International Journal of Electrical and Electronics Research ISSN 2348-6988 (online)
Vol. 3, Issue 2, pp: (626-630), Month: April - June 2015, Available at: www.researchpublish.com

High Efficiency DC-DC Power Electronics Converter for Testing Purpose

[1] K.Nishad, [2] R.Gunaa Jerre, [3]J.Mohan

[1,2] PG Scholar , [3]Assistant Professor, Department of Electrical and Electronics Engineering, Ranganathan Engineering College, Coimbatore, India

Abstract**: The goal of High Efficiency DC-DC Power Electronics Converter for testing purpose is to design and test a functional proof of concept of a high efficiency DC-DC power electronics converter. This paper presents the analysis and design of a new DC-DC converter applied to industrial heating purposes and testing of transformer oils, insulators etc. The proposed converter can operate with PWM using energy storage elements, filters etc., to generate output voltage greater than input. Moreover this topology is to generate transient output voltage and therefore current in the load is reduced for the same power of output. As a result, the converter efficiency is significantly improved to a far extent.**

Keywords: **DC-DC, PWM.**

I. INTRODUCTION

In many technical applications, it is required to convert a set voltage DC source into a variable-voltage DC output. A DC-DC switching converter converts voltage directly from DC to DC and is simply known as a DC Converter. A DC converter is equivalent to an AC transformer with a continuously variable turn's ratio. It can be used to step down or step up a DC voltage source, as a transformer. DC converters are widely used for traction motor control in electric automobiles, trolley cars, marine hoists, forklifts trucks, and mine haulers. They provide high efficiency, good acceleration control and fast dynamic response. They can be used in regenerative braking of DC motors to return energy back into the supply. This attribute results in energy savings for transportation systems with frequent steps. DC converters are used in DC voltage regulators; and also are used, with an inductor in conjunction, to generate a DC current source, specifically for the current source inverter. Efficiency, size, and cost are the primary advantages of switching power converters when compared to linear converters. Earlier we use a large step up transformer to step up the voltage and rectified using a bridge rectifier and hence system is so complicated to control and it uses many coolers to reduce the produced heat and hence the overall system cost is very high compared to the recent cost and now we are using SMPS technology in this project. The switching power converter efficiencies can run between 70-80%, whereas linear converters are usually 30% efficient. The DC-DC Switching Boost Converter is designed to provide an efficient method of taking a given DC voltage supply and boosting it to a desired value.

Power Electronics is the art of converting electrical energy from one form to another in an efficient, clean, compact, and robust manner for convenient utilization. The never ending drive towards smaller and lighter product poses serious challenges for power supply designers. The aim of the project is to design, test and implement a switched mode

Power supply (SMPS) circuit for AC to DC conversion, having a power MOSFET for switching operation and a PWM based feedback circuit to drive the MOSFET switch using NI MULTISIM circuit design environment and NIELVIS Breadboard.

II. GENERAL CONVERTER PERFORMANCE

The performance factors of efficiency, voltage and current regulation, and current ripple are strongly related to converter transfer function. In terms of *Efficiency*, Boost converters have the highest efficiency because for a given power they have the lowest output current hence lowest diode on-state voltage and copper *I*2*R* losses, while the buck converter has the highest output current whence the highest diode conduction voltage and *I*2*R* losses. Converters where inductor current comprises the input and output current have the highest *I*2*R* losses, hence lowest efficiency. In terms of *Voltage and current regulation*, the buck converter has the lowest output voltage (and increased output current), whence the diode voltage which is the largest of the three basic converters, represents a more significant proportion of the output voltage. Hence buck converters should employ silicon Schottky diodes. Mitigating the poor output voltage regulation of the buck converter, as seen in Fig. 6, it has the best current regulation, with the boost converter having the poorest current regulation at low current. That is, in open loop, the buck converter is best for accurately tracking the current transfer ratio (with an efficiency sacrifice), which is a fundamental aspect of current source input and output converters. Generally, good current regulation is expected because the current transfer function is independent of input and output voltages, which are effectively reduced due to series component voltage drops, thus affect voltage regulation. With regard to *Ripple current*, the higher circuit voltages, the boost converter tends to have the highest ripple current, while the buck converters have the lowest voltages, hence the lowest ripple currents. Although converters where the two inductors are effectively in series with the output or input tend to have a lower ripple current. Constant input ripple current, for a given δ, occurs when an inductor experiences only the input voltage, otherwise ripple current is regulation and/or capacitor voltage ripple dependent. Converters with significant ripple current increase at higher currents, have a capacitor voltage swing which tends to become exponentially shaped. Generally, the boost converter is used when continuous source current is required, without the use of energy source decoupling capacitance. It is, therefore, commonly interfaced to renewable sources. The disadvantage of the boost converter is that the controlled output voltage is always greater than the input voltage. The presented buck–boost converters offer not only continuous input and output current but controlled voltage down to zero, but with voltage polarity inversion. Nevertheless the buck–boost topologies offer an alternative first stage for PV and fuel cell applications. The buck converters, in a reversible form, may be viable in battery dc-link backup systems of grid connected PV and fuel cell applications.

III. CIRCUIT DESCRIPTION

High efficiency dc –dc power electronics converter for testing purpose is a new DC-DC converter applied to industrial heating purpose and testing of transformer oils, insulators etc. In mainly consists of five sections.

- Fundamental AC section
- Low voltage DC section
- High voltage DC section
- Power supply section
- Control Board or Display section

Fundamental AC section:

AC voltage 230V is directly given to the main board. Here in the section it consists of many protection circuits such as Thermistor and Varistor to control the high voltage and current produced in this circuit and also it contains a coil to reduce the harmonics produced in this section and the filtered output is given to a bridge diode. From these it converts to the required DC voltage.

Low voltage DC section:

The obtained voltage is then fed to filter capacitor from this we have two main function. One is to filter and other to increase the voltage to 330V DC. In some sections it requires less voltage and this is made low by using resistors. Here in this section too many protection circuits are used. The filter capacitor here used to 450V, 330uF. A filter capacitor is an electronic component that removes voltage or signal spikes in electronic circuits. Capacitors are used as filter devices due to their ability to absorb and effectively store electrical charges at predetermined values. This characteristic is used in a filter capacitor application to soak up or buffer voltage values which exceed set parameters.

High voltage DC section:

In this section it mainly consists of MOSFET and Chopper transformer. Using these two components the voltage is made high. In this section too many protection circuits are used to protect the MOSFET. The protection circuits include zener diode, resistor and capacitor and the chopper transformer output contains two output one lower output of 1000V and a high output voltage of 1400V. The 330V low voltage is converted to high voltage DC using chopper transformer and MOSFET using PWM technique. PWM is a modulation technique that controls the width of an electrical pulse, formally the pulse's duration based on modulator signal information. PWM is also used in efficient voltage regulators. By switching voltage to the load with the appropriate duty cycle, the output will approximate a voltage at the desired level. The switching noise is usually filtered with an inductor and a capacitor. PWM or Pulse Width Modulation is the modulation technique where frequency and Amplitude of the pulse signal is not varied and the pulse width or the duty cycle is varied to encode the information. A common use of PWM is to control the average current or Voltage input to a device. In this project the PWM Signal is generated as feedback control signal for driving the switching of the MOSFET switch. Switching period determines the voltage at the output. Thus if there is any change at the output Voltage, the corresponding change in the PWM pulse width will nullify its effect and the output voltage will be restored to desired value. The high output obtained from chopper transformer is given to power diodes. Where they are arranged in a form of Centre tapped and the output from power diodes are rectified using Centre tapped rectification method. In this resistors and capacitors are placed to get the Centre tapped voltage to be maintained as zero. The output voltage obtained here is not transient it is in Steady state. This output voltage is used to heating purpose and multi drive operation. In this project, the obtained the transient voltage is fed the capacitor which are connected in a particular manner. The output of the capacitor is connected to a flash tube. Whenever a transient voltage is needed by a tube, at that time capacitor discharges with the help of a trigger coil.

Power supply board:

Many control circuit of main board and display section are working in 5V DC. This 5V DC is produced by using power supply board. The 230V is supplied to 230/9-0-9 transformer to step-down the voltage level. This 9V is rectified using Centre tapped full wave rectifier.

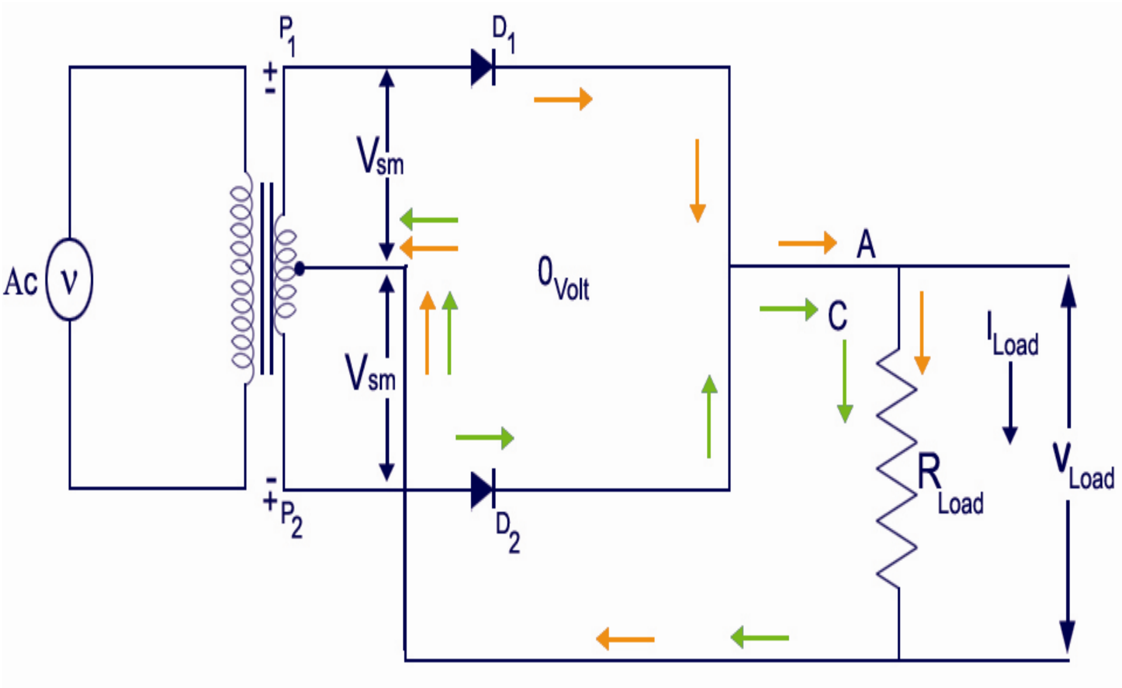

Fig.1 Centre-tap-full wave rectifier circuit

Control board or Display board:

The next section is control board or display section. This section is mainly used to control the trigger section and to show the display. How much trigger is given and when to give all are decided by this section by the user. This xenon flash tube contains a coil around it. It is used for the easy discharge. The main capacitor is placed in series parallel arrangement. Low-input current ripple is associated with topologies where the current controlling inductor experiences voltages related to capacitor voltage ripple as opposed to impressed stepped voltages due to a switching device. In mitigation, larger current ripple is associated with a faster transient response topology, better regulation, and higher efficiency.

International Journal of Electrical and Electronics Research ISSN 2348-6988 (online)
Vol. 3, Issue 2, pp: (626-630), Month: April - June 2015, Available at: www.researchpublish.com

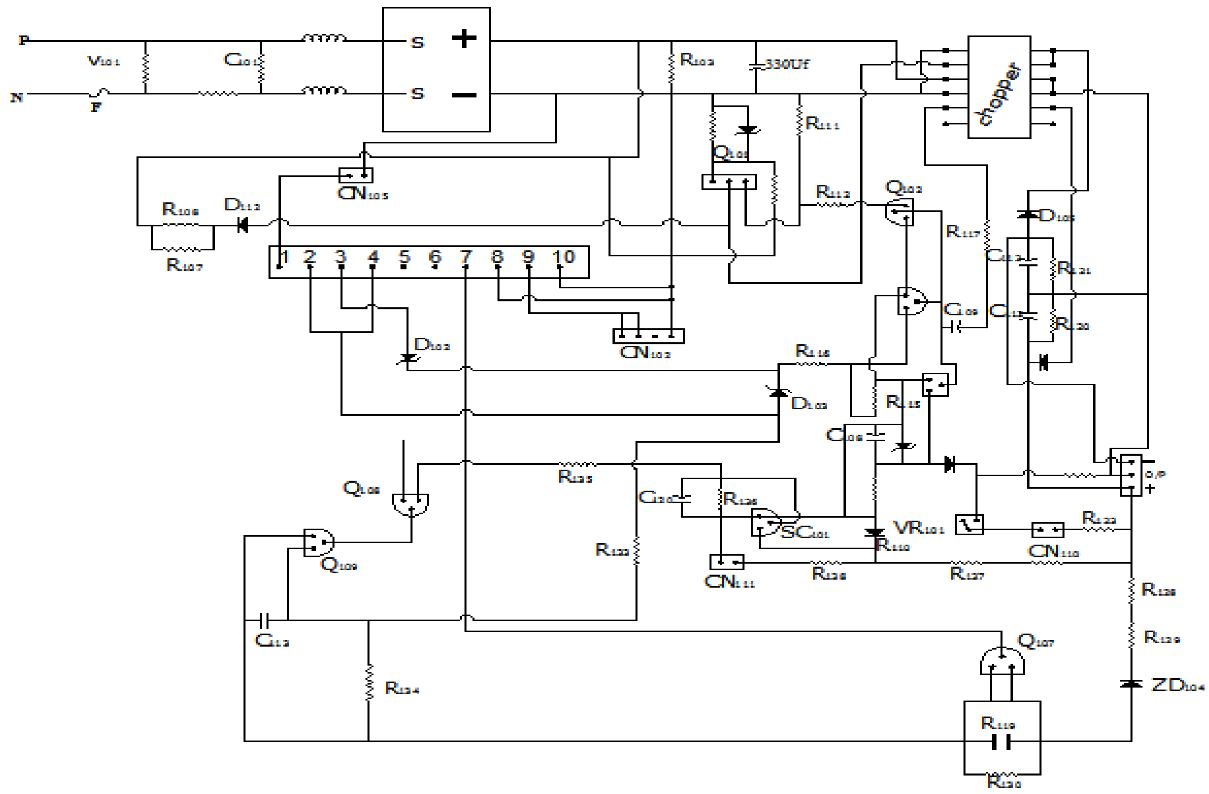

Fig 2: Main Circuit Diagram

IV. SIMULATION SOFTWARE

PSpice is a SPICE analog circuit and digital logic simulation program for Microsoft Windows. The name is an acronym for Personal SPICE – SPICE itself being an acronym for Simulation Program with Integrated Circuit Emphasis. During its development, PSpice has evolved into an analog mixed signal simulator. The software, now developed towards more complex industry requirements, is integrated in the complete systems design flow in OrCAD and Cadence Allegro. It includes features such as analysis of a circuit with automatic optimization, encryption, a model editor, support for parameterized models, auto-convergence and checkpoint restart, several internal solvers, a magnetic part editor, and support for Tabrizi core model for non-linear cores. SPICE (Simulated Program with Integrated Circuit Emphasis) is a general purpose software that simulates different circuits and can perform various analysis of electrical and electronic circuits including time domain response, small signal frequency response, total power dissipation, determination of nodal voltages and branch current in a circuit, transient analysis, determination of operating point of transistors, determinations of transfer functions etc. This software is designed in such a way so that it can simulate different circuit operations involving transistors, operational amplifiers (op – amp) etc. and contains models for circuit elements (passive as well as active). In PSpice, the circuit is first described to a computer by using a file called circuit file. It contains the circuit details, viz., and the information about source and commands for what to do and what to display as output. The PSpice accepts circuit file as an input and after executing commands, creates an output file to store results. However, the circuit to be analyzed is specified in terms of element names, element values different sources (voltage or current) and different parameters. PSpice calculates all nodal voltages and branch currents over a range of time interval by giving the output of their instantaneous values. It can also perform other operations as will be evident later. For circuits with variable frequency sources, AC analysis is used. Each circuit element is connected between two nodes. All nodes must be connected to at least two elements and therefore appear twice at least. The node 0 is predefined as ground. PSpice is a computer program itself so one need not have to do "programming" to use this software. The job is just to prepare the circuit file that instructs SPICE what to do. The circuit file consists of statements describing the circuit, specifying the analysis to be performed and statements to control the output formats and variables. PSpice is to do the simulation called for in the circuit file and stores the result in an output file. The results may be displayed and or printed.

The type of simulation performed by PSpice depends on the source specifications and control statements. The analyses usually executed in PSpice are as follows.

DC analysis:

It is used for circuits with time–invariant sources (e.g. steady-state dc sources). It calculates all nodal voltages and branch currents over a range of values. The types of dc sweep analyses and their corresponding. (Dot) commands are linear sweep, Logarithmic sweep and Sweep over List of values. These sweep types can also be nested by adding another set of parameter name and values at the end.

Transient analysis:

It is used for circuits with time variant sources (e.g., sinusoidal sources/switched dc sources). It calculates all nodes voltages and branch currents over a time interval and their instantaneous values are the outputs.

AC analysis:

It is used for small signal analysis of circuits with sources of varying frequencies. It calculates the magnitudes and phase angles of all nodal voltages and branch currents over a range of frequencies.

V. CONCLUSION

Thus we conclude that by using this high efficiency DC-DC power electronics converter we can generate a transient output voltage and their current in the load is reduced for the same output power and also it has an improved efficiency. And it's mainly used to generate transient output voltage and therefore current in the load is reduced for the same output power and also it has an improved efficiency .The advantages of the system are High efficiency, Low cost, Control of high voltage section is easy because the gate voltage or base voltage of MOSFET & transistor are very low and Construction is simple. It has wide applications in the areas of Industrial heating purpose, testing of transformer oils and Insulators, Melting of rubber surface.

REFERENCES

[1] Barry.W.Williams, "DC to DC Converters with continuous Input and Output Power", IEEE Transactions on Power Electronics ", Vol.28,No.5, May 2013, pp. 2307-2316.

[2] T. Suntio, J. Leppaaho, J. Huusari, and L. Nousiainen, "Issues on solargenerator interfacing with current-fed MPP-tracking converters," IEEE Trans. Power Electron., vol. 25, no. 9, pp. 2409–2419, Sep. 2010.

[3] M. A. Myrzik, "Novel inverter topologies for single-phase standalone or grid-connected photovoltaic systems," in Proc. IEEE Conf. Power Electron. Drive Syst., 2001, pp. 103–108.

[4] O. Hegazy, J. V. Mierlo, and P. Lataire, "Analysis, modeling, and implementation of a multidevice interleaved DC/DC converter for fuel cell hybrid electric vehicles," IEEE Trans. Power Electron., vol. 27, no. 11, pp. 4445–4458, Nov. 2012.

[5] C. N. Onwuchekwa and A. Kwasinski, "A modified-time-sharing switching technique for multiple-input DC–DC converters," IEEE Trans. Power Electron., vol. 27, no. 11, pp. 4492–4502, Nov. 2012.

[6] B. Zhao, Q. Yu, and W. Sun, "Extended-phase-shift control of isolated bidirectionalDC–DC converter for power distribution inmicrogrid," IEEE Trans. Power Electron., vol. 27, no. 11, pp. 4667–4680, Nov. 2012.

[7] N. Vazquez, H. Lopez, C. Hernandez, E. Rodriguez, R. Orosco, and J. Arau, "A grid connected current source inverter," Int. Conf. Clean Elect. Power, pp. 439–442, Jun. 9–11, 2009.

[8] P. P. Dash andM.Kazerani, "Dynamicmodeling and performance analysis of a grid-connected current-source inverter-based photovoltaic system," IEEE Trans. Sustainable Energy, vol. 2, no. 4, pp. 443–450, Oct. 2011.

[9] Suroso and T. Noguchi, "Novel single phase grid connected currentsource PWM inverter with harmonic suppression," in Proc. IEEE 2nd Int. Power Energy Conf., pp. 1373–1378, 2008.

[10] R. J. Wai and R. Y. Duan, 'High-efficiency DC/DC converter with highvoltage gain", in lEEE Proc. Electr. Power Appl., Vol. 152, No.4, July 2005, pp. 793-802.

[11] R. J. Wai and R. Y. Duan, "High-efficiency DC/DC converter with high voltage gain", in lEEE Proc. Electr. Power Appl., Vol. 152, No.4, July 2005, pp. 793-802.

[12] Woywode, O.; Guldner, H. "Application of Statistical Analysis to DC-DC Power Electronics Conference, IPEC 2000, Tokyo,Japan 2000.