Design and Simulation of Voltage Mode DC to DC Converter using LTspice

Harshitha.C[1] Vipula Singh[2],

[1] Dept of ECE , RNS Institution of Technology, harshu02.1996@gmail.com,
[2] HOD, Dept of EC ,RNS Institution of Technology, vipulasingh@yahoo.com

---***---

Abstract— In this research work, we have designed and simulated, a MOSFET-switch Boost, Buck DC-DC converter working in a voltage mode. This converter benefits from the input current waveform's consistency, allowing for the most efficient use of renewable energy sources. We analyzed the converter in both continuous and discontinuous conduction modes. A switching model of LT-SPICE SPA11N60C3, for the simulation work, is used for the calculations. The Simulation has been carried out for the circuit with a boost converter design is 30V output voltage by considering 12V input voltage 20KHz switching frequency and boost converter design is 24V output voltage by considering 48V input voltage 20KHz switching frequency for both the converters. The results are verified through LT spice software. The Simulation results establish that the proposed design is capable to produce continuous output voltages for different input voltages anddifferent loads.

Keywords: DC/DC Boost, Buck Converter, Quadratic Voltage gain

I. INTRODUCTION

Isolated and non-isolated DC-DC topologies are the two most common types of topologies. In isolated topologies, the turn ratio of the coils plays a key role in increasing the gain in voltage and self-determine the high duty cycle value. The isolation provided by the presence of the high- frequency transformer protects the responsive loads from input source errors. Despite the benefits indicated above, adding a magnetic core has disadvantages such as increased volume, mass, expense and loss in power. Furthermore, the disconnects between the input current and the derivation inductors show the use of insufficient circuits, increasing the number of circuit components and the converter's difficulty. When there is no compelling reason to use an isolated circuit, it is therefore a sensible choice to use a non-isolated topology. The buck and boost converters can step down and step up the voltage level of the source voltage among the basic non isolated converters, such as buck, boost and buck-boost converters. The main disadvantages of buck-boost converters are the discontinuous input and output currents, as well as the negative opposition of the output voltage. The 50 percent duty cycle is the ideal choice since it stores energy in the inductor during the first mode and then releases it to the load. As a result, in such a case, the buck-boost converter performs a pass-through rather than a step up. The input voltage can be stepped up using a boost converter. Higher levels of voltage gain can theoretically be accomplished by increasing the duty cycle and its proximity to unity.

II. LITERATURE REVIEW

In this paper author has explained eefficiency, size, and cost are the basic advantages of switching power

converters when compared to linear converters. The switching power converter efficiency can run between 70- 80%, but linear converters are 30% efficient. The DC-DC Switching Boost Converter is designed to make available DC voltage supply and boost it to a needed voltage.[1]

In this paper, the authors have presented a converter that provides a high voltage transfer gain with a lower duty cycle.American Power Systems, high-efficiency DC-DC boost converter increases output voltage and steps down output current. No need for additional batteries which reduces the space in the engine compartment and overall cost. [2]

DC-DC boost converters for diverse mobile applications, including non-standard commercial vehicles, light tactical vehicles, coaches, work trucks, and others.[3]

In this paper, the authors have shown the results, and analysis for the buck, boost and buck-boost converters by varying different input parameters like an inductor, capacitor, and switching frequency in order to observe the change in the output has been simulated in PSPICE.[4]

In this paper, the authors have implemented and simulated buck converter design in the cadence virtuoso tool with 180nm Technology as well as hardware. which uses Pulse width Modulation (PWM) technique in voltage mode. The switches used are PMOSFET and NMOSFET. A comparator and an op-amp with a large gain and good stability have been designed for fulfilling the closed loop requirements of a converter.[5]

In this paper, the authors have discussed the solar system is modeled with QBBC, Cascaded Buck and Buck-Boost Converter and with a single Buck-boost converter. The comparative analysis between the DC-DC converters, based on switching losses, and efficiency. Designing and Simulation are done through MATLAB/Simulink.[7]

In this paper, the authors have discussed the hybrid converter for a Buck-Boost Converter with Augmented Sinusoidal Pulse Width Modulation (ASPWM) it provides both AC and DC voltages using a single DC source as input. The DC voltage gain of a buck-boost converter acts as a boost converter for all ranges of DC offset under switching frequency selection. The circuit is simulated in MATLAB/Simulink for various load conditions.[8]

In this paper, the authors have discussed the increasing demands and rigor required for the functioning ofelectronic circuits, modern simulators must allow for more

complex analyses. MATLAB tool is used to analyse periodic circuit equations is more efficient and to closer look at the actual operating conditions.[9]

In this proposed system the authors have discussed the DC-DC Buck-Boost power converters for renewable energy sources, industrial equipment and etc. DC-DC Buck-Boost power converter topology used to extend output voltage using a single power semiconductor. Additionally, this converter operates with reduced voltage stress in the power semiconductor this has proven through simulation with experimental tests.[10]

In this paper, the authors have discussed the design of a simple powerful soft switching scheme for a DC-DC Boost Converter with a closed loop control by a single switch using PI controller. A 50W/50KHz soft switched PWM Boost converter is simulated using PSIM.[11]

In this paper, the authors have explained the small-signal averaged models for synchronous dc-dc converters operating at variable switching frequencies varying from 100khz to 700khz using PID controller.[12]

III. PROPOSED SYSTEM

The proposed system is designed for regulated output voltage of the buck and the boost converters, the output of the buck and boost converter is given to the voltage divider

circuit to divide the voltage and matched to the reference voltage (Vref).

The proposed system shows a regulated dc output voltage because the output of the buck and boost converter is an unregulated dc voltage so that output will be given to the error amplifier with the Type II compensator circuit to produce the stabilized output voltage and to remove the error in the output signal and then the output of the error amplifier is given to the comparator one of the inverting terminal of the comparator and other input of thecomparator will be of ramp signal generator for generating the output signal PWM for the switching control of the MOSFET(on/off).
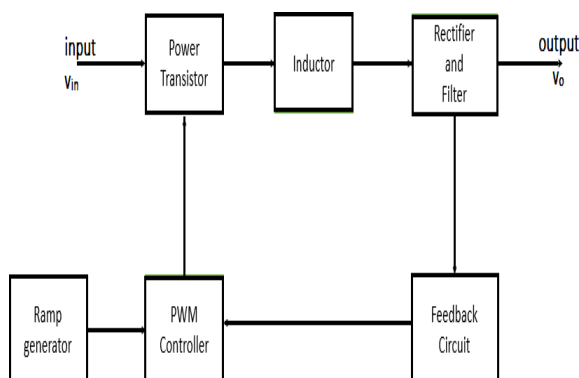

Figure 1: General Block Diagram of DC-to-DC converters

Figure 1 shows the general block diagram of the DC-to- DC converters. Input of the DC or AC is given to the power switch like BJT, MOSFET and IGBT for the switching. The design for the inductor, capacitor and the design of the compensator circuit is designed as the per the derived equation in the basic dc to dc converters. Here the ramp signal generator is used for the comparison of the output of the error amplifier circuit. This was the avenue used for the first switching regulator designs and it served the production well for many years. The major characteristic of this layout there is only one voltage feedback path and a pulse width modulation is achieved by comparing the voltage of an error amplifier signal to a constant ramp signal generator. Limiting of a current must be done separately. Figure 2 and 3 both represent a typical buck converter and boost converter circuit with feedback with voltage-mode control. The output voltage is in contrast to the reference voltage, and the error between the two is amplified by the compensated   operational amplifier.

1. Design for Buck Converter: Input Voltage (Vin) = 48V, Output Voltage (Vout)= 24V for the duty cycle of 50% is calculated using the below equations.

Duty Ratio(D): The desired duty cycle ratio is calculated as follows along with other required converter parameters

$$D = \frac{Vout}{Vin} \qquad \text{.......(1)}$$

Inductor

$$Lmin = 1 - \frac{R(1-D)}{2f} \qquad \text{.......(2)}$$

Capacitor

$$C = \frac{1-D}{8Lf^2(\Delta Vout/Vout)} \qquad \text{.......(3)}$$

Average   Current(IL)

$$Il = \frac{Vout}{R} \qquad \text{.......(4)}$$

The below Figure 2 shows the Buck Converter with closed loop containing the Feedback Amplifier (error amplifier with the type II compensator and comparator) Circuit using LT spice for the simulation of the circuit. Starting with the value of switching frequency, the corner frequency (fc) is chosen as 1/10th of switching frequency. The values of pole zero frequency (fz) are

determined (Equation 5) and later the RC values can be suitably calculated (Equation 6). The calculations are shown for 10KHz switching frequency. Choose the value of cross over frequency to be 1/10th of the switching frequency. Using the switching we have arrived at pole and zerofrequencies.

$$Fz1 = \frac{1}{2\pi R1c1} \qquad \ldots\ldots\ldots\ldots (5)$$

The values of compensating resistors and capacitors canbe calculated using the above frequencies

$$Fp2 = \frac{1}{2\pi R1c2} \qquad \ldots\ldots\ldots\ldots (6)$$
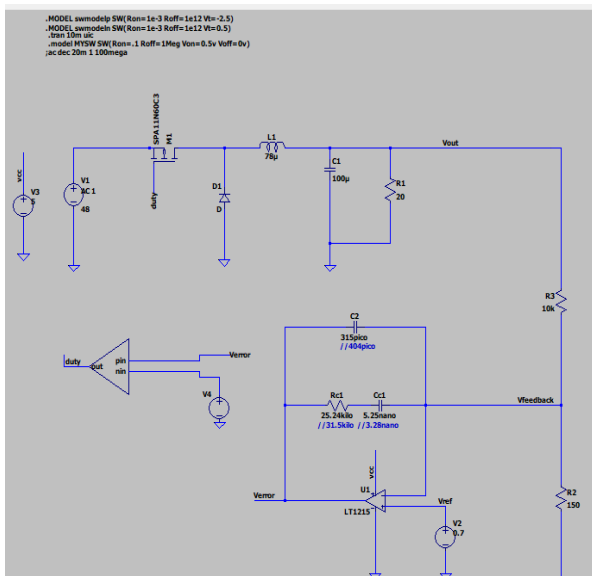
Figure 2: Buck Converter with Feedback Amplifier Circuit using LT spice(tool-snapshot)

2. Design for Boost Converter: Input Voltage (Vin) = 12V, Output Voltage (Vout)= 30V for the duty cycle of 60% is calculated using the below equations.

Duty Ratio(D): The desired duty cycle ratio is calculatedas follows along with other required

$$D = 1 - \frac{Vout}{Vs} \qquad \ldots\ldots.(1)$$

Inductor

$$Lmin = \frac{d(1-d)^2}{2f}R \qquad \ldots\ldots.(2)$$

The below Figure 3 shows the Boost Converter with closed loop containing the Feedback Amplifier (error

10KHz switching frequency. Choose the value of cross over frequency to be 1/10th of the switching frequency. Using the switching we have arrived at zero and polefrequencies.

$$Fz1 = \frac{1}{2\pi R1c1} \qquad \ldots\ldots\ldots\ldots (5)$$

The values of compensating resistors and capacitors can be calculated using the above frequencies

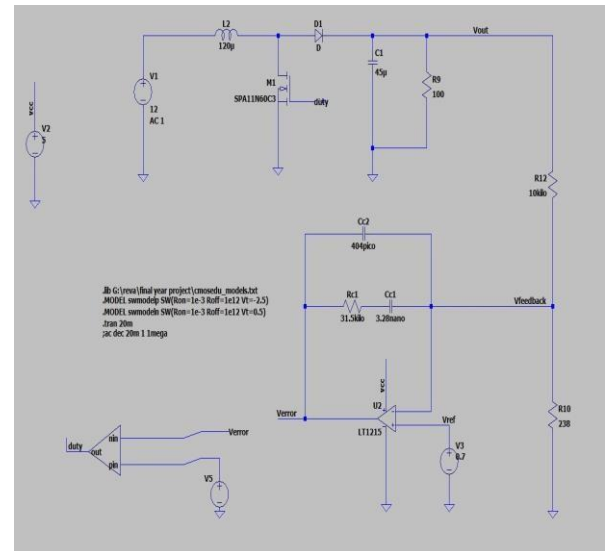$$Fp2 = \frac{1}{2\pi R1c2} \qquad \ldots\ldots\ldots\ldots (6)$$

Figure 3: Boost Converter with Feedback Amplifier Circuit using LT spice(tool-snapshot)

In both the circuits we have used Type II compensator in the error amplifier to decrease the steady state accuracy ofthe output voltage by decreasing ripple voltage this is done to make the output voltage stabilities (minimize the overshoot of the output waveform).

amplifier with the type II compensator and comparator) Circuit using LT spice for the simulation of the circuit. Starting with the value of switching frequency, the corner frequency (fc) is chosen as 1/10th of switching frequency. The values of pole zero frequency (fz) are determined (Equation 5) and later the RC values can be suitably calculated (Equation 6). The calculations are shown for
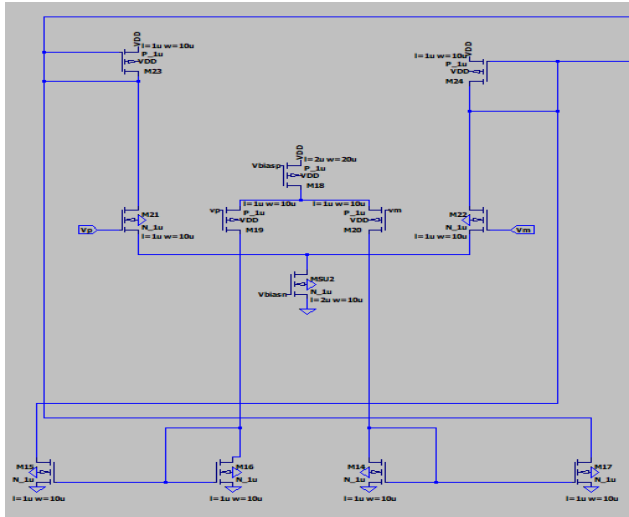

Figure 4: Comparator design symbol

Figure 4 shows the design and a symbol of the comparator. A triangle wave generator and comparator are

used in this circuit to generate a pulse width modulated (PWM) waveform with a duty cycle of 60% that is inversely proportional to the input voltage. A triangle waveform is generated by an op-amp and comparator, which is sent into the inverting input of a second comparator. A PWM waveform is created by advancing the input voltage to the non-inverting comparator input. To ensure high precision and linearity of the output voltage negative feedback of the PWM waveform to an error amplifier is used.
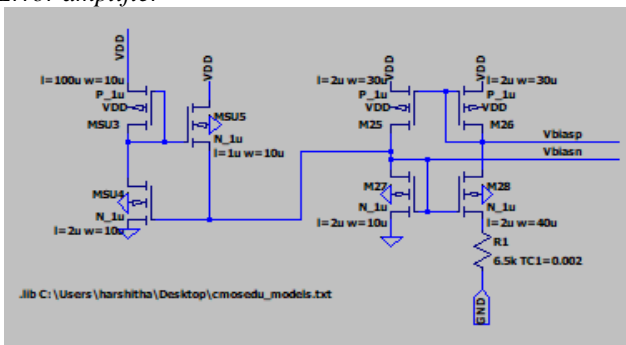
A. Error amplifier

Figure 5: Error amplifier design

Figure 5 shows the error amplifier design of LT1215 taken in the circuit. 3MHz, 50V/µs, Single-Supply Dual and Quad Precision Op-amps are taken to design the LT1215 circuit. An error amplifier is most typically encountered in feedback unidirectional voltage control circuits, where the sampled output voltage of the circuit under the rule is relayed back and contrasted to a stable reference's value.. An error amplifier does exactly what its name implies, it boosts an erroneous signal. The fluctuation between a reference signal

and the input signal causes this mistake. It's also possible to think of it's as the difference between the two points.

IV. SYSTEM DESIGN

The system is designed to the regulated output voltage of the buck and the boost converters, the output of the buck and boost converter is given to the voltage divider circuit to divide the voltage, The proposed system shows a regulated dc output voltage because the output of the buck and boost converter is an unregulated dc voltage so that output will be given to the error amplifier with the Type II compensator circuit to produce the stabilized output voltage and to remove the error in the output signal and then the output of the error amplifier is given to the comparator one of the inverting terminal of the comparator and other input of the comparator will be of ramp signal generator for generating the output signal PWM for the switching control of the MOSFET(on/off). The circuit is simulated using a LTspice simulator for checking the circuit stability and output constraints of the buck and the boost converter circuit. In Figure 1,2,3,4 & 5 shows the design of the closed loop converter of the buck and boost with the constant output voltage with respect to the constant duty cycle designed for both the converters. Table 3 & 4 are tabulated to show the change in the input voltage with the constant output voltage.

I. EXPERIMENTAL RESULT

To show the regulated output voltage of the buck and boost converters, the output of the buck and boost converter is an unregulated dc voltage so to converter the unregulated dc voltage to generate the regulated dc output voltage we have used a feedback circuit that FB network contains error amplifier to remove the error in the output signal and then the output of the error amplifier is given to the comparator that will generate the PWM signal for the MOSFET switch. Figure 2 shows the Buck Converter with Feedback Amplifier Circuit using LT spice and the snap short of the circuit is shown in the proposed system. Figure 3 shows the Boost Converter with Feedback Amplifier Circuit using LT spice and the snap short of the circuit is shown in the proposed system. We have done a basic converter that is a buck and boost converter circuit after that further we implemented the feedback-controlled buck and boost converter. In the basic circuit, input is constant by varying duty ratio we get varied output voltage all those values are tabulated in the four tables.

Table 1: Buck converter open-loop output voltage

Input voltage in volts	Duty cycle	Output voltage in volts
24	10%	1.5
	20%	3.8
	30%	6.3
	40%	8.5
	50%	11
	60%	12.8
	70%	14.5
	80%	16
	90%	18.5
	100%	20

Table 3: Boost converter closed-loop output voltage

Input voltage in volts	Duty cycle	Output voltage in volts
12	10%	13
	20%	15
	30%	18
	40%	21
	50%	24
	60%	35
	70%	46
	80%	68
	90%	125
	100%	135

Table 3: Buck converter closed-loop output voltage

Input voltage in volts	Input voltage in volts
48	24
46	
44	
42	
40	
38	
36	
34	
32	
30	

Table 4: Boost converter closed-loop output voltage

Input voltage in volts	Input voltage in volts
12	30
15	
18	
20	
22	
24	
26	
28	
32	
34	

Table 1,2,3 and 4 shows the Comparison between a buck and a boost converter of the previous and proposed system which is an open-loop and a closed-loop circuit. In the open-loop, the input voltage is kept constant with a variation of the duty ratio the output voltage will be varied but in the closed-loop output will be constant with respect to certain variations of the input voltage but the duty cycle is designed based on design constraints (constant dutycycle). So now all the power chargers, in the industrial, commercial and in the economical everyone use power management chips to reduce the area, power dissipation and increase the speed.

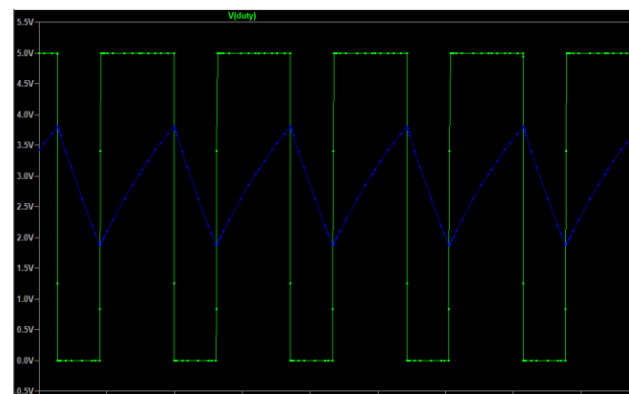Figure 6: Plot of duty cycle, Input voltage, Feedback voltage, voltage Reference Output Voltage

Figure 7: Inductor and a capacitor waveform

Figure 7 shows the waveform of the inductor and the capacitor. When the rising edge of the pulse shows the rising edge of the capacitor current and vice versa.

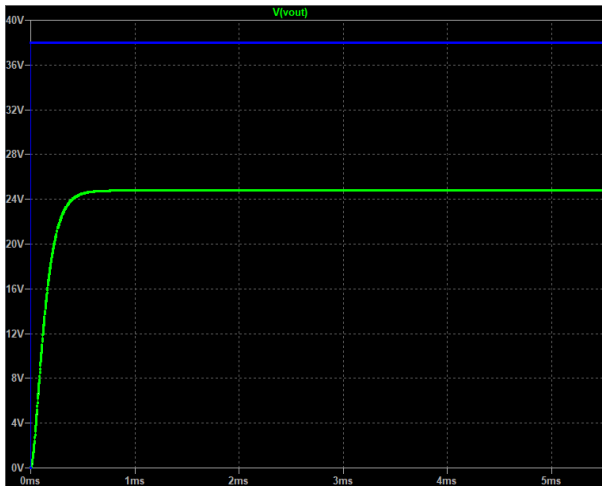Figure 8: Output voltage of buck converter

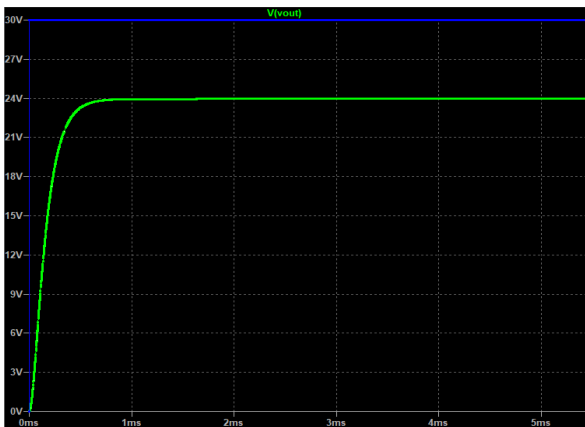Figure 9: Varying input voltage of buck converter to

38v

Figure 10: Varying input voltage of buck converter
30v

Figure 8, 9 & 10 shows the constant output voltage of the buck converter and varying the input voltage in accordance with the load resistor by varying the input voltage the output is constant up to the certain condition. In this we have set the input voltage as 48v, 38v, 30v and the output voltage is 30v. Setting the Equivalent series resistance (ESR) value by equation it will reduce the ripple voltage in the output waveform.
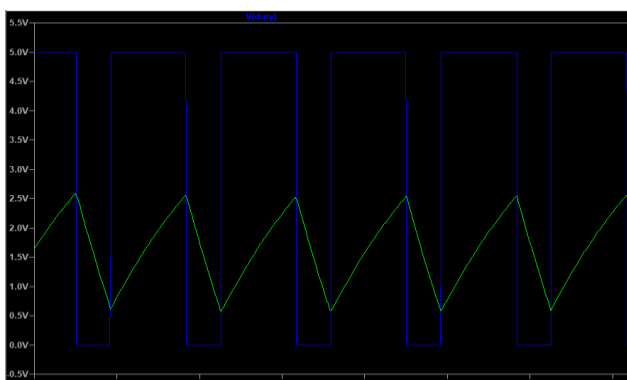

Figure 12: Inductor and a capacitor waveform

Figure 12 shows the waveform of the inductor and the capacitor. When the raising edge of the pulse shows raising edge of the capacitor current and vice versa.
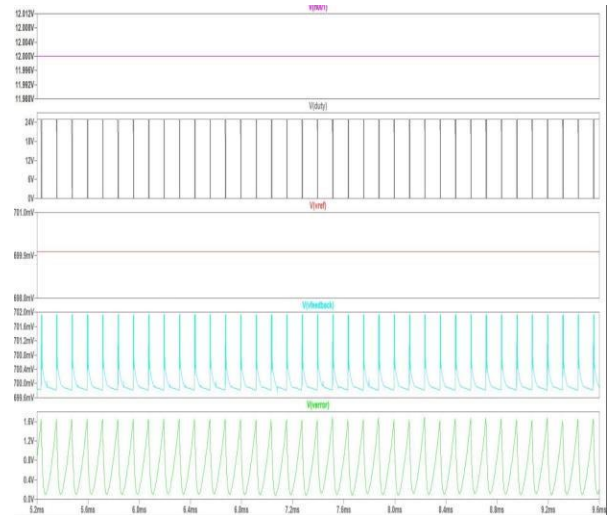

Figure 11: Plot of duty cycle, Input voltage, Feedback voltage, voltage Reference &error voltage

Figure 11 shows the plot of duty cycle that is generated by the comparator, Input voltage in the continuous voltage can be varied, Feedback voltage is divided and given the error amplifier, voltage Reference is given 0.7 for the error detection and the Output Voltage is constant up to some required input voltage.

Figure 13, 14 & 15 shows the constant output voltage of the boost converter and varying the input voltage in accordance with the load resistor by varying the input voltage the output is constant up to the certain condition. In this we have set the input voltage as 12v, 15v, 18v and the output voltage is 30v. Setting the Equivalent series resistance (ESR) value by equation it will reduce the ripple voltage in the output waveform.
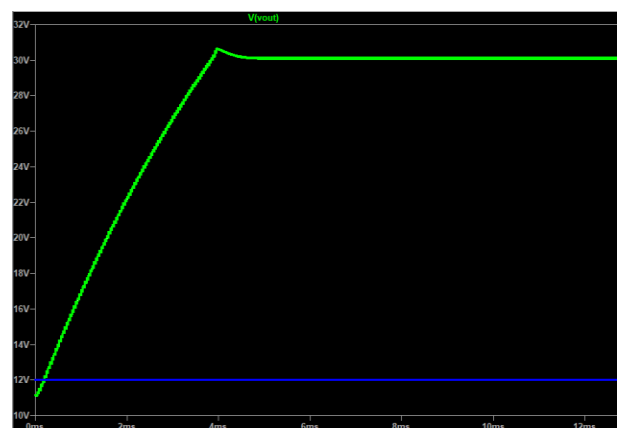

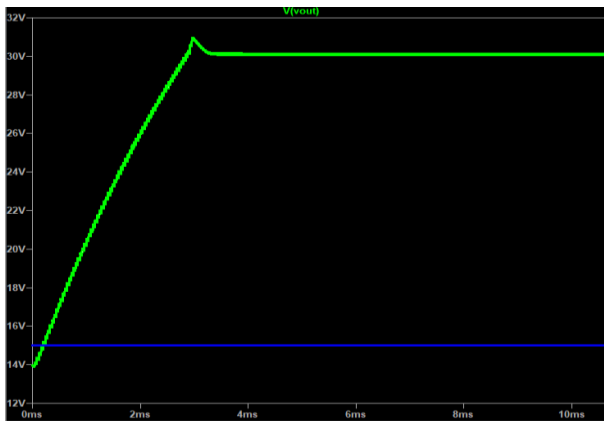Figure 13: Output Voltage for boost converter

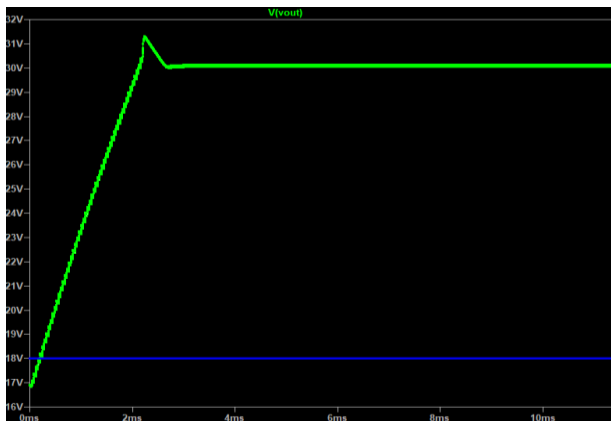Figure 14: Varying input voltage of boost converter 15v

Figure 15: Varying input voltage of boost converter 18v

V. CONCLUSION

This paper outline is to develop a DC-DC converters operating in the continuous conduction mode with a regulated dc voltage is working according the design. The circuit is working for the designed variable input voltage, the duty cycle is constant for the efficiency of the output voltage it is observed as per the PWM required for switching control of MOSFET. The circuit model, created using LTSPICE tool is working as per the design. The duty ratio, input voltage and current are all time averaged values. This model is helpful in doing the frequency analysis. The output voltage is also shown which is regulated to 24v for buck converter and 30v for boost converter.

REFEENCES

[1] Poorali, Behzad, and Ehsan Adib. "Analysis of the integrated SEPIC-flyback converter as a single-stage single-switch power-factor-correction LED driver." IEEE Transactions on Industrial Electronics 63.6 (2016): 3562- 3570.

[2] Sudharshan, K.M. and Divakar, B.P., Design and Simulation of Error amplifier used power management chips. International Journal of Recent Technology and Engineering" ISSN, pp.2277-3878.

[3] Mallik, Ayan, Bryan Faulkner, and Alireza Khaligh. "Control of a single-stage three-phase boost power factor correction rectifier." Applied Power Electronics Conference and Exposition (APEC), 2016 IEEE. IEEE, 2016.

[4] Wang, Chien-Ming, et al. "Design and realization of a zero-voltage transition pulse-width modulation interleavedboost power factor correction converter." IET Power Electronics 8.8 (2015): 1542-1551

[5] K.M. Sudharshan, Anilkumar C.S, Pallavi.R, Yatheesh H G, Santhosh, S.V. SivaReddy, "Design and Simulation of DC-DC Buck Converter using Cadence Tool", 2018 3rd IEEE International Conference on Recent Trends in Electronics, Information & Communication Technology (RTEICT-2018), MAY 18th & 19th 2018.

[6] Lovepreet Kaur Grewal and Navdeep Kaur Brar,"Analysis, Design and Simulation of DC-DC Boost Converter", International Journal For Research In Emerging Science And Technology, Volume-3, Issue-7, Jul-2016

[7] Mohammed Abdulla Abdulsada, "Analysis, Design and Simulation of 12v/24v Boost Switching Converter", International Journal of Engineering Sciences & Research Technology, Volume 3, no.6, June 2014, pp. 836-846 Chakraborty,

[8] Hafinaz A. Lopa, S. Hossain, M. K. Hasan, T. K Chakraborty, "Design and Simulation of DC-DC Converters", International Research Journal of Engineering and Technology, Volume 3, Issue 1, Jan-2016

[9] Robert Priewasser, Matteo Agostinelli, "Modeling,Control, and Implementation of DC–DC Converters for Variable Frequency Operation" IEEE transactions on power electronics, vol. 29, no. 1, January 2014. Pages 287-301.

[10] Rashid, M. H. (Ed), (2007): Power Electronics Handbook: Devices, Circuits, and Applications. Florida: Elsevier Inc.

[11] Mohammed Abdulla Abdulsada, Analysis, Design and Simulation of 12v/24v Boost Switching Converter‖, International Journal of Engineering Sciences & Research Technology, Volume 3, no. 6, June 2014, pp. 836-846

[12] Kondo, Ken-ichi, and Jumpei Baba. "Reactive power control by use of boost critical conduction mode power factor correction converter for suppressing voltage rise in distribution network." Power Electronics and Applications (EPE'14-ECCE Europe), 2014 16th European Conference on.IEEE, 2014.

[13] Sudharshan, K.M. and Divakar, B.P., Frequency Analysis of Single Stage Boost PFC using Average Model. Journal of Advanced Research in Dynamical & Control Systems, 10.

[14] António P. Martins and António M. Cardoso, "Input Current Distortion and Output Voltage Regulation of the Boost PFC Converter Operating with Different Control Methods", International Conference on Renewable Energies and Power Quality (ICREPQ-2012)

[15] Chiu, Huang-Jen, et al. "A single-stage LED lamp driver with low DC bus voltage for general lighting applications." International journal of circuit theory and applications 39.11 (2011): 1161-1175.