DESIGN SWITCH MODE POWER SUPPLY BY USING
BOOST CONVERTER TOPOLOGY

KHAIRUL IZZAT BIN MOHAMAD SAHIMI

MAY 2008

DESIGN SWITCH MODE POWER SUPPLY BY USING BOOST CONVERTER TOPOLOGY

KHAIRUL IZZAT BIN MOHAMAD SAHIMI

This Report Is Submitted In Partial Fulfillment Of Requirements For The Degree of Bachelor In Electrical Engineering (Power Electronic and Drive)

Faculty of Electrical Engineering
Universiti Teknikal Malaysia

April 2008

"I hereby declared that I have read through this report and found that it has comply the partial fulfillment for awarding the degree of Bachelor of Electrical Engineering (Power Electronic and Drive)."

Signature : …….

Supervisor's Name : …….

Date : …….

"I hereby declared that this report is a result of my own work except for the excerpts that have been cited clearly in the references.."

Signature : ……

Name : ……

Date : ……

Dedicated to :

~ my beloved parents ~

~ my supervisor, En. Fazlli Bin Patkar ~

~ my friends ~

..and to the reader of this journal

ABSTRACT

This project is about design a Switch Mode Power Supply (SMPS) using a boost topology. A switched-mode power consists of converter and a controller circuit that controls the switch of the converter. A converter is a circuit that is used to amplify or reduce the voltage by using switches like power transistor (MOSFET, IGBT and etc) where it will be operates on and off in a high frequency and the control circuit will be the one which control the switching frequency. A boost topology is a circuit where the input voltage can be increased to a demanded value. In this project about designing a SMPS, a feedback controller will be added to a boost converter where it will be able to maintain its output voltage even there are changes to its load or input voltage. To achieve this result, the feedback will control the duty cycle (D) of the converter. The feedback controller will be designed to check the output voltage and give signal and changed the value of D. The controller circuit is compilation of Error Amplifier and Pulse Width Modulator. All of this project will be analyzed and simulated through electrical simulation software such as PSpice and Matlab. Each steps of designing the SMPS will be describe thoroughly so that the report of this project will become a reference for designing SMPS in the future.

ABSTRAK

Projek ini adalah mengenai merekacipta sebuah Pembekal Bekalan Kuasa Ragam Pensuisan (BKRP) dengan menggunakan topologi boost. BKRP terdiri daripada penukar dan juga litar pengawal yang mengawal suis pada penukar. Penukar adalah litar yang digunakan untuk meningkat atau mengurangkan voltan pada masukan dengan menggunakan transistor kuasa (MOSFET,IGBT dan sebagainya) dan ia akan beroperasi secara buka dan tutup pada frekuensi tinggi dan litar pengawal adalah pengawal frekuensi tersebut. Topologi boost adalah jenis litar yang meningkatkan voltan masukan kepada nilai yang diperlukan mengikut kehendak. Projek ini menekankan tentang merekacipta sebuah BKRP, litar pengawal akan ditambah pada penukar boost dimana ia mampu mengekalkan voltan keluaran walaupun terdapat perubahan pada beban atau voltan masukan. Untuk mencapai tujuan ini, litar pengawal akan mengawal kitar kerja, D pada penukar. Litar pengawal akan direka untuk mengesan voltan keluaran and memberi isyarat untuk menukar nilai D. Litar pengawal terdiri daripada Penguat Ralat dan Pemodulat Jalur Lebar. Keseluruhan projek ini akan dianalisis dan disimulasi dengan menggunakan perisian computer seperti PSpice dan Malab. Setiap langkah akan diterangkan secara terperinci agar laporan ini akan dijadikan rujukan untuk merekacipta SMPS pada masa akan datang.

CONTENTS

LIST OF FIGURE

LIST OF FIGURE

LIST OF NOMENCLATURE

SMPS	Switch Mode Power Supply
PSU	Power Supply Unit
PWM	Pulse Width Modulation
DC	Direct Current
AC	Alternating Current
esr	Effective Series Resistance
VMC	Voltage Mode Control
CMC	Current Mode Control

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Objective

The objective of this project is to design, analyze, and control a Switch Mode Power Supply (SMPS) using a boost converter topology. A boost converter is known for its characteristic to amplify or increase the input voltage. The increment of input voltage is being controlled by a switch and a diode instead of a transformer which is widely used in electronic design. The switch and the diode will act by conducting and breaking the flow of current through the circuit and with the additional usage of inductor, it will be able to increase the voltage across the circuit. The inductor characteristic is an element that is able to store energy inside it by charging current through it and also be able to re-use the energy to act as a source in the circuit. A basic circuit of boost converter only is able to increase the voltage without controlling it. The SMPS will be design to be able to maintain its output voltage even with disturbances. This can be done by creating a feedback circuit and applying it into the boost converter circuit

The second objective of this project is that the SMPS design will be simulated to observe the output results. The simulation is crucial so that we will be able to

prove that the circuit be able to maintain its output even though there are disturbances with our design. There are many type of electronic simulation programme available in the market but, in this project we will only be using the simulation programme named PSpice so that the designing process will be much easier.

Then, the next objective is that the hardware of the design will be build exactly similar to the simulation design. Each if the components and elements will be carefully chosen in this stage so that the hardware will be not be much of differences

compared with the simulation design because, so that we be able to observe and prove the first objective where the output be able to maintain its value even though there are disturbances. Both of the simulation and the hardware results will be compared so that we be able to recognize the actual error that occurs which affecting the output results that cannot be detected during simulation.

The final objective of this project is that all the designing process and analyzing results will be compiled together making a full guidance in designing a SMPS. The guide will provide a complete step, from the first step of designing an SMPS until the analysis of its output.

1.2 Project Scope

This project focuses primarily on step by step design, analyze and control the switch mode power supply. The design process is a process on designing and theoretically realizing a basic converter topology for SMPS. Meanwhile, the analyzing process is about observing and studying the results of the design's output. Finally controlling a SMPS is a process where the SMPS will apply a feedback circuit to control its output.

1.3 Problem Statements

SMPS design is known for its capability to increase or decrease input voltage based on their topologies. Each of the topologies had their own characteristic but each of them use a switching sequence as a methods of amplifying the voltage and each of them had a same problems, high voltage ripples and unable to maintain its output upon disturbances. There an easier ways on controlling the output voltage ripples, where the value of inductor, capacitor, and switching frequency of the SMPS are varies causing the output voltage ripples to decrease. Even so, applying this method would not solve the second problems where the output voltage cannot maintain its set up value when there are disturbances. So, another approaches are needed where a controller circuit which can control the switching frequency and at the same time controlling the duty ratio of the SMPS is applied. This controller circuit also known as a feedback circuit contain of Error Amplifier and PWM. These

two materials will detect differences/changes of the output voltage and give an appropriate signal to the switch and change the value of duty ratio. This will cause a much more maintain output compared to its original states and in the mean time, causing the output voltage ripples to decreased. As a conclusion, this method of applying a feedback circuit will be used in this project.

CHAPTER 2

LITERATURE REVIEW

A switched-mode power supply, switching-mode power supply or SMPS, is an electronic power supply unit (PSU) that incorporates a switching regulator. While a linear regulator uses a transistor biased in its active region to specify an output voltage, an SMPS actively switches a transistor between full saturation and full cutoff at a high rate. The resulting rectangular waveform is then passed through a low-pass filter (typically an inductor and capacitor) to achieve an approximated output voltage. Advantages of this method include smaller size, better power efficiency, and lower heat generation. Disadvantages include the fact that SMPSs are generally more complex than linear supplies, generate high-frequency electrical noise that may need to be carefully suppressed, and have a characteristic ripple voltage at the switching frequency.

This project is based on designing a SMPS using a boost converter topology which it consist of basic converter that comply with a feedback cicrcuit. A feedback circuit is a controller of the basic converter where it was able to maintain its output due to disturbances. As the SMPS using a boost topology of converter, the circuit will step up the value of input voltage and give a higher output voltage. The circuit consists of high frequency switch, fast recovery diode, inductor and capacitor as shown in Figure 2.1

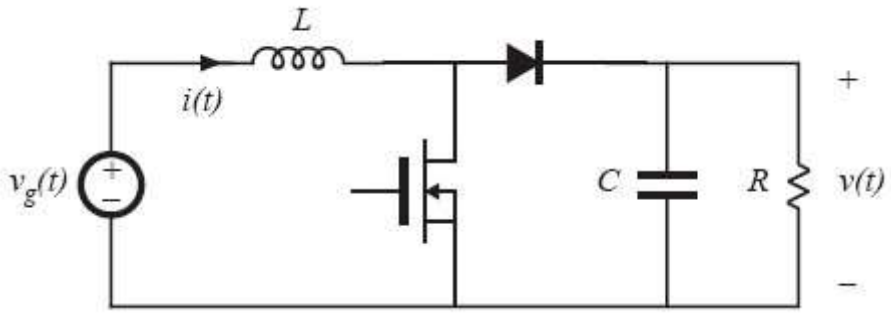

Figure 2.1: Basic boost converter topology

2.1 Power Stage Operation

2.1.1 During Switch ON

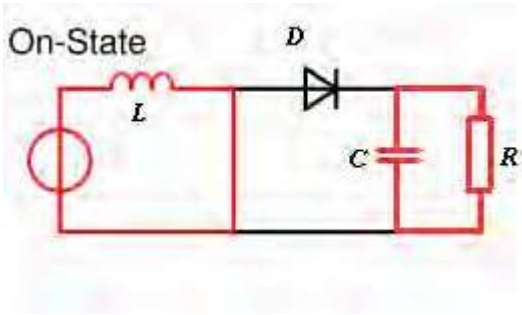

Figure 2.2: Current flow through the circuit during ON time

The input voltage will not be directly transferred to the output where at this state, the input voltage will charge up the inductor and store the energy inside. The characteristic of inductor is that it is able to store an energy at a certain capacity depends on its model just like capacitor but instead of voltage, the inductor is charge by current. The output voltage currently will be supplied by the capacitor.

2.1.2 During Switch OFF

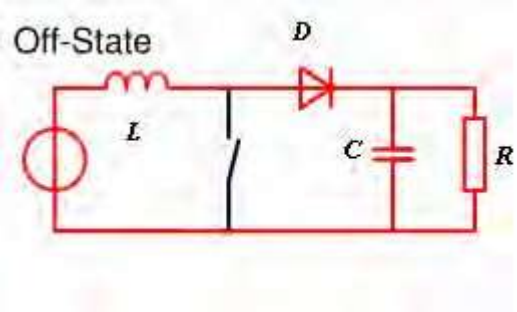

Figure 2.3: Current flow through the circuit during OFF time

The output currently is a accumulation of input voltage and inductor voltage. Inductor at this state act as a source to the circuit since it is fully charged and this had caused the output voltage to increase than the input voltage.

The operation of switching resulting a below graphs, where the output voltage is much larger value than the input voltage, and the output current is ramping up and down by the switching sequence.

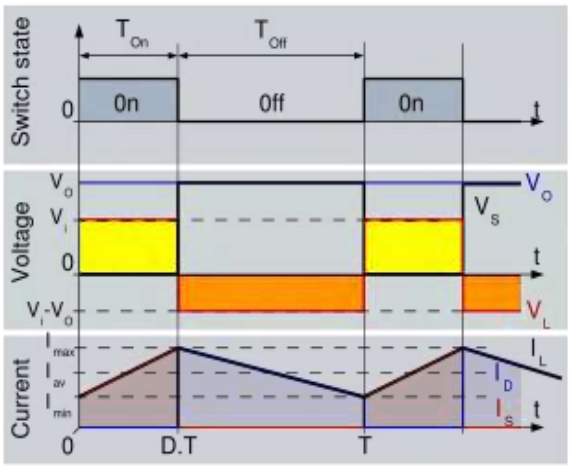

Figure 2.4: Waveforms of voltage and current during switching operation

As the current is ramping up and down, a ripple is resulted. A large may cause damage to the equipment that is connected to it, so a method of lowering the ripple as much as we could is conducted. We may increase the value of resistor, capacitor or switching frequency to decrease the ripples since the voltage ripple is indirectly proportional to resistor, capacitor, and switching frequency.

$$r = \frac{D}{RCf} \qquad (2\text{-}1)$$

Then, to control the output voltage, feedback circuit will be applied to the circuit where it will control the switching frequency of the converter and also controlling the duty cycle.

2.2 Feedback Control Circuit

Feedback control circuit consists of Error Amplifier and Pulse Width Modulator, PWM (Figure 2.4). The Error Amplifier acts as a sensor which will sense the output voltage meet the desired voltage or not. A voltage reference shall be set into a comparator and the output of the boost converter shall be connected to it. The comparator shall compare the reference voltage and the output voltage, to sense a

drop or increase in output voltage. If the voltage output does not meet the required voltage, the comparator shall give out a signal or output which acts as an error signal and going into the PWM. The PWM act as a controller to the switching of the MOSFET thus also called as duty cycle

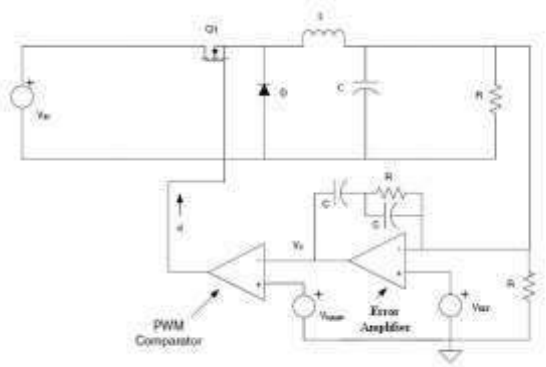

Figure 2.5: Feedback control circuit

. The simplest way to generate a PWM signal is the interceptive method, which use 2 type of signal to compare; a saw tooth or a triangle waveform and an error signal. A concept of generated PWM signal is shown in Figure 2.5.

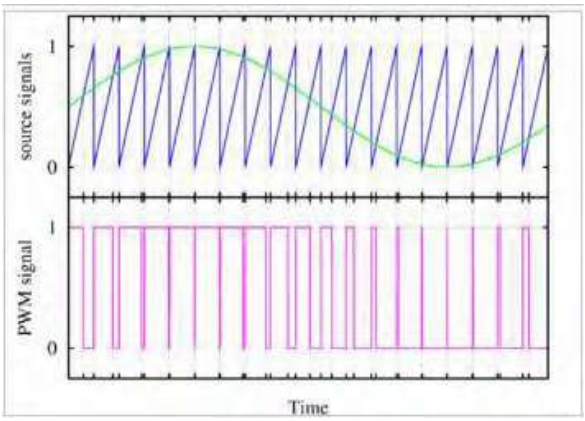

Figure 2.6: Concept of Generated PWM signal

A comparator is used to act as device that makes a comparison for both of these signals. If the error signal is much higher than the sawtooth, the output will be high and vice versa. The comparison of these wavefrom will create a PWM such in Figure 2.5. The width on time or duty cycle varies depend on error voltage a higher

amplitude create a larger value of D and the smaller the error voltage, create a smaller duty cycle.