

MODELING AND SIMULATION OF SINGLE PHASE INVERTER
WITH PWM USING MATLAB/SIMULINK

AZUAN BIN ALIAS

This thesis is submitted as partial fulfillment of the requirement for the
award of the Bachelor Degree Electrical Engineering (Power System)

Faculty of Electrical & Electronic Engineering
Universiti Malaysia Pahang

NOVEMBER, 2007

DECLARATION

I declare that this thesis entitled “*Modeling and simulation of single phase inverter with PWM using MATLAB/Simulink*” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :
Author : AZUAN BIN ALIAS
Date : 22 NOVEMBER 2007

DEDICATION

Specially dedicate to
My beloved parents, sisters and brothers

ACKNOWLEDGEMENT

Alhamdulillah, the highest thank to God because with His Willingness I possible to complete the final year project.

I would like to thank my supervisor Mr. Muhamad Zahim bin Sujod for his advice and support throughout this project. At the same time I would like to express my gratitude to Mr. Fadhil Bin Abas for sharing his valuables ideas as well as his knowledge.

I also wish acknowledgement to the people who gives support direct or indirectly to the project and during the thesis writing. Once again, thank you very much.

ABSTRACT

This project is about modeling and simulation of single phase Pulse Width Modulation (PWM) inverter. The model was implemented using MATLAB/Simulink with the SimPowerSystems Block Set. The Insulated Gate Bipolar Transistor (IGBT) model was used as switching device. This project is purposed to use MATLAB/Simulink software to design, analysis and evaluation of power electronic converter and their controllers. Besides, it can show what differential in simulation of this software with others. For modeling, Simulink provides a graphical user interface (GUI) for building model as block diagram, using click-and-drag mouse operation. Simulink includes a comprehensive block library of sink, sources, linear and nonlinear components and connectors. We also can customize and create our own block. After a model is defined, it can simulate, using a choice of integration methods, either from Simulink menus or by entering command in MATLAB's command window. In addition, the parameter can be changed and immediately see what happen for 'what if' exploration. In inverter full bridge inverter circuit, an AC output is synthesized from a DC input by closing and opening the switches in appropriate sequence or switching scheme. For that, the Pulse Width Modulation technique is used in control the closing and opening switches. The switching scheme applied is unipolar. The PWM signal is used to control ON/OFF switching state of the IGBTs will functions in driver model that created to control the switching scheme. Then, the simulation is made from the inverter model in Simulink. The output voltage was obtained from Simulink and Pspice. At the end of this project, the results from simulation were compared between Simulink and Pspice.

ABSTRAK

Projek ini adalah mengenai mereka bentuk dan simulasi satu litar penukar arus ulang alik menggunakan teknik 'modulasi keluasan denyut nadi'. Litar ini telah diimplementasi menggunakan program MATLAB/Simulink dan 'PowerSystemBlock Set'. IGBT telah digunakan sebagai suis. Projek ini bertujuan menggunakan program MATLAB/Simulink untuk mereka bentuk, menganalisis dan menilai penukar elektronik kuasa dan pengawalan. Selain itu, ia bertujuan untuk membezakan program ini dengan program yang lain. Untuk mereka bentuk, Simulink menyediakan GUI untuk membina model sebagai blok diagram melalui cara 'klik dan tarik' yang menggunakan operasi tetikus. Simulink menyediakan 'blok perpustakaan' untuk 'sink, sources, linear and nonlinear' bagi komponen dan penyambung. Ia juga boleh digunakan untuk mereka bentuk blok sendiri. Selepas model dikenal pasti, simulasi dilakukan menggunakan dua pilihan sama ada menggunakan menu pada Simulink atau memasukkan arahan dalam tettingkap arahan MATLAB. Selain itu, parameter boleh diubah serta merta untuk melihat perubahan yang berlaku pada simulasi. Dalam litar penukar arus ulang alik, keluaran AC diproses dari DC oleh pembukaan dan penutupan suis dalam susunan yang sesuai atau skim pemsuisan. Untuk itu, teknik modulasi keluasan denyut nadi digunakan untuk mengawal pembukaan dan penutupan suis. Skim pemsuisan yang digunakan ialah 'unipolar'. Isyarat modulasi keluasan denyut nadi digunakan untuk mengawal keadaan 'ON/OFF' pemsuisan oleh IGBT yang berfungsi dalam model pemacu untuk mengawal skim pemsuisan. Kemudian, simulasi dilakukan daripada model dalam Simulink. Keluaran voltan diperoleh daripada simulasi Simulink dan Pspice. Pada akhir projek, keputusan daripada simulasi dibandingkan antara Simulink dan Pspice.

TABLE OF CONTENTS

TITLE	i
DECLARATION	ii
DEDICATION	iii
ACKNOWLEDGEMENT	iv
ABSTRACT	v
ABSTRAK	vi
TABLE OF CONTENT	vii
LIST OF CHAPTERS	viii
LIST OF TABLES	x
LIST OF FIGURES	xi
LIST OF ABBREVIATION	xiii
LIST OF APPENDIX	xiv

LIST OF CHAPTERS

CHAPTER	TITLE	PAGE
1	INTRODUCTION	
	1.1 Background	1
	1.2 Project objective	2
	1.3 Project scope	2
	1.4 Thesis outline	3
2	LITERATURE REVIEW	
	2.1 MATLAB/Simulink	4
	2.2 Blockset Power Systems	6
	2.2.1 Sophiscated Blocks Manage Diagram Interpretation	7
	2.2.2 Simulate It	8
	2.2.3 Powerful Analysis Method	8
	2.2.4 Area of Power System Block Set	9
	2.3 Inverter	12
	2.4 Pulse Width Modulation	14
	2.5 IGBT	19
	2.5.1 Characteristic of Bipolar, IGBT and MOSFET	19
	2.6 Snubber	22
	2.6.1 Function	23
	2.6.2 Operation	24

3	METHODOLOGY	
	3.1 Overall system design	29
	3.2 Inverter circuit design	30
	3.2.1 Circuit design in MATLAB/Simulink	31
	3.2.2 Circuit design in Pspice	32
	3.3 Control Switch and driver circuit	33
	3.3.1 Design in Simulink	33
	3.3.2 Design in Pspice	35
	3.4 Modulation Index Calculation	36
4	RESULT AND ANALYSIS	
	4.1 Simulink's and Pspice result	37
	4.2 Application of PWM Control	43
	4.3 PWM inverter analysis	46
5	CONCLUSION AND RECOMMENDATION	
	5.1 Conclusion	48
	5.2 Recommendation	49
	5.3 Commercialization	49
	REFERENCE	50
	APPENDIX A	53

LIST OF TABLES

TABLE NO.	TITLE	PAGE
1	Differential between IGBT and MOSFET	21
2	The types of snubbers circuit	27
3	Switching sequence	30
4	Parameter for inverter circuit	34
5	Modulation index with different reference frequency	41
6	Output voltage (Simulink vs Pspice)	43
7	Parameter for PWM inverter analysis	47
8	Normalized Fourier Coefficients for Unipolar PWM	47
9	Fourier Series Quantities For PWM Inverter	48

LIST OF FIGURES

FIGURE	TITLE	PAGE
2.1	Library Browser for Simulink	5
2.2	Window for model using functional block	6
2.3	Library Browser for SimPower System	11
2.4	Basic inverter diagram	13
2.5	Full-bridge converter for unipolar PWM	16
2.6	Producing unipolar PWM output	17
2.7	Duty cycle for 10% of PWM output	17
2.8	NPT IGBT cross section	20
2.9	Typical MOSFET cross section	20
2.10	Where MOSFETs and IGBTs are preferred, not counting output power	21
2.11	Snubber circuit at IGBT switch	22
2.12	The switching trajectory for inductive load with and without snubber	23
2.13	Differentials of FBSOA and RBSOA	25
2.14	The switching trajectory for resistive load	26
2.15	The switching trajectory for inductive load	26
3.1	Block diagram of the system	29
3.2	Inverter circuit and switching sequence	30
3.3	Inverter Circuit in Simulink	31
3.4	Inverter Circuit in PSpice	32
3.5	IGBT switch circuit	33

3.6	Driver circuit in Simulink	34
3.7	Driver circuit in PSpice	35
4.1	Driver output in Simulink	37
4.2	Driver output in PSpice	38
4.3	Result of simulation of R load with modulation index, $mi = 0.1$	39
4.4	Result of simulation of R load with modulation index, $mi = 0.8$	40
4.5	Graph MI versus Output Voltage (RMS)	41
4.6	Simulink's simulation result of output voltage with $mi = 0.6$	42
4.7	Pspice's simulation result of output voltage with $mi = 0.6$	42
4.8	Graph Simulink versus PSpice (Vrms)	43
4.9	Application for PWM control in 3-phase machine	44
4.10	Simulation results for 3-phase machine by verify modulation index	45
4.11	Modulation Index versus magnitude voltage (Vrms)	46

LIST OF ABBREVIATION

DC	-	Direct Current
AC	-	Alternate Current
PWM	-	Pulse Width Modulation
IGBT	-	Insulated Gate Bipolar Transistor
MOSFET	-	Metal Oxide Field Effect Transistor

LIST OF APPENDIX

APPENDIX	TITLE	PAGE
A	Datasheet for IRGBC20F	53

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: MODELING AND SIMULATION OF SINGLE PHASE INVERTER
WITH PWM USING MATLAB/SIMULINK

SESI PENGAJIAN: 2007/2008

Saya AZUAN BIN ALIAS (850527-11-5033)
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

☐

SULIT

(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)

☐

TERHAD

(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☒

TIDAK TERHAD

Disahkan oleh:

(TANDATANGAN PENULIS)

(TANDATANGAN PENYELIA)

Alamat Tetap:

**NO 307, KG RAHMAT,
FELDA CHALOK,
21450 BANDAR PERMAISURI,
TERENGGANU.**

MUHAMAD ZAHIM BIN SUJOD

(Nama Penyelia)

Tarikh: **22 NOVEMBER 2007**

Tarikh: : **22 NOVEMBER 2007**

- CATATAN:
- * Potong yang tidak berkenaan.
 - ** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.
 - ♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

“I hereby acknowledge that the scope and quality of this thesis is qualified for the award
of the Bachelor Degree of Electrical Engineering (Power System)”

Signature : _____
Name : MUHAMAD ZAHIM BIN SUJOD
Date : 22 NOVEMBER 2007

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

This project is focus on modeling and simulation of single phase inverter as a frequency changer modulated by Sinusoidal Pulse Width Modulation (SPWM). An inverter is a circuit that converts DC sources to AC sources. Pulse Width Modulation is a technique that use as a way to decrease total harmonic distortion in inverter circuit.

The model is implemented using MATLAB/Simulink software with the SimPower System Block Set based on computer simulation. Computer simulation plays an important role in the design, analysis, and evaluation of power electronic converter and their controller. MATLAB is an affective tool to analyze a PWM inverter. Advantages of using MATLAB are the following: faster response, availability of various simulation tools and functional blocks and the absence of convergence problems.

Safe-commutation strategy want be implemented is to solve switching transients. So, Insulated Gate Bipolar Transistor (IGBT) is use as switching devices. IGBT is preferable because it is easy to control and low losses.

The result from Simulink was verified using Pspice simulation prior to experimental verifications.

1.2 PROJECT OBJECTIVE

1. To design an inverter model by using MATLAB/Simulink and making analysis on the output voltage.
2. To study the function of PWM in single phase inverter.
3. To make comparison of the output waveform between MATLAB/Simulink & Pspice.

1.3 PROJECT SCOPE

1. Modeling and simulation using MATLAB/Simulink and Pspice.
2. Using PWM method for the switching operation.

1.4 THESIS OUTLINE

Chapter 1 explains the operation of an inverter and advantages of PWM method. The overview of project objectives and project scopes also discuss in this chapter.

Chapter 2 focuses on the literature review that related to this project. MATLAB/Simulink software, Power System Block Set, inverter circuit, Pulse Width Modulation (PWM), Insulated Gate Bipolar Transistor (IGBT) and snubber.

Chapter 3 discusses about methodology of this project. This chapter also discuss about circuit design and the system work.

Chapter 4 explains and discusses all the results obtained and the analysis of the project. The comparisons of simulation's results are made between Simulink and Pspice.

Chapter 5 discusses the conclusion of advantages of the method had been implemented into the project. This chapter also gives the recommendation about the future development of the project by using the software which used in this project.

CHAPTER 2

LITERATURE REVIEW

2.1. MATLAB/SIMULINK

Simulink is a software package for modeling, simulating, and analyzing dynamical systems. It supports linear and nonlinear systems, modeled in continuous time, sampled time, or a hybrid of the two. Systems can also be multi rate, i.e., have different parts that are sampled or updated at different rates [2].

For modeling, Simulink provides a graphical user interface (GUI) for building models as block diagrams, using click-and-drag mouse operations. With this interface, you can draw the models just as you would with pencil and paper (or as most textbooks depict them). This is a far cry from previous simulation packages that require you to formulate differential equations and difference equations in a language or program. Simulink includes a comprehensive block library of sinks, sources, linear and nonlinear components, and connectors. You can also customize and create your own blocks [2].

Models are hierarchical, the models are built using both top-down and bottom-up approaches the system can viewed at a high level, then double-click on blocks to go

down through the levels to see increasing levels of model detail. This approach provides insight into how a model is organized and how its parts interact [2].

After a model is defined, it can simulate, using a choice of integration methods, either from the Simulink menus or by entering commands in MATLAB's command window. The menus are particularly convenient for interactive work, while the command-line approach is very useful for running a batch of simulations. Using scopes and other display blocks, the simulation results can be seen while the simulation is running. In addition, the parameters can be changed and immediately see what happens, for "what if" exploration. The simulation results can be put in the MATLAB workspace for post processing and visualization [2].

Model analysis tools include linearization and trimming tools, which can be accessed from the MATLAB command line, plus the many tools in MATLAB and its application toolboxes. And because MATLAB and Simulink are integrated, you can simulate, analyze, and revise your models in either environment at any point [2].

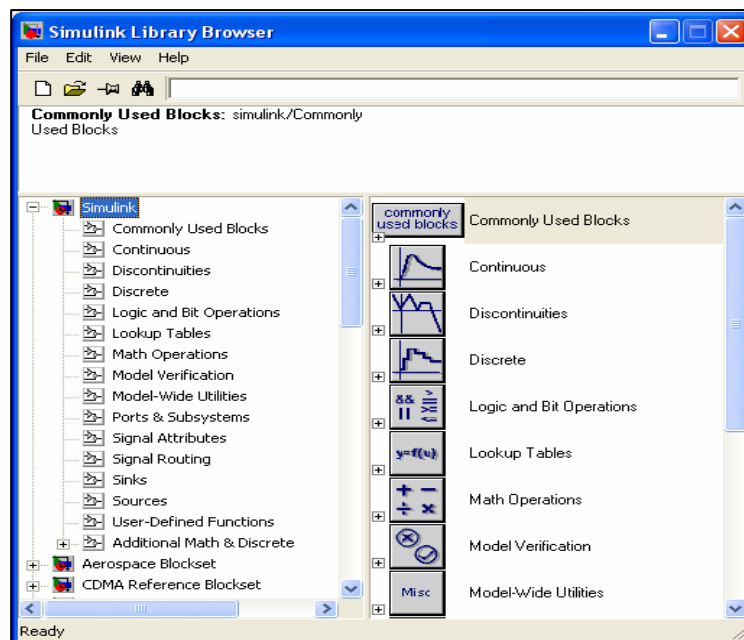


Figure 2.1: Library Browser for Simulink

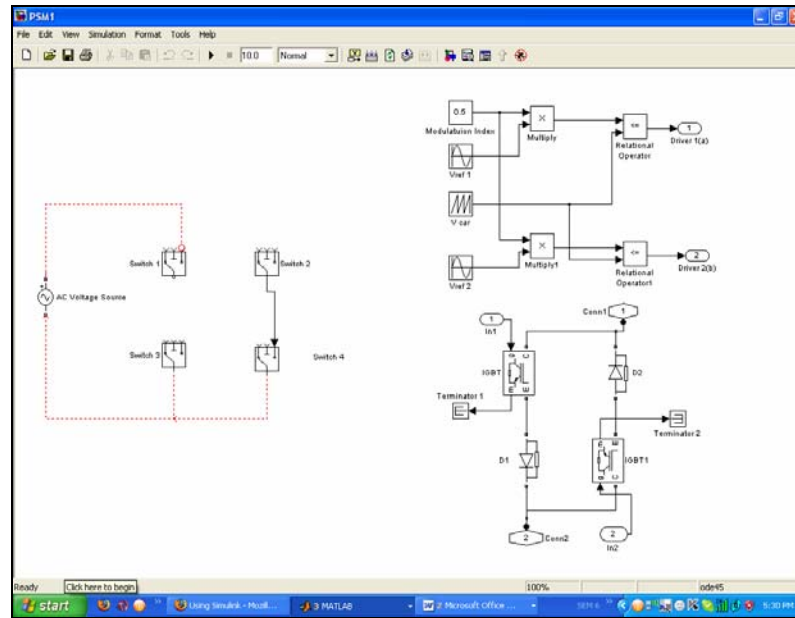


Figure 2.2: Window for model using functional block

2.2 BLOCKSET POWER SYSTEM

The Power System Block Set or SimPower System (after renamed) had just introduced to the modeling environment of Simulink by Mathworks. The Power System Block Set provides tools for modeling and simulating electrical power systems within Simulink using the standard notations for electrical circuits. Its block library contains blocks that represent standard components found in electrical power networks. It is easily to incorporate electrical systems and controllers into complex system models [3].

2.2.1 SOPHISTICATED BLOCKS MANAGE DIAGRAM INTERPRETATION

The blocks in the Power System Block Set provide methods that interpret diagram connectivity in a topological manner. This capability allows us to model electrical systems without worrying about the directionality of signals and currents. You can create electrical Simulink diagrams on the computer just as you would on paper [3].

The Power System Block Set allows scientists and engineers to build models that simulate power systems. The blockset uses the Simulink environment, allowing a model to be built using click and drag procedures. Not only can the circuit topology be drawn rapidly, but the analysis of the circuit can include its interactions with mechanical, thermal, control, and other disciplines. This is possible because all the electrical parts of the simulation interact with Simulink's extensive modeling library. Because Simulink uses MATLAB as the computational engine, MATLAB's toolboxes can also be used by the designer [4].

The blockset libraries contain models of typical power equipment such as transformers, lines, machines, and power electronics. These models are proven ones coming from textbooks, and their validity is based on the experience of the Power Systems Testing and Simulation Laboratory of Hydro-Quebec, a large North American utility located in Canada. The capabilities of the blockset for modeling a typical electrical grid are illustrated in demonstration files [4].

2.2.2 SIMULATE IT

The Power System Blockset is completely integrated with Simulink at the block level. Combining Power System and other Simulink blocks creates a unique environment for multi-domain modeling and controller design. This environment allows the combination of electrical, power-electronic, mechanical, hydraulic, and other systems models [3].

For time-domain simulation, the Power System Blockset takes advantage of Simulink's powerful variable-step integrators and zero-crossing detection capabilities to produce highly accurate simulations of power system models. In addition, you have access to all of the block building and masking features, allowing you to build more complex components from electrical primitives [3].

2.2.3 POWERFUL ANALYSIS METHODS

The new blockset contains features that allow you to specifically analyze the electrical portions of your Simulink model. You can extract the linear portions of the electrical model into the MATLAB workspace as a state-space system for further analysis. This capability is useful if, for example, you want to examine the impedance frequency characteristics of your system [3].

You can work with the electrical state-space system using any of the tools from The MathWorks. In developing a control system for an electrical subsystem, for example, you can use one of the Control System Toolboxes to design controllers around the plant model you have extracted from the Simulink diagram [3].

The blockset also provides a GUI that allows you to interact directly with the blocks and states of the electrical system. You can set the state variables of the electrical system through the GUI. For example, you may want to set the states of the system (voltages or currents) to simulate some transients. The GUI also provides the steady-state values of voltage and current so that you can return to a steady-state simulation easily for other types of analysis [3].

Finally, to aid in working with the blockset library of machines and drives, the Power System Block Set provides a function that performs load flow analysis. Load flow analysis is done to balance the electrical and mechanical loads between machines in your model [3].

2.2.4 AREA OF THE POWER SYSTEM BLOCK SET

Power system networks

- RLC branches and loads
- Pi section lines
- Linear and saturable transformers/td
- Surge arresters
- Breakers
- Mutual inductances
- Distributed parameter lines
- AC voltage and current source
- DC voltage sources

Electric machinery

- Complete and simplified models of synchronous machines
- Asynchronous machines
- Permanent magnet synchronous machines
- Excitation systems
- Hydraulic turbines
- Governors

Power electronics

- Diodes
- Simplified and complex thyristors
- GTOs
- Switches
- MOSFETs
- IGBT

Control and measurement blocks

- Voltage and current measurements
- RMS measurements
- Active power calculations
- Timers
- Synchronized 6-pulse generators

Triphase library

- Triphase loads and branches
- Pi section lines
- AC voltage sources
- 6-pulse thyristor bridges
- Diode rectifiers
- Triphase transformers in Y-delta, Y-Y, and delta-Y configurations

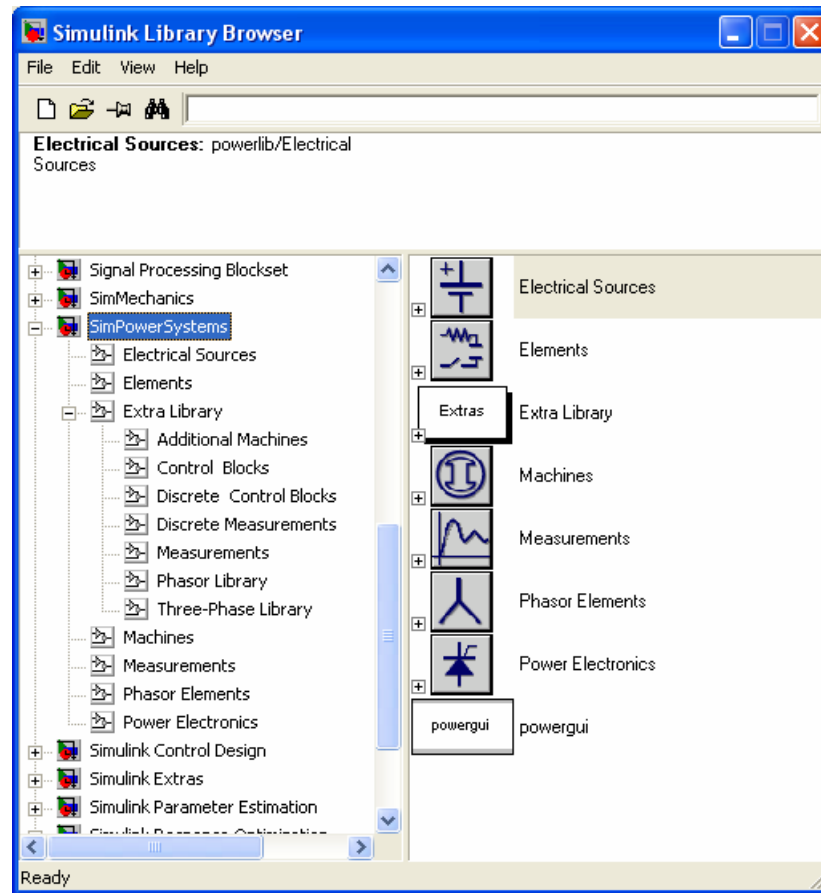


Figure 2.3: Library Browser for SimPower System

2.3 INVERTER

Inverter or power inverter is a device that converts the DC sources to AC sources. Inverters are used in a wide range of applications, from small switched power supplies for a computer to large electric utility applications to transport bulk power. This makes them very suitable for when you need to use AC power tools or appliances [5].

Power inverters produce one of three different types of wave output:

- Square Wave
- Modified Square Wave (Modified Sine Wave)
- Pure Sine Wave (True Sine Wave)

The three different wave signals represent three different qualities of power output. Square wave inverters result in uneven power delivery that is not efficient for running most devices [6].

Square wave inverters were the first types of inverters made and are obsolete. Modified square wave (modified sine wave) inverters deliver power that is consistent and efficient enough to run most devices fine. Some sensitive equipment requires a sine wave, like certain medical equipment and variable speed or rechargeable tools [6].

Modified Sine Wave

Modified sine wave or quasi-sine wave inverters were the second generation of power inverter. They are a considerable improvement over square wave inverters. These popular inverters represent a compromise between the low harmonics (a measure of waveform quality) of a true sine wave inverter and the higher cost and lower efficiency of a true sine wave inverter [7].

Modified sine wave inverters approximate a sine wave and have low enough harmonics that they do not cause problems with household equipment. They runs TV's, stereos, induction motors (including capacitor start), universal motors, computers, microwave, and more quite well. The main disadvantage of a modified sine wave inverter is that the peak voltage varies with the battery voltage. Inexpensive electronic devices with no regulation of their power supply may behave erratically when the battery voltage fluctuates [7].

True Sine Wave

True sine wave inverters represent the latest inverter technology. The waveform produced by these inverters is the same as or better than the power delivered by the utility. Harmonics are virtually eliminated and all appliances operate properly with this type of inverter. They are, however, significantly more expensive than their modified sine wave cousins [7].

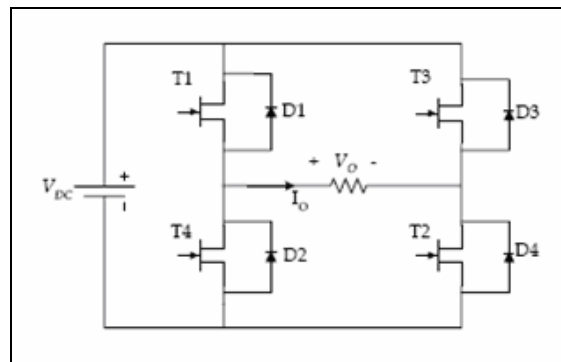


Figure 2.4: Basic inverter diagram

2.4 PULSE WIDTH MODULATION

Pulse width modulation (PWM) is a powerful technique for controlling analog circuits with a processor's digital outputs [8]. The applications of PWM are wide variety used like ranging from measurement and communications to power control and conversion.

PWM provides a way to decrease the Total Harmonic Distortion (THD) of load current. The THD requirement can be met more easily when the output of PWM inverter is filtering. The unfiltered PWM output will have a relatively high THD, but the harmonic will be at the much higher frequencies than for a square wave, making filtering easily [8].

The total harmonic distortion, or THD, is defined as the ratio of the sum of the powers of all harmonic components to the power of the fundamental [8].

In analog circuit, analog signal has a continuously varying value, with infinite resolution in both time and magnitude. As an example of an analog device for 5 volt battery, output voltage is not precisely 5V, changes over time, and can take any real-numbered value. The amount of current drawn from a battery is not limited to a finite set of possible values. Analog signals are distinguishable from digital signals because the latter always take values only from a finite set of predetermined possibilities, such as the set $\{0V, 5V\}$ [9].

Analog voltages and currents can be used to control things directly, like the volume of a car radio. In a simple analog radio, a knob is connected to a variable resistor. As the knob is turned, the resistance goes up or down. As that happens, the current flowing through the resistor increases or decreases. This changes the amount of

current driving the speakers, thus increasing or decreasing the volume. An analog circuit is one, like the radio, whose output is linearly proportional to its input [9].

The analog control system it is not always economically attractive or otherwise practical. Analog circuits tend to drift over time and can, therefore, be very difficult to tune. Precision analog circuits, which solve that problem, can be very large, heavy, and expensive. Analog circuits can also get very hot; the power dissipated is proportional to the voltage across the active elements multiplied by the current through them. Analog circuitry can also be sensitive to noise. Because of its infinite resolution, any perturbation or noise on an analog signal necessarily changes the current value [9].

If the analog circuit is controlled digitally, the system costs and power consumption can be drastically reduced. Besides, there are many [microcontrollers](#) and [DSPs](#) already include on-chip PWM controllers, making implementation easy [9].

PWM is a way of digitally encoding analog signal levels. The duty cycle of a square wave is modulated to encode a specific analog signal level by using the high-resolution counter. The PWM signal is still digital because, at any given instant of time, the full DC supply is either fully on or fully off. The voltage or current source is supplied to the analog load by means of a repeating series of on and off pulses. The on-time is the time during which the DC supply is applied to the load, and the off-time is the period during the supply is switched off. Given a sufficient bandwidth, any analog value can be encoded with PWM [9].

Control of the switches for the sinusoidal PWM output requires a reference signal (modulating or control signal) which is a sinusoidal wave and a carrier signal which a triangular wave that control the switching frequency. There two type of the switching for PWM, unipolar switching and bipolar switching.

In a unipolar switching scheme for PWM, the output is switched from either high to zero or low to zero, rather than between high and low as in bipolar switching. The unipolar scheme has switch control as follow [8]:

S_1 is on when $V_{\text{sine}} > V_{\text{tri}}$

S_2 is on when $-V_{\text{sine}} < V_{\text{tri}}$

S_3 is on when $-V_{\text{sine}} > V_{\text{tri}}$

S_4 is on when $V_{\text{sine}} < V_{\text{tri}}$

Another unipolar switching scheme has only one pair of switches operating at the carrier frequency while the other pair operates at reference frequency, thus having two high-frequency switches and two low-frequency switches [8].

S_1 is on when $V_{\text{sine}} > V_{\text{tri}}$ (high frequency)

S_4 is on when $V_{\text{sine}} < V_{\text{tri}}$ (high frequency)

S_2 is on when $V_{\text{sine}} > 0$ (low frequency)

S_3 is on when $V_{\text{sine}} < 0$ (low frequency)

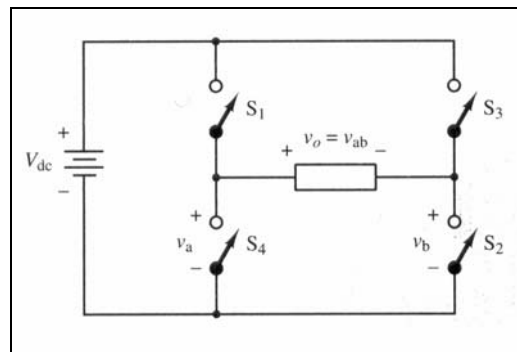


Figure 2.5: Full-bridge converter for unipolar PWM

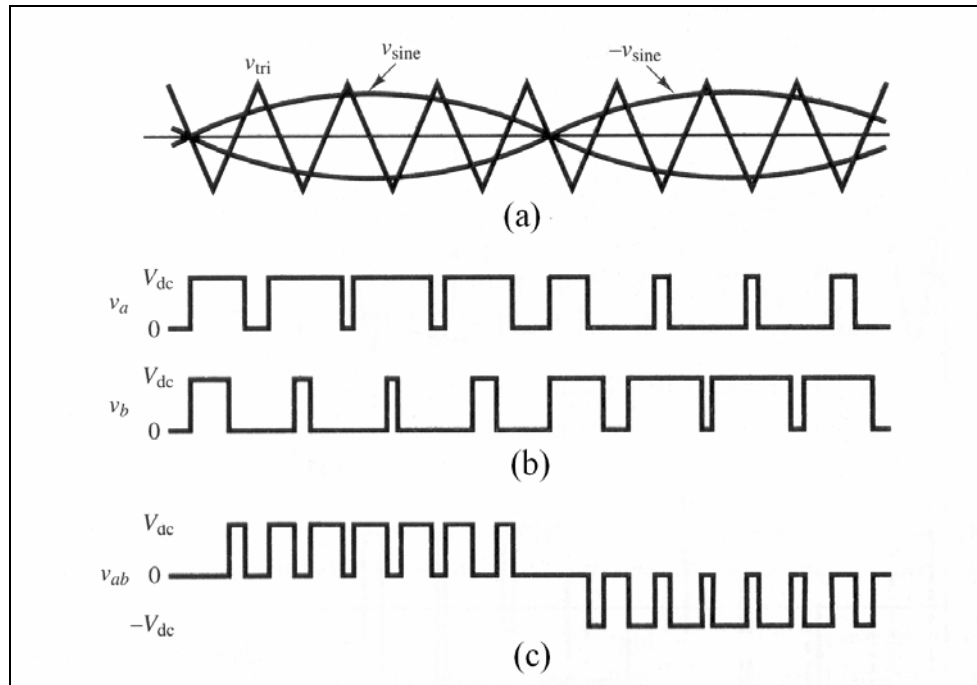


Figure 2.6: Producing unipolar PWM output (a) Reference and carrier signals (b) bridge voltage v_a and v_b (c) output voltage.

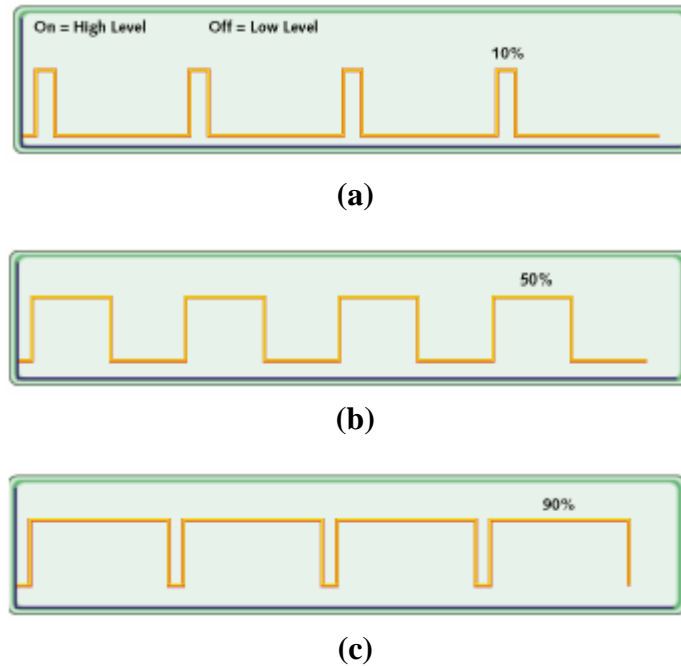


Figure 2.7: PWM output (a) 10% of duty cycle (b) 50% duty cycle (c) 90% duty cycle

Figure 2.7(a) shows a PWM output at a 10% duty cycle. That is, the signal is on for 10% of the period and off the other 90%. Figures 2.7(b) and 2.7(c) show the PWM outputs at 50% and 90% duty cycles, respectively. These three PWM outputs encode three different analog signal values, at 10%, 50%, and 90% of the full strength. If, for example, the supply is 5V and the duty cycle is 10%, a 0.5V analog signal results [9].

The benefit of choosing the PWM over analog control is increased noise immunity which the PWM is sometimes used for communication. Switching from an analog signal to PWM can increase the length of a communications channel dramatically. At the receiving end, a suitable RC (resistor-capacitor) or LC (inductor-capacitor) network can remove the modulating high frequency square wave and return the signal to analog form. So, the filter requirement can be reduced and the overall inverter size can be reduced.

The disadvantages of PWM are like more complex circuit for the switching, higher switching loss due more to frequent switching, difficult to implement and more Electro Magnetic Interference (EMI) loss.

2.5 IGBT

2.5.1 CHARACTERISTIC OF BIPOLAR, IGBT AND MOSFET

Insulated Gate Bipolar Transistors (IGBTs) combine the best of conventional bipolar transistors and FETs. Like FETs, they only require a voltage across the base in order to conduct. However, like conventional bipolars, they are efficient conductors of current through their collector/emitters [11].

The bipolar transistor requires a high base current to turn on, has relatively slow turn-off characteristics (known as current tail), and is liable for thermal runaway due to a negative temperature co-efficient. In addition, the lowest attainable on-state voltage or conduction loss is governed by the collector-emitter saturation voltage $V_{CE(SAT)}$.

The MOSFET, however, is a device that is voltage- and not current-controlled. MOSFETs have a positive temperature coefficient, stopping thermal runaway. The on-state-resistance has no theoretical limit; hence on-state losses can be far lower. The MOSFET also has a body-drain diode, which is particularly useful in dealing with limited free wheeling currents [11].

The IGBT has the output switching and conduction characteristics of a bipolar transistor but is voltage-controlled like a MOSFET. In general, this means it has the advantages of high-current handling capability of a bipolar with the ease of control of a MOSFET. However, the IGBT still has the disadvantages of a comparatively large current tail and no body drain diode [11].

Another potential problem with some IGBT types is the negative temperature co-efficient, which could lead to thermal runaway and makes the paralleling of devices hard to effectively achieve. This problem is now being addressed in the latest

generations of IGBTs that are based on “non-punch through” (NPT) technology. This technology has the same basic IGBT structure (see Figure 2.8) but is based on bulk-diffused silicon, rather than the epitaxial material that both IGBTs and MOSFETs have historically used.

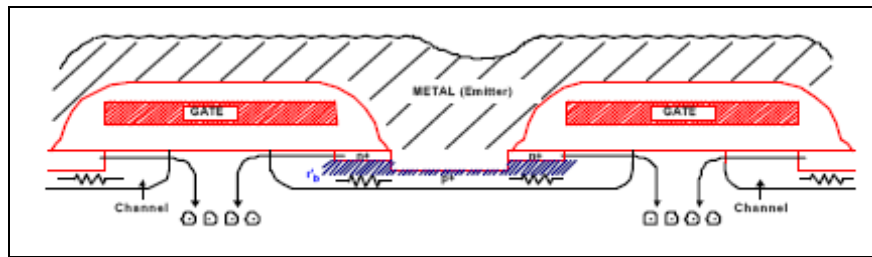


Figure 2.8: NPT IGBT cross section

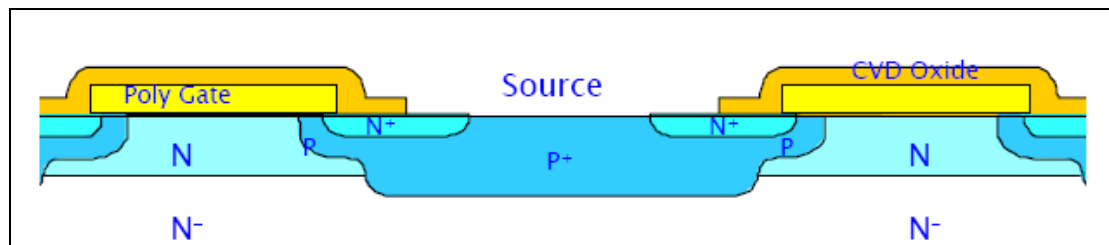


Figure 2.9: Typical MOSFET cross section

Comparing from Figure 2.8 and 2.9, the MOSFET and IGBT structures look very similar. The basic difference is the addition of a p substrate beneath the n substrate. The IGBT technology is certainly the device of choice for breakdown voltages above 1000V, while the MOSFET is certainly the device of choice for device breakdown voltages below 250V [11].

Choosing between IGBTs and MOSFETs is very application-specific and cost, size, speed and thermal requirements should all be considered [11].

Figure 2.10 shows some of the boundaries where it is fairly clear as to what is preferred, the MOSFET or IGBT, the specifics are further detailed below [11].

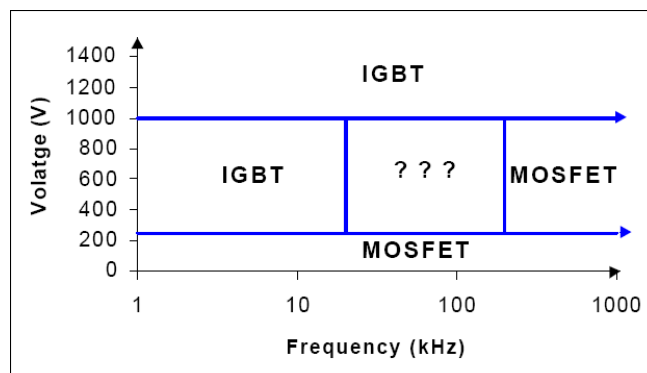


Figure 2.10: Where MOSFETs and IGBTs are preferred, not counting output power.

Table 1: Differential between IGBT and MOSFET

IGBT	MOSFET
Preferred conditions: <ul style="list-style-type: none"> ➤ Low duty cycle ➤ Low frequency (<20 kHz) ➤ Narrow or small line or load variations ➤ High-voltage applications (>1000V) ➤ Operation at high junction temperature is allowed (>100°C) ➤ >5kW output power 	Preferred condition: <ul style="list-style-type: none"> ➤ High frequency applications (>200kHz) ➤ Wide line or load variations ➤ Long duty cycles ➤ Low-voltage applications (<250V) ➤ < 500W output power
Typical IGBT applications include: <ul style="list-style-type: none"> ➤ Motor control: Frequency <20 kHz, short circuit/in-rush limit protection ➤ Uninterruptible power supply (UPS): Constant load, typically low frequency ➤ Welding: High average current, low frequency (<50 kHz), ZVS circuitry ➤ Low-power lighting: Low frequency (<100 kHz) 	Typical MOSFET applications include: <ul style="list-style-type: none"> ➤ Switch mode power supplies (SMPS): Hard switching above 200kHz ➤ Switch mode power supplies (SMPS): ZVS below 1000 watts ➤ Battery charging

2.6 SNUBBER

A snubber is a circuit connected around a power device for the purpose of altering its switching trajectory. Snubbers usually have the objective of reducing power loss in the power device.

Snubber circuits are generally categorized into two basic categories:

- Turn-off snubber
- Turn-on snubber

They are also classified as:

- lossy snubber (RCD and RLD snubber)
- lossless snubber (energy recovery snubber)

Snubber circuits act to prevent fast change of voltage (turn-off snubber) or current (turn-on snubber) during switching, so that the commutation process can become more nearly linear.

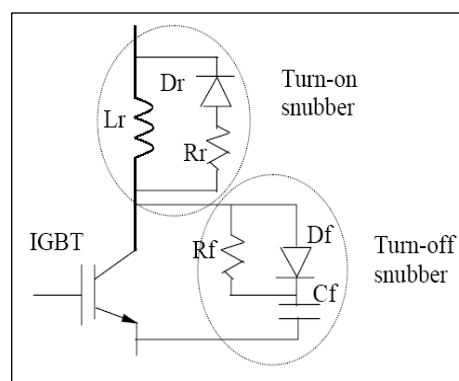


Figure 2.11: Snubber circuit at IGBT switch

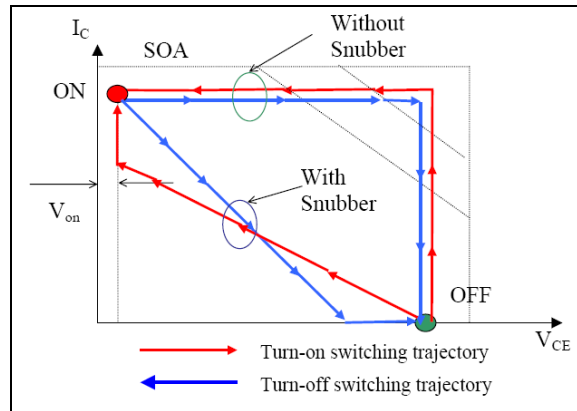


Figure 2.12: The switching trajectory for inductive load with and without snubber

The power absorbed by the power device is reduced by the snubber circuit. A large capacitor reduces the power loss in the power device, but at the expense of power loss in the snubber resistor.

The use of the snubber may reduce or may not reduce the total power loss, but perhaps more important, the snubber can reduce the losses in the power device and reduces the cooling requirement for the device.

The power device is more prone to failure and is harder to cool than the resistor, so the snubber makes the design more reliable.

2.6.1 FUNCTION

Power semiconductors are the heart of power electronics equipment. Snubbers are circuits which are placed across semiconductor devices for protection and to improve performance. Snubbers can do many things [13]:

- Reduce or eliminate voltage or current spikes
- Limit dI/dt or dV/dt
- Shape the load line to keep it within the safe operating area (SOA)
- Transfer power dissipation from the switch to a resistor or a useful load
- Reduce total losses due to switching
- Reduce EMI by damping voltage and current ringing

2.6.2 OPERATION

Switching Characteristics of Power Devices

Device utilization can be greatly improved by understanding the device switching characteristics.

The main performance switching characteristics of power devices:

- The safe operating area (SOA) of the device.
- The turn-on and turn-off switching transients.
- The switching trajectory.

The safe operating area (SOA)

The area within the output characteristics of the devices where it can be operated without failure due to power dissipation.

In general there are three boundaries for the SOA:

- The maximum operational voltage
- The maximum operational current (current limit)
- The boundary where both voltage and current become simultaneously large.

Two SOAs are defined for a power device:

- The Forward Bias SOA (FBSOA)
- The Reverse Bias SOA (RBSOA)

Different devices have different SOA.

SOA for IGBT

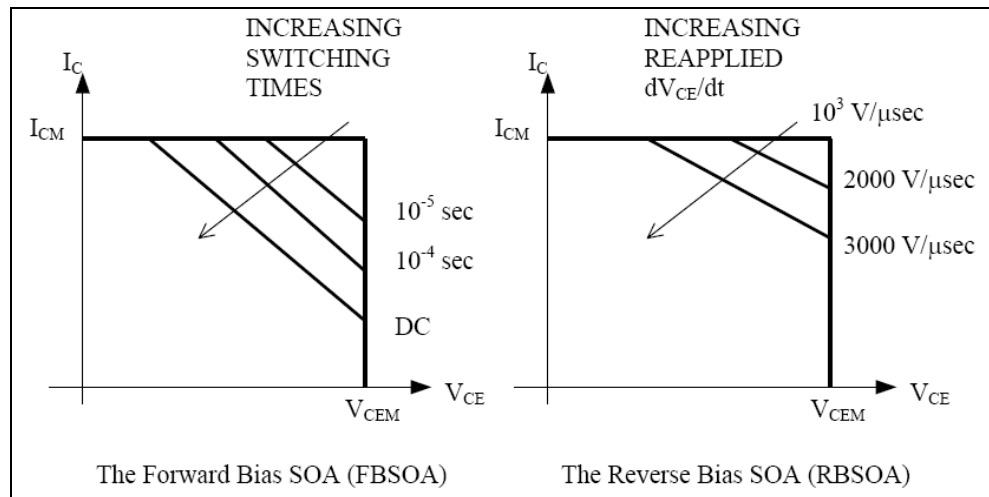


Figure 2.13 Differentials of FBSOA and RBSOA

The turn-on transient is subjected to FBSOA and the turn-off transient is subjected to RBSOA.

The FBSOA of IGBTs is square for short switching times, identical to the FBSOA in MOSFETs. Thus it has a minimal need for snubbers.

The higher rate dV_{CE}/dt (called reappplied rate) during turn-off flattens out the upper right corner in RBSOA.

The turn-on and turn-off switching transients

The real device requires a finite time to switch from on-state to off-state and vice versa. The voltage and current vary during this transition.

The product of voltage and current gives instantaneous power dissipated in the device.

There are three types of power loss in the device:

- The turn-on transition power loss

- The turn-off transition power loss
- The conduction loss

The power devices experience high voltages and currents during both turn-on and turn-off transients.

Parasitic effects (diode reverse recovery, parasitic device inductance) cause additional device voltage and current stresses.

Operating the device under this condition requires large and square SOA.

Snubber circuits shape the volt-amp switching trajectory of power devices.

Snubber circuit allows the utilization of power devices with less than perfect square SOA.

The Switching trajectory of power devices

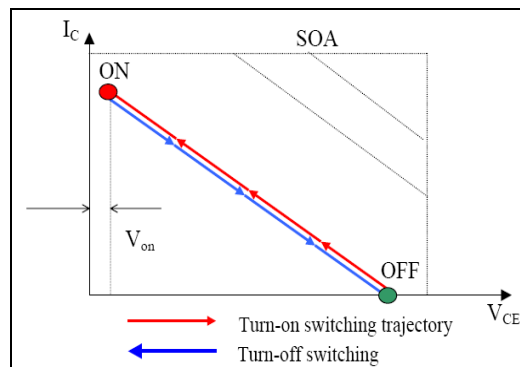


Figure 2.14: The switching trajectory for resistive load

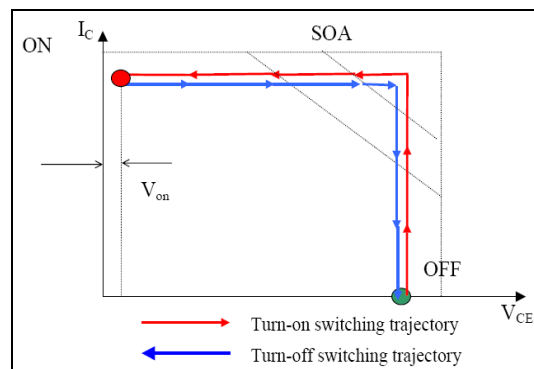
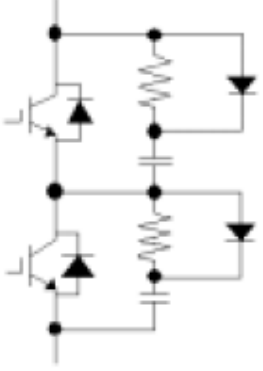
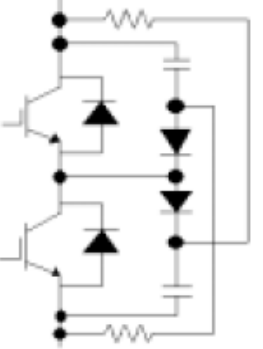
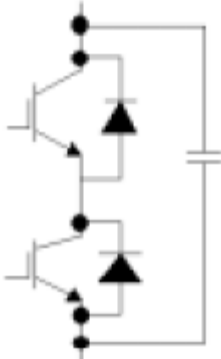
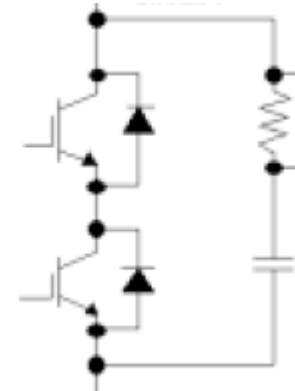


Figure 2.15: The switching trajectory for inductive load

Table 2: The types of snubbers circuit

Snubber Circuit Schematic	Features	Application
<p>Charge and discharge RCD snubber circuit</p> 	<ul style="list-style-type: none"> - The effect of turn-off surge voltage is moderate. - As opposed to RC snubber circuit, a snubber diode has been added. This allows snubber's resistance to increase and consequently avoid the IGBT's load condition at turn-on problem. - Since the power dissipation loss of this circuit (primarily cause by snubber's resistance) is much greater than that of the discharge suppressing snubber circuit, it is not considered for high frequency switching application. 	
<p>Discharge suppressing RCD snubber circuit</p> 	<ul style="list-style-type: none"> - The effect of turn-off surge voltage is small. - Suitable for high frequency switching application. - Power dissipation loss caused by snubber circuit is small. 	Inverter
<p>Capacitor snubber circuit</p>	<ul style="list-style-type: none"> - This is the simplest circuit. - The LC resonance circuit, which consists of main circuit inductance coil and snubber capacitor, may cause 	Inverter

	<p>the C-E voltage to oscillate.</p>	
<p>RCD snubber circuit</p> 	<ul style="list-style-type: none"> - If the wrong snubber diode is used a high voltage spike will be generated and the output voltage will oscillate at the diodes reverse recovery. 	<p>Inverter</p>

CHAPTER 3

METHODOLOGY

3.1 OVERALL SYSTEM DESIGN

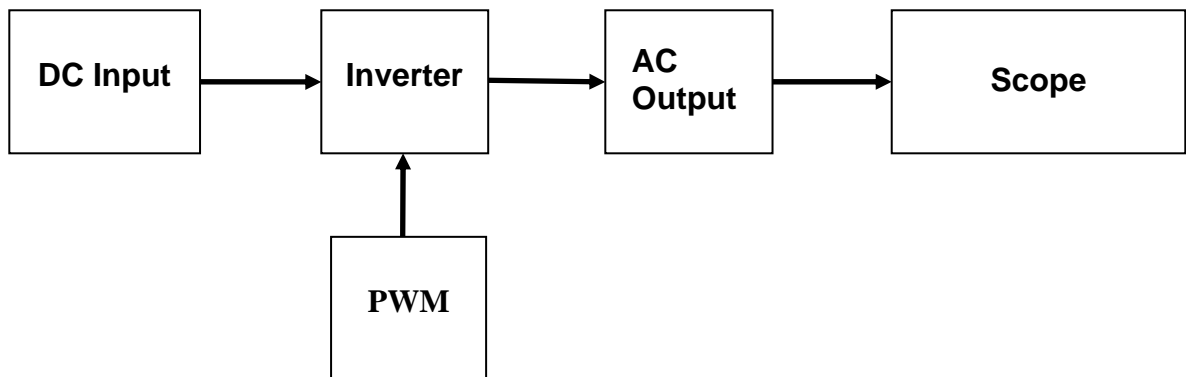


Figure 3.1: Block diagram of the system

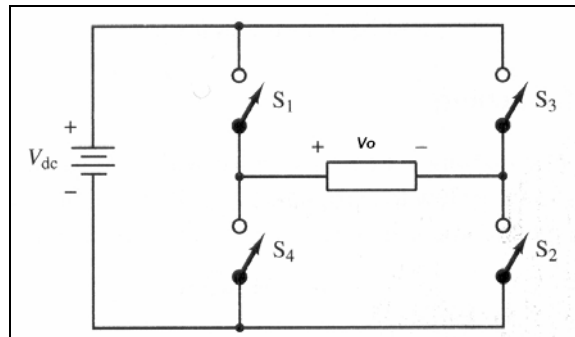
The block diagram is about the application of Pulse Width Modulation inverter. The Pulse-Width Modulation is implemented in inverter which at the end output of the inverter is obtained from MATLAB/Simulink's simulation. The commutation strategy to solve switching transient is implemented by using IGBT as switching device. The scope is used to see the output waveform of the inverter.

3.2 INVERTER CIRCUIT DESIGN

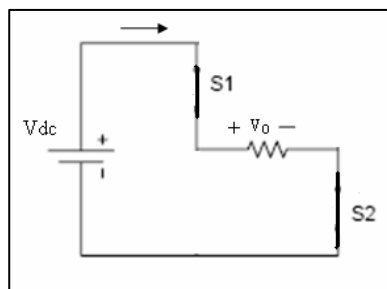
The full-bridge converter is the basic circuit used to convert DC to AC. The full bridge converter is a part of a DC power supply circuit. An AC output is synthesized from a DC input by closing and opening the switches in appropriate sequence. The output voltage, V_O can be $+V_{dc}$, $-V_{dc}$, or zero, depending on which switches are, closed [8].

Table 3: Switching sequence

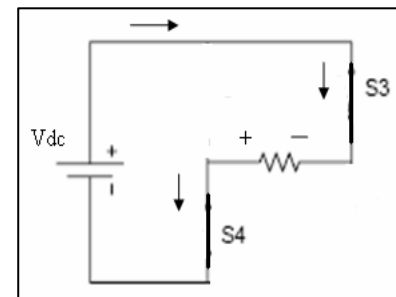
Switches closed	Output voltage, V_O
S1 and S2	$+V_{dc}$
S3 and S4	$-V_{dc}$
S1 and S3	0
S2 and S4	0



(a)



(b)



(c)

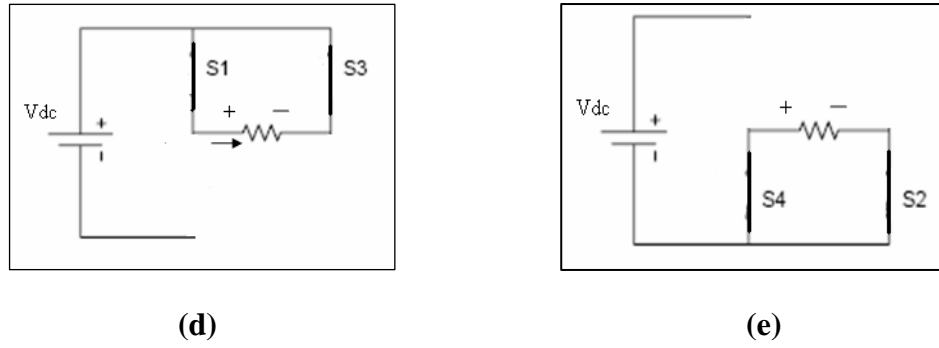


Figure 3.2: Inverter circuit and switching sequence (a) Full-bridge converter (b) S1 and S2 closed (c) S3 and S4 closed (d) S1 and S3 closed (e) S2 and S4 closed

3.2.1 CIRCUIT DESIGN IN MATLAB/SIMULINK SOFTWARE

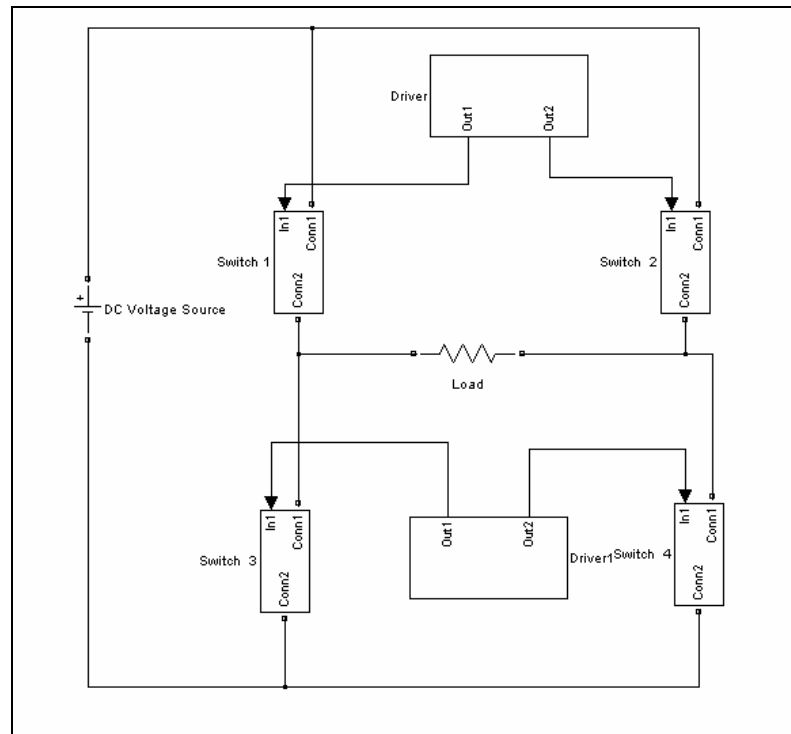


Figure 3.3: Inverter Circuit in Simulink

Figure 3.3 shows the inverter circuit had designed in Simulink. This circuit is same as basic inverter circuit with four switches. DC Voltage Source block from Simpower System library is used to supply DC voltage to the circuit. Subsystem block from Simulink library is used to design driver and switch block which the driver circuit and switch circuit is build in subsystem's window. The subsystem's window will appear if double click on the Subsystem block. Series RLC branch block from SimPower System library is used to design load circuit. Only resistive load is used for this circuit.

3.2.2 CIRCUIT DESIGN IN PSPICE SOFTWARE

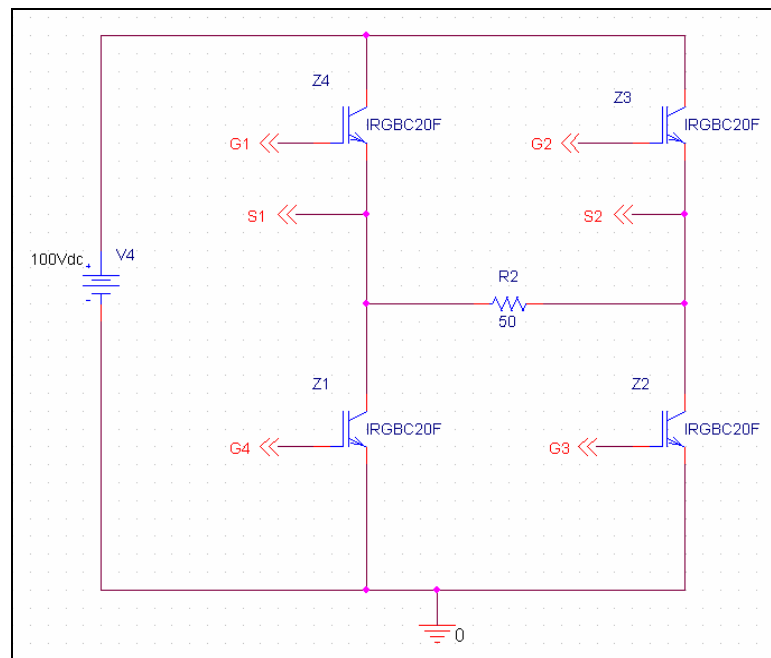


Figure 3.4: Inverter Circuit in Pspice

Figure 3.4 shows the inverter circuit had designed in Pspice software. The IGBT used is IRGBC20F. The source is 100VDC and load is 50ohms for resistive load.

3.3 CONTROL SWITCH AND DRIVER CIRCUIT

There are two types of the PWM switching schemes, unipolar and bipolar. In this circuit, the switching scheme applied is unipolar.

For PWM technique, a high frequency triangular carrier signal V_C , is compared with a sinusoidal reference signal, V_{ref} , of desired frequency. The crossover points are used to determine the switching instants. The magnitude ratio of reference signal to that of the triangular signal is known as modulation index. The magnitude of fundamental component of output voltage is proportional to mi . the amplitude V_C of the triangular signal is generally kept constant. By varying the modulation index, the output voltage could be controlled.

3.3.1 DESIGN IN SIMULINK

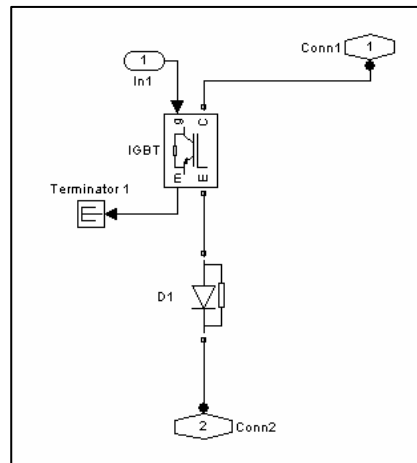


Figure 3.5: IGBT switch circuit

Figure 3.5 show the IGBT switch circuit had been used in this inverter circuit which designed in SIMULINK.

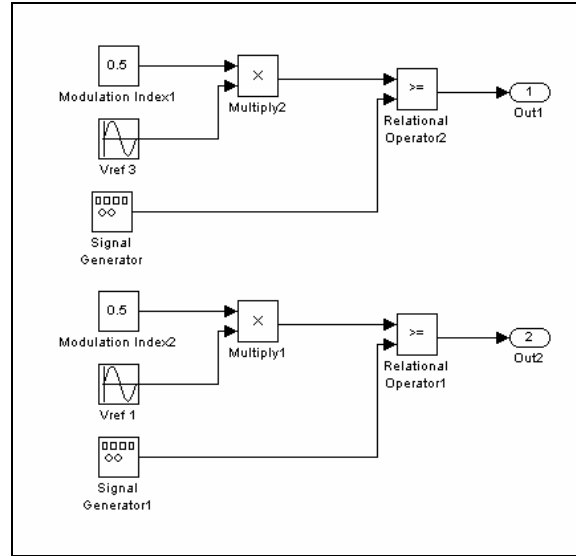


Figure 3.6: Driver circuit in Simulink

Figure 3.6 shows the design of driver circuit to generate Sinusoidal Pulse Width Modulation (SPWM) pattern that is controlled using the switching state as in Table 3. Every output will control one switch. This driver had designed in SIMULINK. In Figure 3.6, one “sine wave” block is used to generate sinusoidal reference signal V_{ref} . Output from the “sine wave” block is multiplied using a “multiply” block with the output from the constant block that represents the modulation index. The magnitude could be varied changing this “constant” value. The signal generator block is used to generate the triangular carrier signal, ‘ V_C ’. To produce SPWM, the “relational operator” block is used as a comparator that develops and output switching function between “0” and “1”.

Parameters used are shown in Table 4.

Table 4: Parameter for inverter circuit

Input Source (DC)	100Vdc
Frequency Reference Signal(f_r)	50 Hz
Frequency Carrier Signal(f_c)	1 kHz
Sample modulation Index(mi)	0.5
Load	$R = 50 \Omega$

3.3.2 DESIGN IN PSPICE

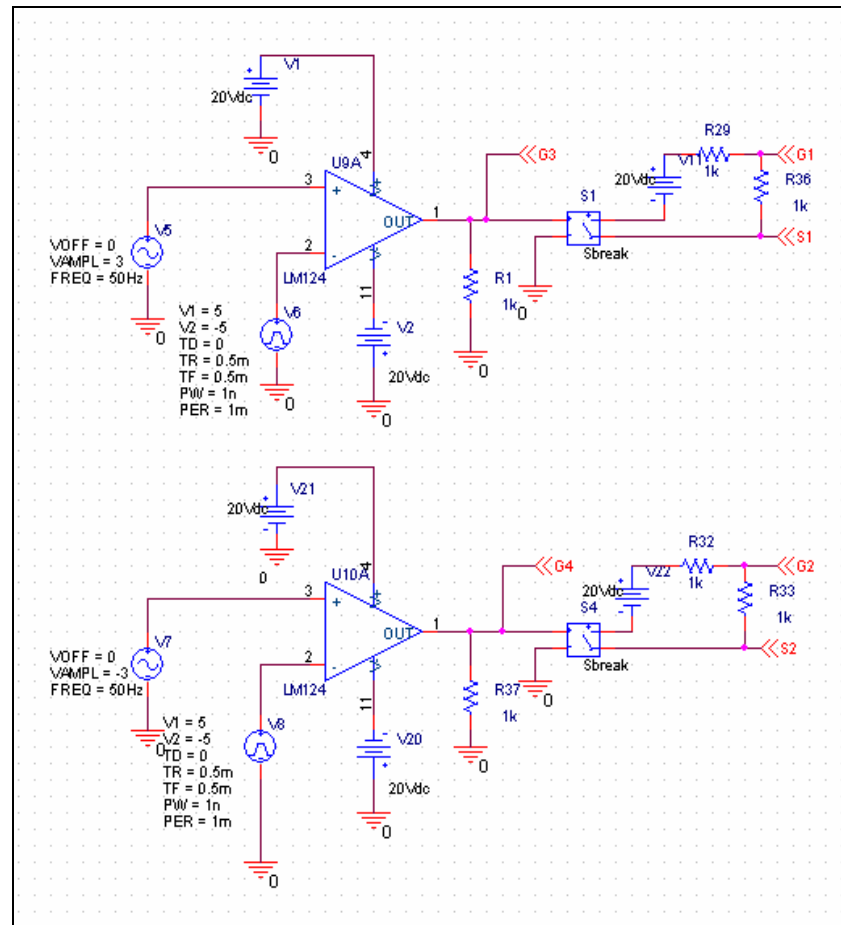


Figure 3.7: Driver circuit in Pspice

Figure 3.7 shows the driver circuit had designed in Pspice software. Vsine source is used to generate sinusoidal reference wave. The Vpulse source is used to generate triangular carrier wave. The comparator, LM 124 is used to compare sinusoidal reference wave and triangular carrier wave to produce Pulse Width Modulation pulse. The parameter used for design driver circuit is same as shown in figure above.

3.4 MODULATION INDEX CALCULATION

The formula of modulation index, mi is:

$$\text{Modulation index, } mi = \frac{V_{ref}}{V_{carrier}} \quad (3.1)$$

For $mi = 0.5$, with amplitude of $V_{reference} = 2.5 \text{ V}$ and $V_{carrier} = 5 \text{ V}$.

$$\begin{aligned} \text{Modulation index, } mi &= \frac{0.25 \text{ V}}{5 \text{ V}} \\ &= 0.5 \end{aligned}$$

CHAPTER 4

RESULT AND ANALYSIS

4.1 SIMULINK AND PSPICE RESULTS

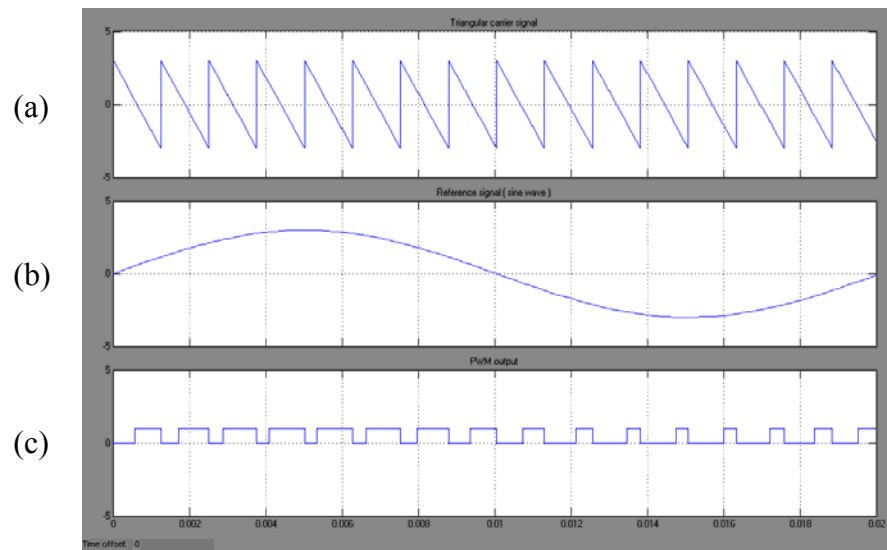


Figure 4.1: Driver output in Simulink (a) triangular carrier wave (b) sinusoidal reference wave (c) PWM pulse

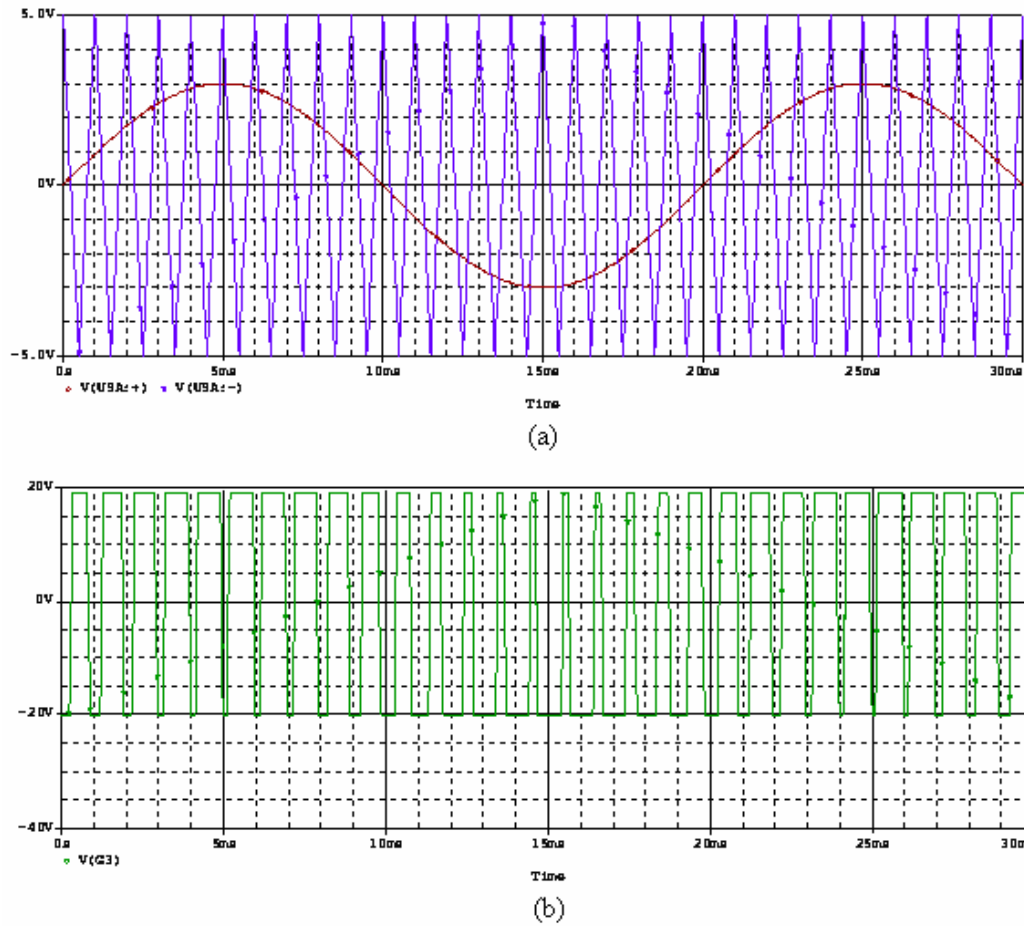
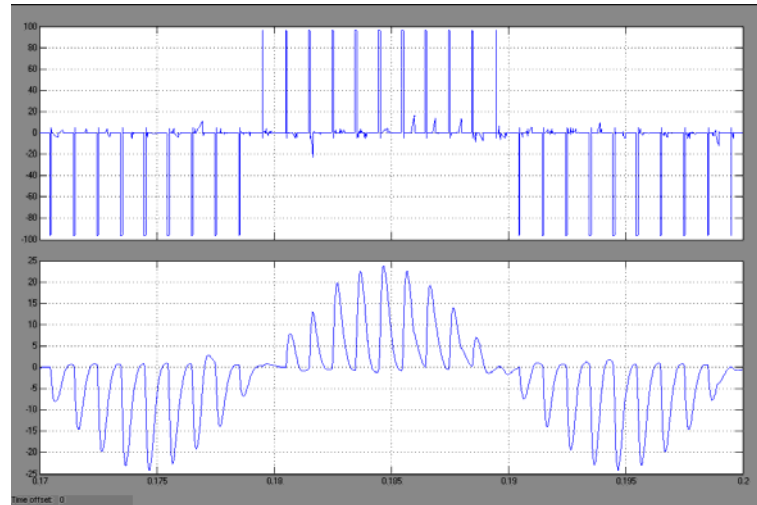


Figure 4.2: Driver output in PSpice (a) sinusoidal reference wave and triangular carrier wave (b) PWM pulse

Figure 4.1 and 4.2 show the output of the driver circuit. The PWM pulse is produced by comparing a triangular carrier wave with a sinusoidal reference wave for PWM switching method.

**Fundamental
output** →

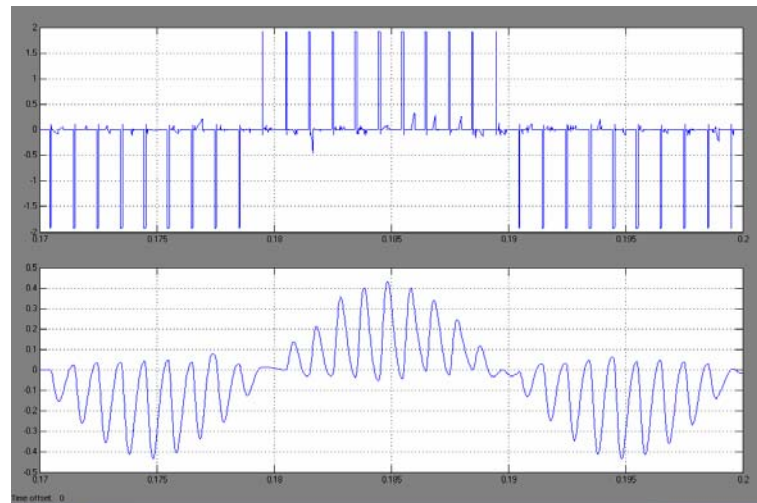
**Output with
LC filter** →



(a)

**Fundamental
output** →

**Output with
LC filter** →

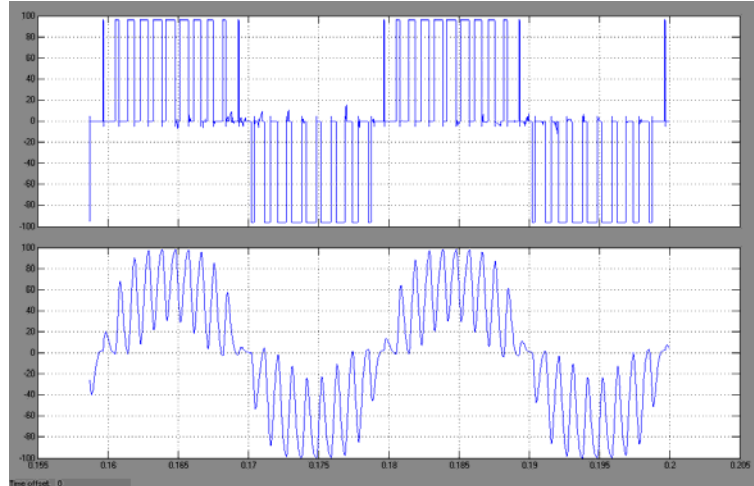


(b)

Figure 4.3: Result of simulation of R load with modulation index, $mi = 0.1$ (a) output voltage (b) output current

**Fundamental
output** →

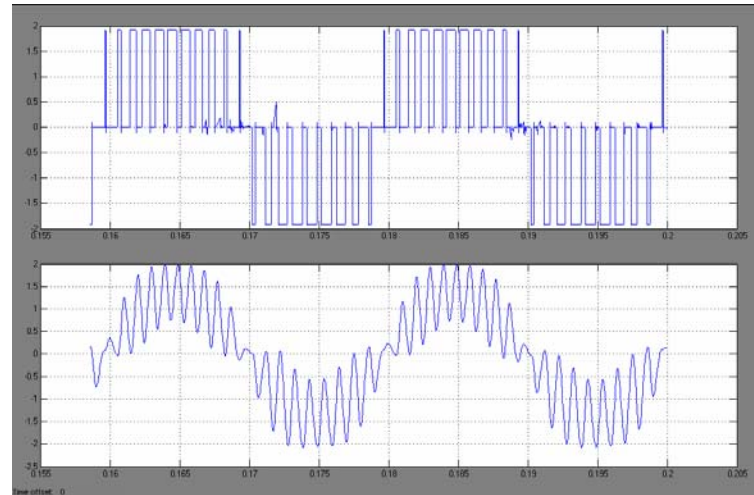
**Output with
LC filter** →



(a)

**Fundamental
output** →

**Output with
LC filter** →



(b)

Figure 4.4: Result of simulation of R load with modulation index, $mi = 0.8$ (a) output voltage (b) output current

From Figure 4.3 and 4.4, the results show how the different mi produced different output voltage. The value output voltage can be manipulated by manipulate the value of mi which the higher mi will produce higher output voltage. The value of mi is manipulating by control the amplitude of V_{ref} in PWM switching method. The magnitude of output waveform was obtained by using LC filter. The design of LC filter becomes easy with the ‘Analog Filter Design block’ function.

Table 5: Modulation index with different reference frequency, f_{ref}

Modulation Index (MI)	Output Voltage (Vrms) with $f_{\text{ref}} = 50\text{Hz}$	Output Voltage (Vrms) with $f_{\text{ref}} = 100\text{Hz}$	Output Voltage (Vrms) with $f_{\text{ref}} = 150\text{Hz}$
0.1	16.55	16.1	16.52
0.2	31.82	31.1	30.18
0.3	44.62	43.49	41.72
0.4	54.71	53.1	52.68
0.5	61.76	60	58.41
0.6	66.33	64.66	64
0.7	68.59	67.32	66.82
0.8	69.72	68.94	68.59
0.9	70	69.86	69.58

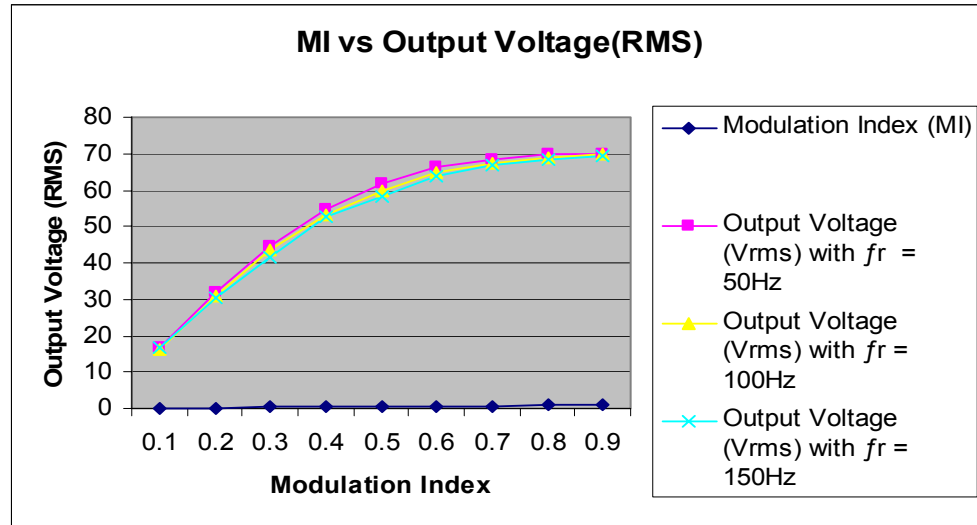
**Figure 4.5:** Graph MI versus Output Voltage (RMS)

Figure 4.5 shows the graph of comparison output voltage for variation modulation index, mi and different frequency for same reference voltage, V_{ref} . For the different frequency, the different output voltage had produced for same mi because time switching for one duty cycle is much shorter for high frequency. From formula $t = 1/f$, if frequency is bigger, the time will become much shorter. This will influence the period of switching for the same duty cycle and it will cause the output voltage produced from high frequency of switching become much lower.

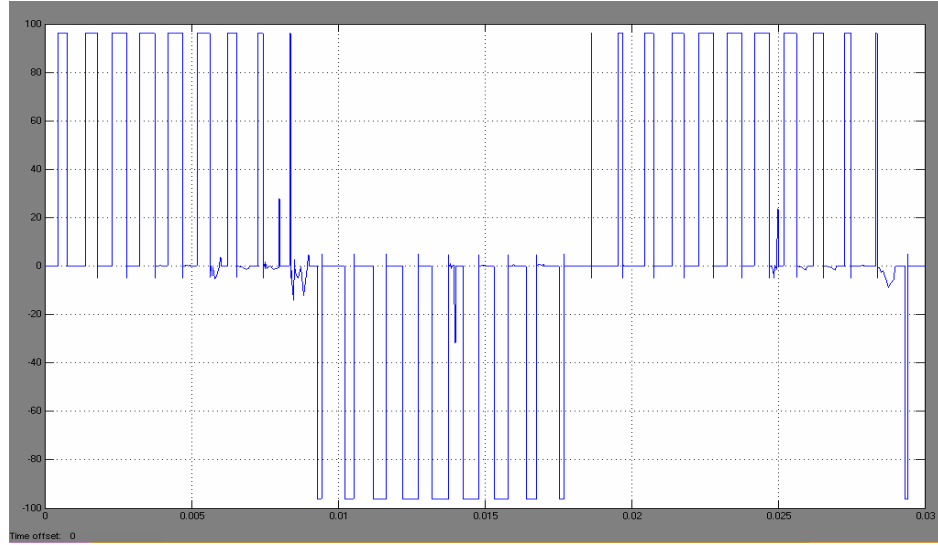


Figure 4.6: Simulink's simulation result of output voltage with $mi = 0.6$

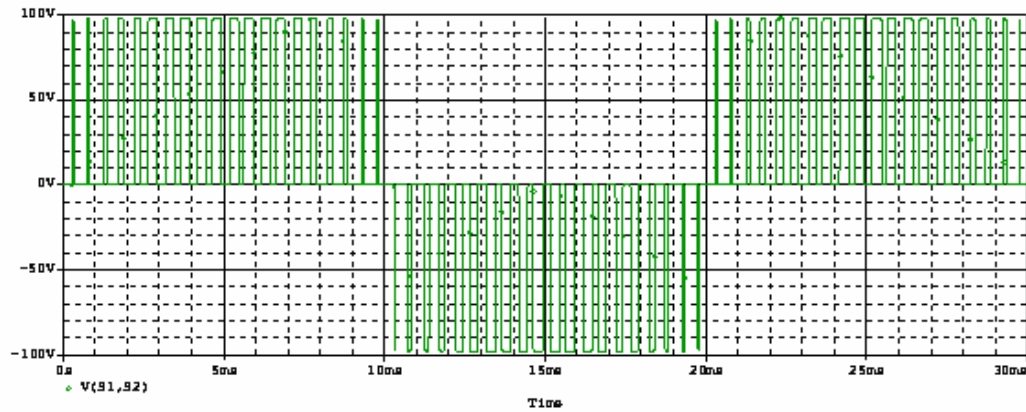
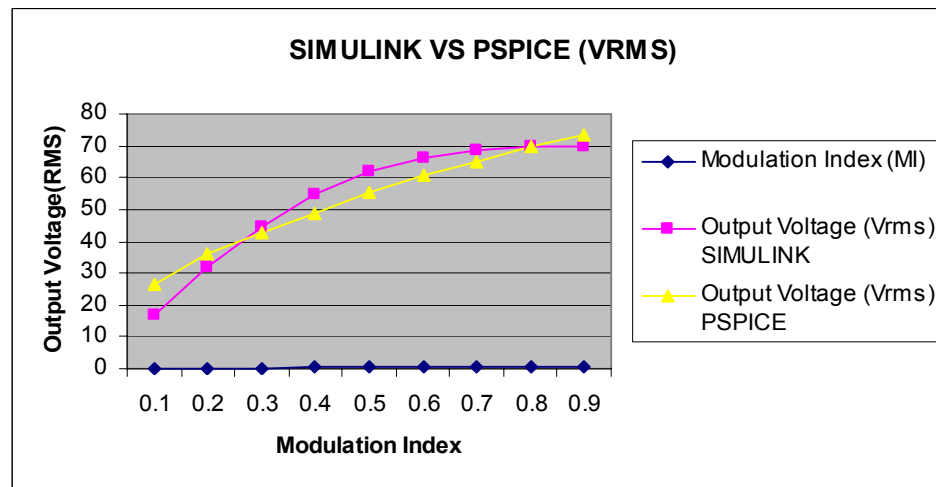


Figure 4.7: Pspice's simulation result of output voltage with $mi = 0.6$

Figure 4.6 and 4.7 show the simulation result of Simulink and Pspice. The simulation was done for $mi = 0.1$ to 0.9 . The result of this simulation is shown in Table 6.

Table 6: Output voltage (Simulink versus Pspice)

Modulation Index (MI)	Output Voltage (Vrms) SIMULINK	Output Voltage (Vrms) PSPICE
0.1	16.55	26.3
0.2	31.82	35.866
0.3	44.62	43
0.4	54.71	49
0.5	61.76	55.5
0.6	66.33	60.5
0.7	68.59	65.24
0.8	69.72	69.6
0.9	70	73.33

**Figure 4.8:** Graph Simulink versus Pspice (Vrms)

For the comparison of the results between Simulink and Pspice, the method applied for switching scheme is same for both. From Figure 4.6, there are not much different output voltage had produced from both simulations. But in Simulink, the simulation is not involved losses in the circuit, compared to Pspice. That why the simulation in Pspice is most used for designing circuit prior to real project.

4.2 APPLICATION OF PWM CONTROL

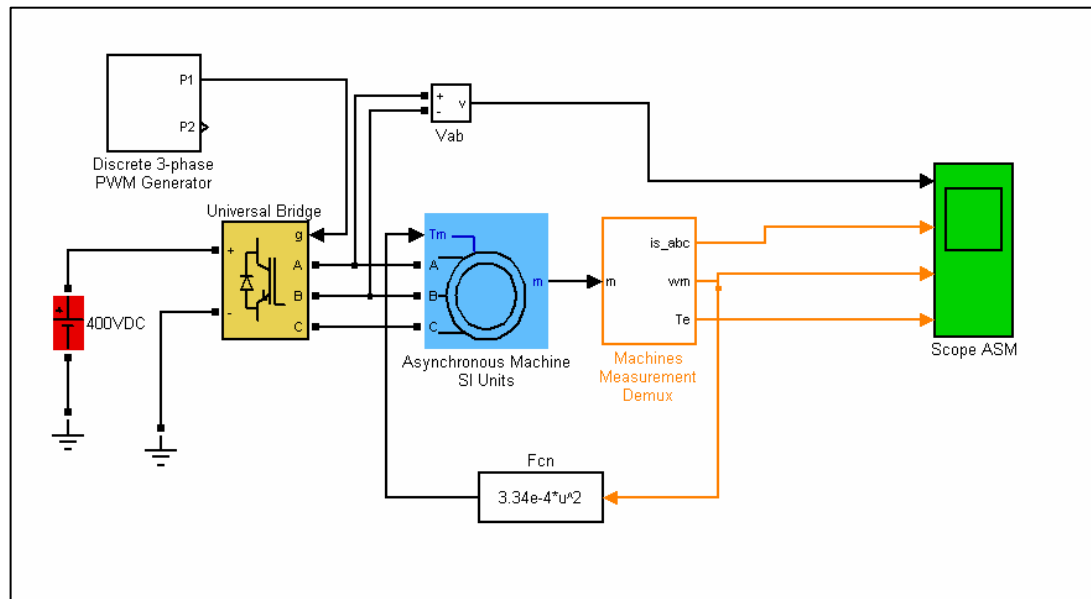
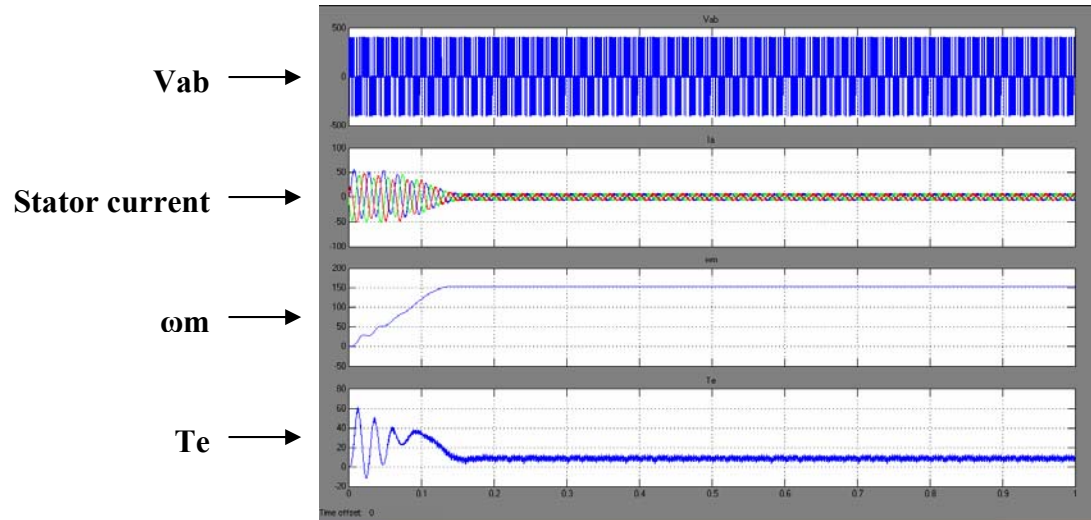
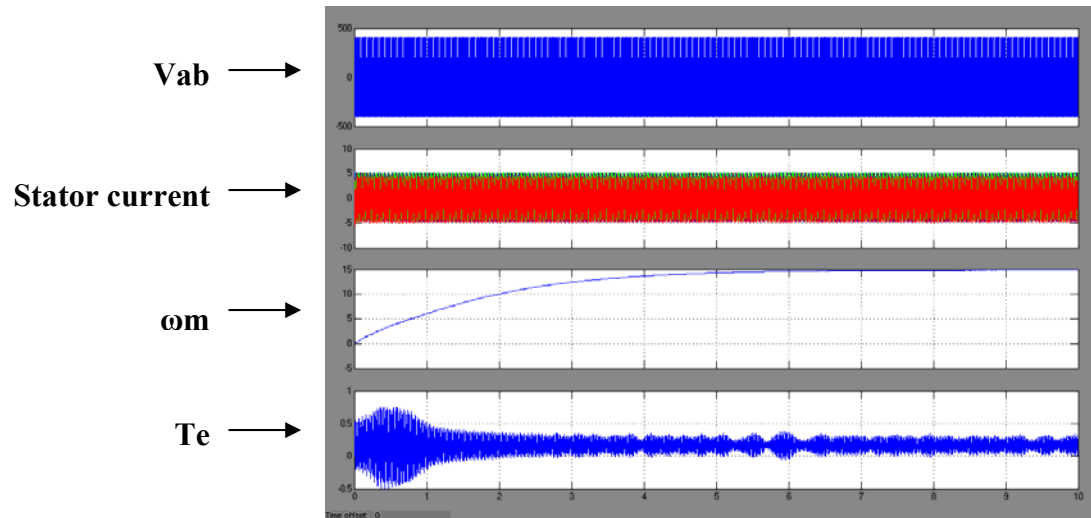


Figure 4.9: Application for PWM control in 3-phase machine

Figure 4.7 shows the application of PWM control in 3-phase machine. This application use Asynchronous Machine block which to show by verify the modulation index, mi output of this machine can be controlled. Universal Bridge block implement a bridge of selected power electronic devices that function same as three phase inverter circuit with selection of power electronic devices like IGBT, MOSFET and GTO. So, IGBT had chosen as power electronic devices for switching same as the single phase inverter had designed. The mi is verified using discrete 3-phase PWM generator block. The Machines Measurement Demux block is a measurement block that used to measure the selection outputs. The selection outputs for this application had chosen for presentation is voltage phase to phase, V_{ab} , rotor speed, ω_m , stator current and electromagnetic torque, T_e .



(a)



(b)

Figure 4.10: Simulation results for 3-phase machine by verify modulation index (a) $mi = 0.1$ (b) $mi = 0.9$

From Figure 4.8(a) and (b), the graphs show the different output had presented with the modulation index, mi is verified which $mi = 0.1$ and $mi = 0.9$ each. This results show how by verify the mi , the output of the machine like the rotor speed, ω_m can be easily controlled.

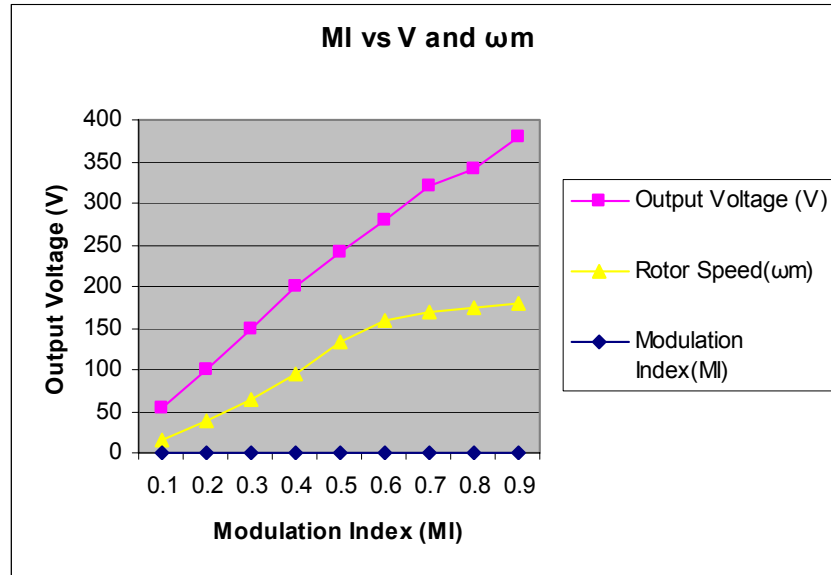


Figure 4.11: Modulation Index versus magnitude voltage (V_{rms})

The graph above shows the output voltage of the machines and rotor speed (ω_m) versus modulation index (MI). The result had presented was record by manipulate the modulation index, mi of the V_{ref} . From the graph above, when the mi increase, the output voltage and speed of rotor proportional increase.

4.3 PWM INVERTER ANALYSIS

Table 7: Parameter for PWM inverter analysis

Input source (DC)	100Vdc
Reference frequency signal (f_r)	50Hz
Carrier Signal (f_c)	1000Hz
Sample modulation Index	0.8
Load	R = 50Ω L=25mH
Frequency modulation, mf	20 [$f_{tri} = (20)(50) = 1000\text{Hz}$]

The amplitude of the 50Hz fundamental frequency

$$V_1 = miV_{dc} = (0.8)(100) = 80 \text{ V}$$

The current amplitude is determined using phasor analysis:

$$I_n = \frac{V_n}{Z_n} = \frac{V_n}{\sqrt{R^2 + (n\omega_o L)^2}} \quad (4.1)$$

For fundamental frequency:

$$I_1 = \frac{80}{\sqrt{50^2 + [(1)(2\pi)(50)(0.025)]^2}} = 1.58 \text{ A}$$

Table 8: NORMALIZED FOURIER COEFFICIENTS FOR UNIPOLAR PWM

	$mi=1$	0.9	0.8	0.7	0.6	0.5	0.4	0.3	0.2	0.1
$n=1$	1	0.9	0.8	0.7	0.6	0.5	0.4	0.3	0.2	0.1
$n= 2mf \pm 1$	0.18	0.25	0.31	0.35	0.37	0.36	0.33	0.27	0.19	0.1
$n = 2mf \pm 3$	0.21	0.18	0.14	0.1	0.07	0.04	0.02	0.01	0	0

With $mf = 20$, the first harmonic are at $n = 41, 39, 43$ and 37 ,

$$V_{41} \text{ and } V_{39} = (0.31)(100) = 31 \text{ V}$$

$$V_{43} \text{ and } V_{37} = (0.14)(100) = 14 \text{ V}$$

Power for each frequency is determined from:

$$P_n = (I_{n,rms})^2 R = \left(\frac{I_n}{\sqrt{2}} \right)^2 R \quad (4.2)$$

Table 9: Fourier Series Quantities For PWM Inverter

n	f_n	$V_n(V)$	$Z_n(\Omega)$	$I_n(A)$	$I_{n,rms}(A)$	$P_n(W)$
1	50	80	50.61	1.58	1.12	62.72
37	1850	14	294.87	0.047	0.033	0.054
39	1950	31	310.36	0.1	0.071	0.252
41	2050	31	325.87	0.095	0.067	0.22
43	2150	14	341.4	0.041	0.029	0.04

Power absorbed by the load resistor:

$$P = \Sigma P_n \approx 62.72 + 0.054 + 0.252 + 0.22 + 0.22 + 0.04 \\ = 63.286 \text{ W}$$

The THD of the load current,

$$THD_I = \frac{\sqrt{\Sigma (I_{n,rms})^2}}{I_{1,rms}} \quad (4.3) \\ = \frac{\sqrt{(0.033)^2 + (0.071)^2 + (0.067)^2 + (0.029)^2}}{1.12} \\ = 0.096 = 9.6\%$$

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

The modeling and simulation of single phase inverter had been presented using Simulink software and Pulse Width Modulation as switching method. The PWM switching method are used in inverter because this method have advantages like the fundamental voltage can be verified, harmonic profile can be controlled, filter requirement can be reduce and overall size can be reduced. Although this PWM method can cause higher switching losses, but it can be control by using IGBT as switching device. In this project, the variable voltage output is produced by varying modulation index, mi . Even though the implementation of PWM method is difficult, but the advantages presented give more benefit to be used in real application.

5.2 RECOMMENDATION

Here is some enhancement of this project in future:

- i. Modeling and simulation for three phase inverter.
- ii. Make simulation of application for 3 phase machine using the three phase inverter had designed.
- iii. Use second integration method by entering command in MATLAB's command window for simulation.

5.3 COMMERCIALIZATION

Making analysis for the project is first step before make real project. The analysis is doing based on the simulation of the project. The successfully of the project is depend on the analysis of the simulation that be done before.

In industry, the engineers always make analysis on the machine like motor either single phase or three-phase motor to see their problems. By making the simulation on that machine, the analysis can be done to look what the problems and how to overcome it.

This project presented the simulation by using MATLAB/Simulink as a tool in making analysis. So, this project can be used as reference for the engineering student or any engineer that work with simulation and doing analysis.

REFERENCE

- [1] H. Mohd Hanafi, Z.Idris, M.K. Hamzah, Member, IEEE and Azilah Saparon, “Modeling & Simulation of Single-phase Matrix Converter as a Frequency Changer with Sinusodial Pulse Width Modulation Using MATLAB/Simulink” , First International Power and Energy Conference PEC on 2006, November 28-29, 2006, Putrajaya, Malaysia.
- [2] 5 February 2007 source URL
<http://www-ccs.ucsd.edu/matlab/toolbox/simulink/simulink.html>
- [3] 9 February 2007 Paul Barnard, “New Power System Blockset Enables You to Model Electrical Power Systems”, Simulink moves into the realm of topological modeling source URL http://www.mathworks.com/company/newsletters/news_notes/sum98/sum98psb.html
- [4] 15 February 2007 source URL
http://www-ccs.ucsd.edu/matlab/toolbox/simulink/c01_ge12.html
- [5] 20 February 2007 source URL
http://en.wikipedia.org/wiki/Inverter_%28electrical%29
- [6] 20 February 2007 ‘What is a Power Inverter?’ source URL
<http://www.wisegeek.com/what-is-a-power-inverter.htm>.
- [7] 1 March 2007 source URL
<http://www.energyalternatives.ca/SystemDesign/inverters1.html>
- [8] Daniel W.Hart , “ Introduction To Power Electronic.
- [9] 9 March 2007 Barr, Michael, "Pulse Width Modulation" Embedded Systems Programming, September 2001, pp. 103-104 source URL
<http://www.netrino.com/Publications/Glossary/PWM.php>.

- [10] Liebert , “All Transistors Are Not Created Equal: The Advantages of IGBT's in 3-Phase UPS Products” of white paper
- [11] Carl Blake and Chris Bull, International Rectifier, “IGBT or MOSFET: Choose Wisely”.
- [12] Wajiha Shireen IEEE Senior Member University of Houston, Rahul A. Kulkarni IEEE Student Member University of Houston, “USING MATLAB FOR HARMONIC ASSESSMENT OF THREE PHASE PWM INVERTERS” source URL <http://www.ijme.us/issues/spring2005/articles/MatlabHarmonicAssessmentInverters.htm>.
- [13] Rudy Severns “DESIGN OF SNUBBERS FOR POWER CIRCUITS”
- [14] IC Illinois Capacitor, Inc.”Application”
- [15] Fuji Electric Device Technology Co.,Ltd ”Protection Circuit Design”.

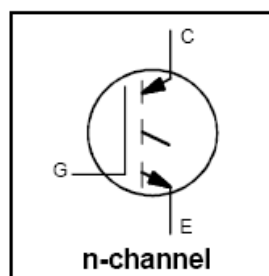
APPENDIX A



PD - 9.686A

IRGBC20F**INSULATED GATE BIPOLAR TRANSISTOR****Fast Speed IGBT****Features**

- Switching-loss rating includes all "tail" losses
- Optimized for medium operating frequency (1 to 10kHz) See Fig. 1 for Current vs. Frequency curve



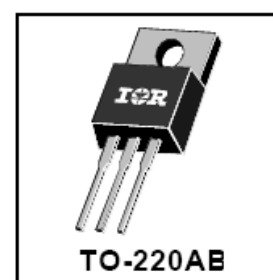
$$V_{CES} = 600V$$

$$V_{CE(sat)} \leq 2.8V$$

$$@V_{GE} = 15V, I_C = 9.0A$$

Description

Insulated Gate Bipolar Transistors (IGBTs) from International Rectifier have higher usable current densities than comparable bipolar transistors, while at the same time having simpler gate-drive requirements of the familiar power MOSFET. They provide substantial benefits to a host of high-voltage, high-current applications.

**Absolute Maximum Ratings**

	Parameter	Max.	Units
V_{CES}	Collector-to-Emitter Voltage	600	V
$I_C @ T_C = 25^\circ C$	Continuous Collector Current	16	A
$I_C @ T_C = 100^\circ C$	Continuous Collector Current	9.0	
I_{CM}	Pulsed Collector Current ①	64	
I_{LM}	Clamped Inductive Load Current ②	64	
V_{GE}	Gate-to-Emitter Voltage	± 20	V
E_{ARV}	Reverse Voltage Avalanche Energy ③	5.0	mJ
$P_D @ T_C = 25^\circ C$	Maximum Power Dissipation	60	W
$P_D @ T_C = 100^\circ C$	Maximum Power Dissipation	24	
T_J	Operating Junction and	-55 to +150	$^\circ C$
T_{STG}	Storage Temperature Range		
	Soldering Temperature, for 10 sec.	300 (0.063 in. (1.6mm) from case)	
	Mounting torque, 6-32 or M3 screw.	10 lbf•in (1.1N•m)	

Thermal Resistance

	Parameter	Min.	Typ.	Max.	Units
$R_{\theta JC}$	Junction-to-Case	—	—	2.1	$^\circ C/W$
$R_{\theta CS}$	Case-to-Sink, flat, greased surface	—	0.50	—	
$R_{\theta JA}$	Junction-to-Ambient, typical socket mount	—	—	80	
Wt	Weight	—	2.0 (0.07)	—	g (oz)

IRGBC20F

Electrical Characteristics @ $T_J = 25^\circ\text{C}$ (unless otherwise specified)

	Parameter	Min.	Typ.	Max.	Units	Conditions
$V_{(BR)CES}$	Collector-to-Emitter Breakdown Voltage	600	—	—	V	$V_{GE} = 0V, I_C = 250\mu A$
$V_{(BR)ECS}$	Emitter-to-Collector Breakdown Voltage ①	20	—	—	V	$V_{GE} = 0V, I_C = 1.0A$
$\Delta V_{(BR)CES}/\Delta T_J$	Temp. Coeff. of Breakdown Voltage	—	0.72	—	$V/^\circ\text{C}$	$V_{GE} = 0V, I_C = 1.0mA$
$V_{CE(on)}$	Collector-to-Emitter Saturation Voltage	—	2.0	2.8	V	$I_C = 9.0A, V_{GE} = 15V$
		—	2.6	—		$I_C = 16A$ See Fig. 2, 5
		—	2.3	—		$I_C = 9.0A, T_J = 150^\circ\text{C}$
$V_{GE(th)}$	Gate Threshold Voltage	3.0	—	5.5		$V_{CE} = V_{GE}, I_C = 250\mu A$
$\Delta V_{GE(th)}/\Delta T_J$	Temp. Coeff. of Threshold Voltage	—	-11	—	$mV/^\circ\text{C}$	$V_{CE} = V_{GE}, I_C = 250\mu A$
g_{fe}	Forward Transconductance ②	2.9	5.1	—	S	$V_{CE} = 100V, I_C = 9.0A$
I_{CES}	Zero Gate Voltage Collector Current	—	—	250	μA	$V_{GE} = 0V, V_{CE} = 600V$
		—	—	1000		$V_{GE} = 0V, V_{CE} = 600V, T_J = 150^\circ\text{C}$
I_{GES}	Gate-to-Emitter Leakage Current	—	—	± 100	nA	$V_{GE} = \pm 20V$

Switching Characteristics @ $T_J = 25^\circ\text{C}$ (unless otherwise specified)

	Parameter	Min.	Typ.	Max.	Units	Conditions
Q_g	Total Gate Charge (turn-on)	—	16	21	nC	$I_C = 9.0A$
Q_{ge}	Gate - Emitter Charge (turn-on)	—	2.4	3.4		$V_{CC} = 400V$ See Fig. 8
Q_{gc}	Gate - Collector Charge (turn-on)	—	7.8	10		$V_{GE} = 15V$
$t_{d(on)}$	Turn-On Delay Time	—	24	—	ns	$T_J = 25^\circ\text{C}$
t_r	Rise Time	—	13	—		$I_C = 9.0A, V_{CC} = 480V$
$t_{d(off)}$	Turn-Off Delay Time	—	160	270		$V_{GE} = 15V, R_G = 50\Omega$
t_f	Fall Time	—	310	600		Energy losses include "tail"
E_{on}	Turn-On Switching Loss	—	0.18	—	mJ	See Fig. 9, 10, 11, 14
E_{off}	Turn-Off Switching Loss	—	0.90	—		
E_{ts}	Total Switching Loss	—	1.08	2.0		
$t_{d(on)}$	Turn-On Delay Time	—	25	—	ns	$T_J = 150^\circ\text{C}$
t_r	Rise Time	—	18	—		$I_C = 9.0A, V_{CC} = 480V$
$t_{d(off)}$	Turn-Off Delay Time	—	210	—		$V_{GE} = 15V, R_G = 50\Omega$
t_f	Fall Time	—	600	—		Energy losses include "tail"
E_{ts}	Total Switching Loss	—	1.65	—	mJ	See Fig. 10, 14
L_E	Internal Emitter Inductance	—	7.5	—	nH	Measured 5mm from package
C_{ies}	Input Capacitance	—	340	—	pF	$V_{GE} = 0V$
C_{oes}	Output Capacitance	—	63	—		$V_{CC} = 30V$ See Fig. 7
C_{res}	Reverse Transfer Capacitance	—	5.9	—		$f = 1.0MHz$

Notes:

① Repetitive rating; $V_{GE}=20V$, pulse width limited by max. junction temperature. (See fig. 13b)

③ Repetitive rating; pulse width limited by maximum junction temperature.

⑤ Pulse width $5.0\mu s$, single shot.

② $V_{CC}=80\%(V_{CES})$, $V_{GE}=20V$, $L=10\mu H$, $R_G=50\Omega$, (See fig. 13a)

④ Pulse width $\leq 80\mu s$; duty factor $\leq 0.1\%$.