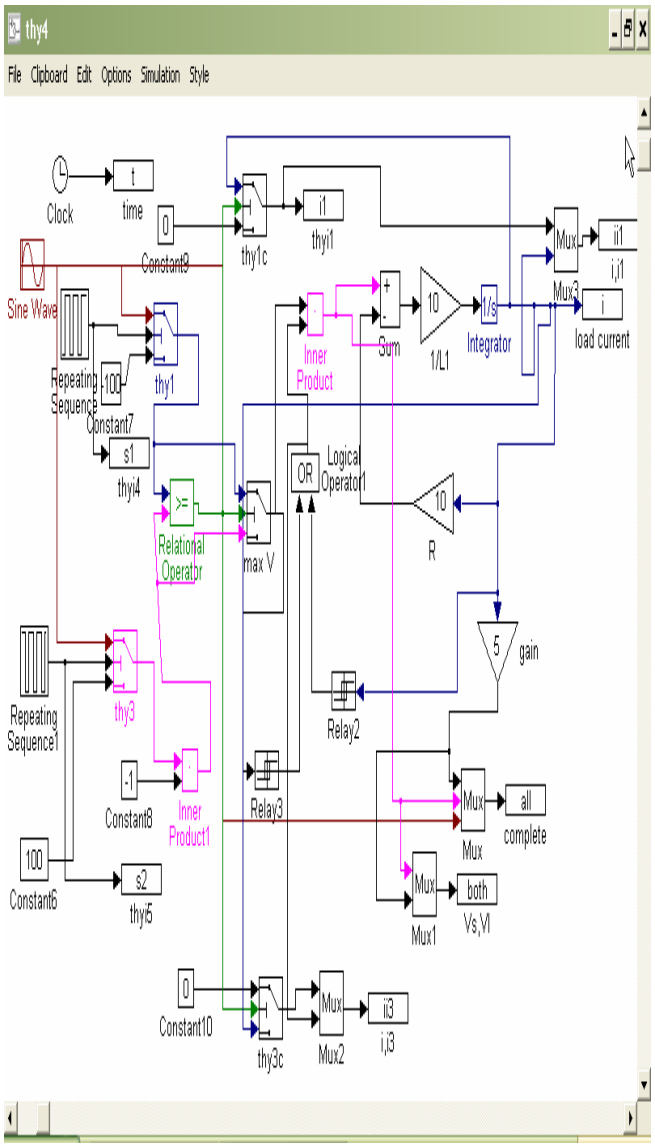
The transient analysis of power electronic circuits presents a case where any small amount of gain in speed in any aspect of the solution accumulates into a large saving in costly computer time due to the number of computations done. Methods have been suggested to increase the computational efficiency using the Matlab/Simulink package version 4.1 and 3.1. This is achieved by representing the thyristors based converter systems by a series of switches. The power of the simulation is evident in the clear presentation of the models in this paper.

Appendix 1



Model for single phase half wave controlled rectifying circuit with commutating diode

APPENDIX 2



Model for fully controlled single phase converter

REFERENCES

[1] Lavers J and Cheung R "Analysis of power electronic circuits with feedback control" IEEE Proc, B, Vol. 137, no. 4 1990, P213-222.
[2] Bimal K. Bose, "Recent Advances in Power Electronics" IEEE Trans. on power electronics, Vol. 7, January 1992, P1-12.
[3] Desai, N.R and Hong, K.V. "Application of PSPICE simulation software to the steady of optoelectronic integrated circuits and devices", IEEE Trans, on Education, Vol 36, no. 4, November 1993, P357-362.
[4] Rogers. G. "IEEE Computer Applications in Power Electronics". ISSN 0895-0156, Vol. 8, January 1995, P12-16.
[5] Cyril Lander. "Power Electronics". 2nd edition, McGraw Hill Book 1987, P37-49.

Open Science Index, Electrical and Computer Engineering Vol:3, No:1, 2009 publications.waset.org/2520/pdf