EXPERIMENT 03

1.0 AIM OF THE EXPERIMENT:

Simulation of Buck-Boost Converters using ngSPICE/PSPICE

1.1 SOFTWARE/APP REQUIRED:

NgSpice 35, KiCAD 5.1

1.3 THEORY:

1.3.1 Buck-Boost Converter

A buck-boost converter provides an output voltage that may be less than or greater than the input voltage hence the name “buck-boost”; the output voltage polarity is opposite to that of the input voltage. This converter is also known as inverting regulator. The circuit arrangement of a buck-boost converter is shown is figure 1.

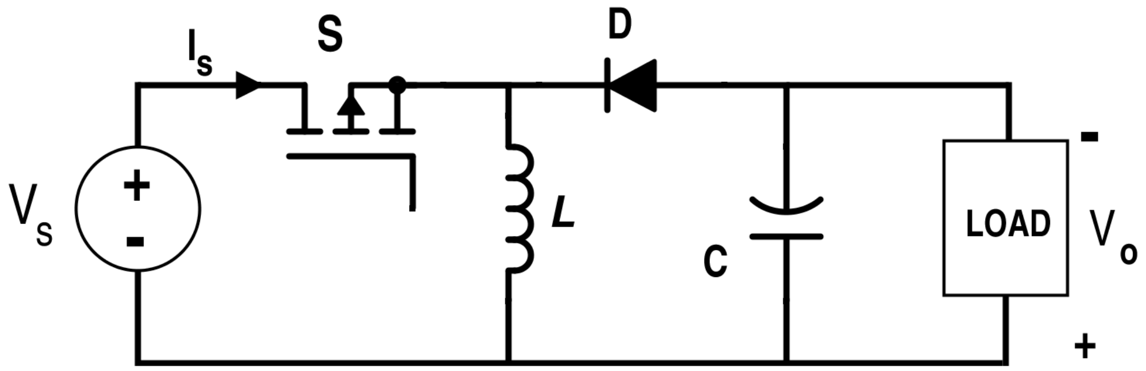


Fig. 1 Circuit Diagram of Ideal Buck-Boost Converter

1.3.2 Circuit Operation

Circuit Operation & Wave forms

Mode: I

The Switch is ON and therefore represents a short circuit ideally offering zero resistance to the flow of current so when the switch is ON all the current will flow through the switch and the inductor and back to the DC input source. The inductor stores charge during the time the switch is ON and when the solid-state switch is OFF the polarity of the Inductor reverses so that current flows through the load and through the diode and back to the inductor. So, the direction of current through the inductor remains the same.

Let us say the switch is on for a time T_{ON} and is off for a time T_{OFF}. We define the time period, $T = T_{ON} + T_{OFF}$, and the switching frequency,

$$f_{switching} = \frac{1}{T}$$

Let us now define another term, the duty cycle $D = \frac{T_{ON}}{T}$

Let us analyse the **Buck Boost converter** in steady state operation for this mode using KVL.

$$\therefore V_{in} = V_L$$

$$\therefore V_L = L\frac{di_L}{dt} = V_{in}$$

$$\frac{di_L}{dt} = \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{DT} = \frac{V_{in}}{L}$$

Since the switch is closed for a time $T_{ON} = DT$ we can say that $\Delta t = DT$.

$$(\Delta i_L)_{closed} = \left(\frac{V_{in}}{L}\right) DT$$

While performing the analysis of the Buck-Boost converter we have to keep in mind that

1. The inductor current is continuous and this is made possible by selecting an appropriate value of L.
2. The inductor current in steady state rises from a value with a positive slope to a maximum value during the ON state and then drops back down to the initial value with a negative slope. Therefore, the net change of the inductor current over any one complete cycle is zero.

Mode: II

In this mode the polarity of the inductor is reversed and the energy stored in the inductor is released and is ultimately dissipated in the load resistance and this helps to maintain the flow of current in the same direction through the load and also step-up the output voltage as the inductor is now also acting as a source in conjunction with the input source. But for analysis we keep the original conventions to analyse the circuit using KVL.

Let us now analyse the **Buck Boost converter** in steady state operation for Mode II using KVL,

$$\therefore V_L = V_o$$

$$\therefore V_L = L\frac{di_L}{dt} = V_o$$

$$\frac{di_L}{dt} = \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{(1-D)T} = \frac{V_o}{L}$$

Since the switch is open for a time

$$T_{OFF} = T - T_{ON} = T - DT = (1 - D)T$$
$$\Delta t = (1 - D)T$$
$$(\Delta i_L)_{open} = \left(\frac{V_o}{L}\right)(1 - D)T$$

It is already established that the net change of the inductor current over any one complete cycle is zero.

$$\therefore (\Delta i_L)_{closed} + (\Delta i_L)_{open} = 0$$
$$\left(\frac{V_o}{L}\right)(1 - D)T + \left(\frac{V_{in}}{L}\right)DT = 0$$
$$\frac{V_o}{V_{in}} = \frac{-D}{1 - D}$$

We know that D varies between 0 and 1. If D > 0.5, the output voltage is larger than the input; and if D < 0.5, the output is smaller than the input. But if D = 0.5 the output voltage is equal to the input voltage.

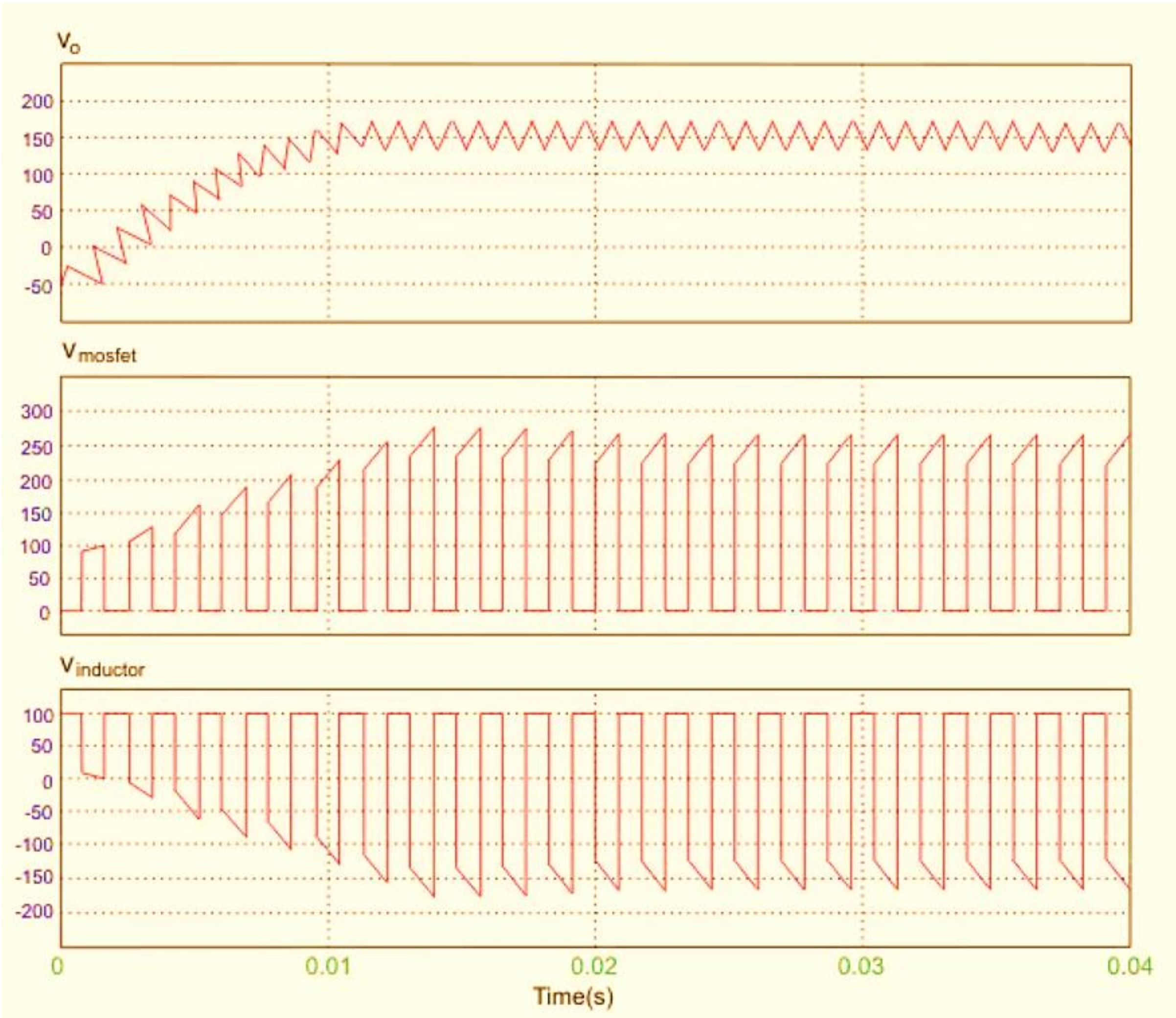


Fig. 2 Waveform of Ideal Boost Converter

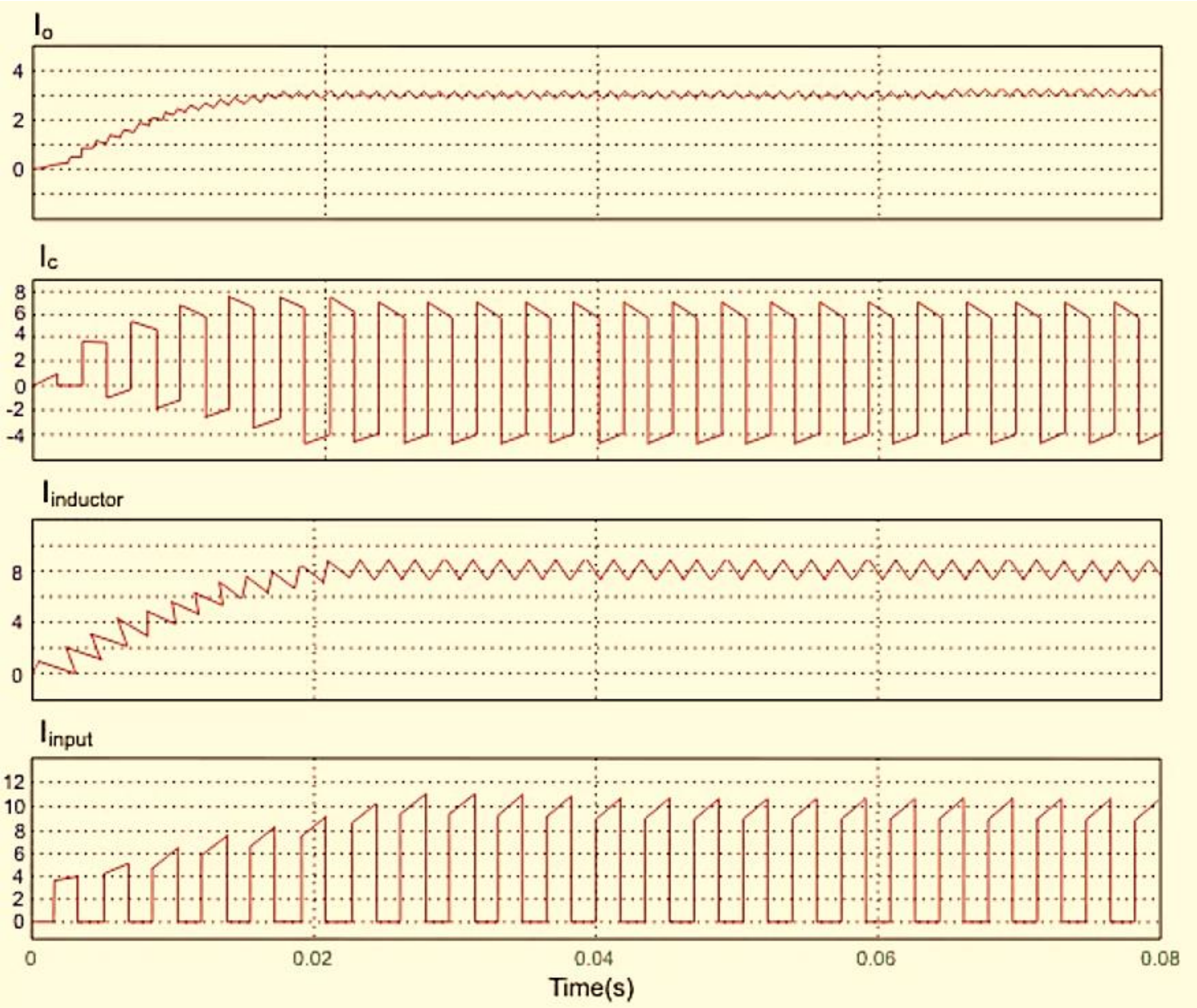


Fig. 3 Current Waveform of Ideal Buck-Boost Converter

1.3.3 Input Output Relationship

$$V_o = -V_s*(\frac{d}{1-d})$$

$$I_{avg} = -I_s*(\frac{d}{1-d})$$

|**Vo**| < **Vi** for **0** ≤ **d** < **0.5 (Buck Operation)**

|**Vo**| = **Vi** for d = **0.5**

|**Vo**| > **Vi** for **0.5** < **d** < **1 (Boost Operation)**

1.4 PROCEDURE:

For simulation of circuit using NgSpice Circuit Simulator we have to generate SPICE netlist which is a set of instruction or information about the whereabout and parameters of various components present in particular circuit we have been asked to simulate. For example, if we have been asked to simulate a boost converter then the netlist consists of the coordinate information about Inductor, Capacitor, Switches and their value across different nodes. We will come to that later.

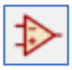
Generation of netlist manually is a very tough and time-consuming work and demands expertise in circuit analysis. It's also very irritating task for a complex network. To ease all of the tough task we use KiCAD, a circuit or schematic capture tool. For our experiment we use KiCAD 5.1. (For latest version click on highlighted KiCAD 5.1) To proceed further follow step by step procedure.

STEP: 1 Download & Install KiCAD & NgSpice from above link (Click on software name mention on "Software/Apps Required section"

STEP: 2 Double click on "**Kicad**", then click on "**File** >> **New** >> **Project**" or directly press "**Ctrl+N**" to create a new project.

STEP: 3 Give a name to your project and select a location for your project and click on "**SAVE**".

STEP: 4 Click on "*your_project_name.**sch***" file to open schematic editor sheet. The file extension (***.sch***) stands for ***schematic.***

STEP: 5 On right hand side, find  this symbol and click on it, then click on middle of sheet which will then open '**Symbol Library**'. Browse the library for required components and click on "**OK**" below to add it to sheet. Repeat the same till all components are added to sheet.

STEP: 6 After adding all components click on  then "**Right click**" on each component and then click on "**Move**" to move it across the sheet to its suitable position. Otherwise after click on specific component "*Press '**M**' in Keyboard*". Repeat the same for all other components.

STEP: 7 After arranging all component, click on  to connect via wire. Just click on one node you want to connect and move the cursor to next connecting node and click again on the second node to terminate the connection.

STEP: 8 Connect a $\mathbf{V_{cc}}$ or Ground node to bottom of the circuit.

STEP: 9 We need to "**Annotate**" the schematic to any conflicts arising due to similar components. To do this we need to go to "**Tools** >> **Annotate Schematic**".

STEP: 10 Save the Schematic by "**Ctrl+S**", then go to "**Tools** >> **Generate Netlist File** >> **SPICE** >> **Generate Netlist** >> ***type_file_name.cir*** >> **Save**".

STEP: 11 Run NgSpice, navigate to project folder in ngspce. Enter command "***souce netlist_filename.cir***". This will run all command placed in netlist_file and also plot the simulation result.

1.5 SIMULATION:

1.5.1 SCHEMATIC CAPTURE USING KiCAD

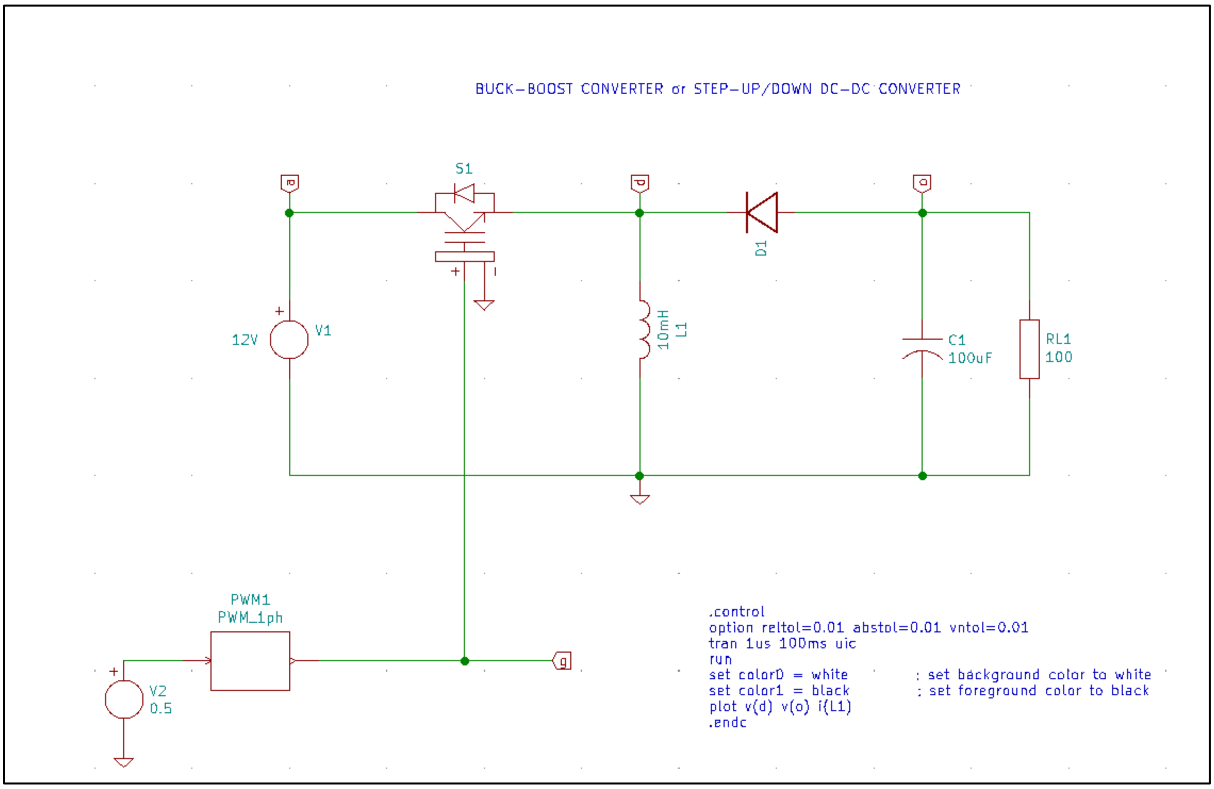


Fig. 4 Schematic of Bock-Boost Converter

1.5.2 NETLIST:

.title KiCad schematic

.include "models/luBlocks.lib"

.include "models/luDevices.lib"

RL1 0 o 100

C1 o 0 100uF

V1 a 0 12V

L1 d 0 10mH

```
XD1 o d diode_pwr
XS1 a d g 0 switch_pwr
V2 Net-_PWM1-Pad1_ 0 0.5
XPWM1 Net-_PWM1-Pad1_ g PWM_1ph fs=10k
.control
option reltol=0.01 abstol=0.01 vntol=0.01
tran 1us 100ms uic
run
set color0 = white        ; set background color to white
set color1 = black        ; set foreground color to black
plot v(d) v(o) i(L1)
.endc
.end
```

1.5.3 Results

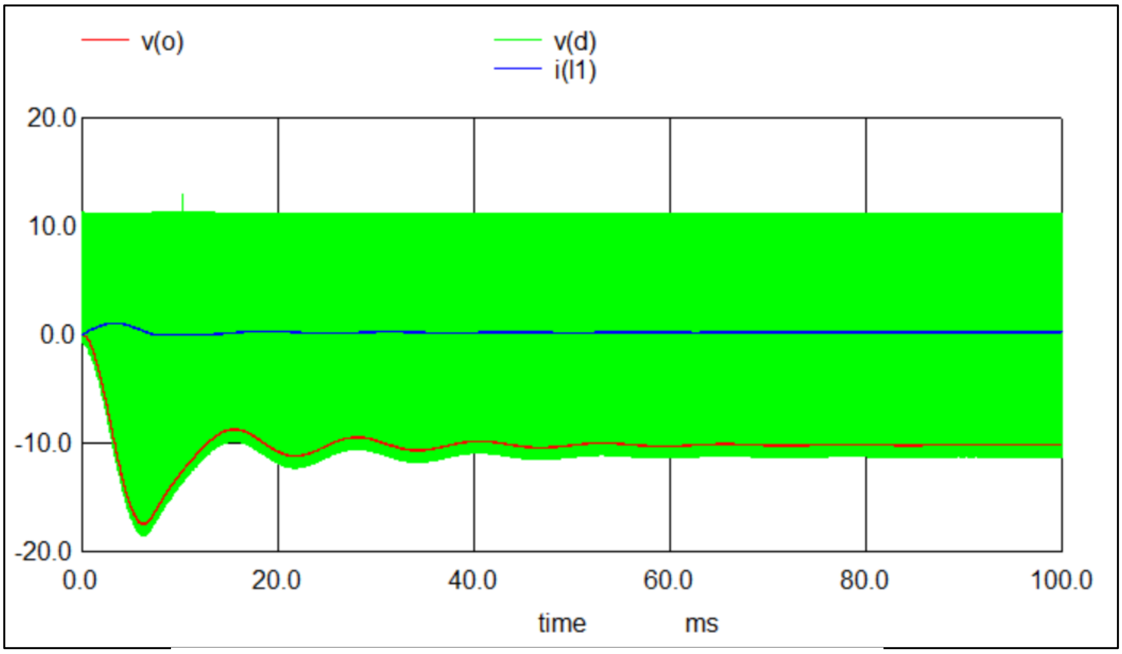


Fig. 5 Simulated waveform of Buck-Boost Converter

1.6 CONCLUSION

From the simulation results it is found that in case of the buck-boost converter, the desired output voltages can be obtained by selecting proper values of inductor, capacitor and switching frequency. All of these individual theories were difficult for anyone to grasp primarily and putting them collectively in the simulator which was extremely puzzling. But it has been done most excellent to formulate an outstanding scheme dissertation with affluent in its contest.

1.7 REFERENCES

The above experiment is done by data acquired from books, links and references from various papers which are listed below.

- Ned Mohan, T. Undeland, and W. Riobbins, "Power Electronics: Converters, Applications and Design," Wiley-India, 2011

- Fundamental of Power Electronics by Prof. L Umanand, IISC Bangalore [NPTEL] https://onlinecourses.nptel.ac.in/noc21_ee01/course

- KiCAD Installation & Getting Started https://youtu.be/bYNg07UhP2Q

- NgSpice Installation & Getting Started https://youtu.be/j0Kg3tYfOVc

- Buck-Boost Converter Simulation https://youtu.be/6pEIM2U6rek

- Download Source Files https://github.com/imrashmi/pecd_burla/

❖