

EXPERIMENT 02

1.0 AIM OF THE EXPERIMENT:

Simulation of Boost Converters using ngSPICE/Pspice

1.1 SOFTWARE/APP REQUIRED:

NgSpice 35, KiCAD 5.1

1.3 THEORY:

1.3.1 BOOST CONVERTER

A boost converter (step-up converter), as its name suggest step up the input DC voltage value and provides at output. This converter contains mostly a diode, a transistor as switches and at least one energy storage element. Capacitors are usually added to output so as to perform the function of removing output voltage ripple and sometimes inductors are also combined with.

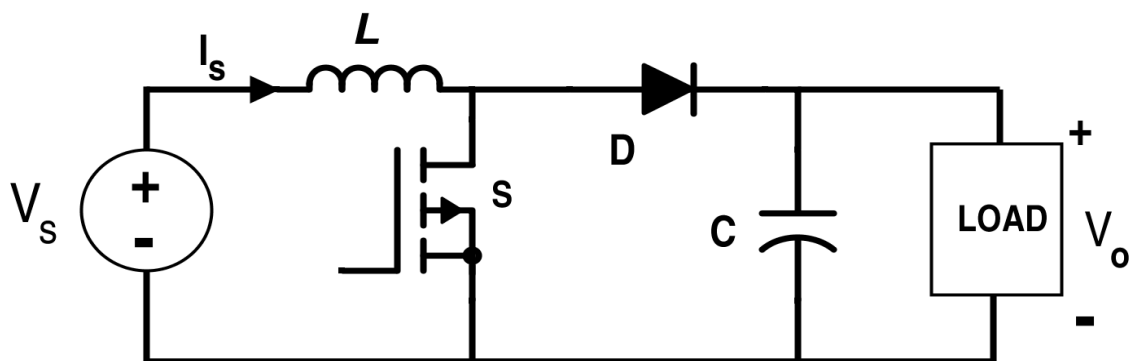


Fig. 3 Circuit Diagram of Ideal Boost Converter

1.3.2 Circuit Operation

Its operation is generally of two separate states,

- During the ON period, switch is made to close its contacts which results in increase of inductor current.
- During the OFF period, switch is made to open and thus the only path for inductor current to flow through the fly-back diode 'D' and the parallel combination of capacitor and load. This enables capacitor to transfer energy gained by it during ON period. In a boost regulator the output voltage is larger than the input voltage hence the name 'boost'.

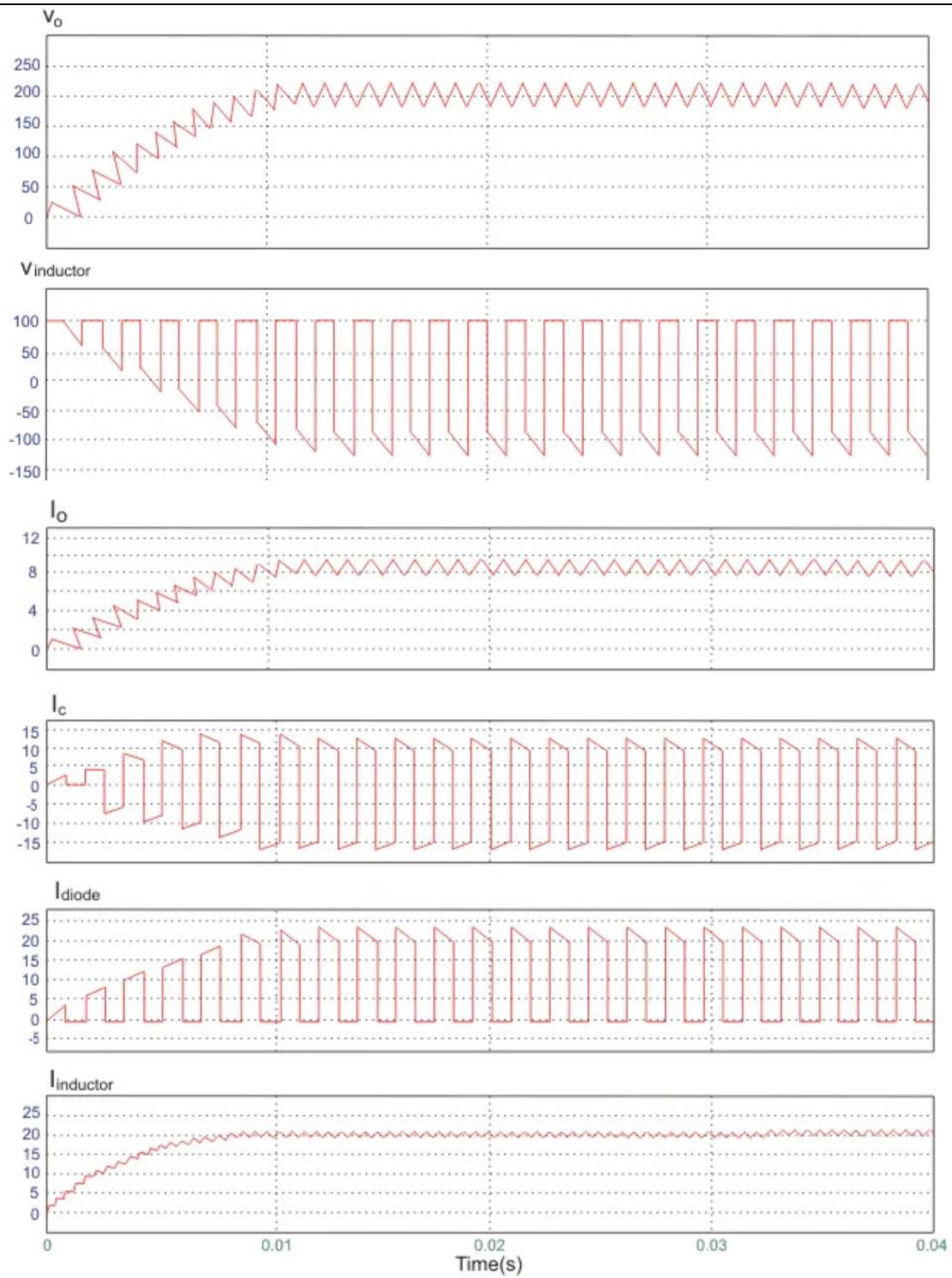


Fig. 4 Waveform of Ideal Boost Converter

1.3.3 Input Output Relationship

$$V_o = \frac{V_s}{1-d}$$

$$I_s = \frac{I_o}{1-d}$$

Where V_o = Output Voltage

V_s = Input Voltage

I_o = Output Current

I_s = Input Current

d = Duty Cycle ($0 \leq d < 1$)

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 whereabouts 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 buck 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 Vcc 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 ngspice. Enter command “source 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

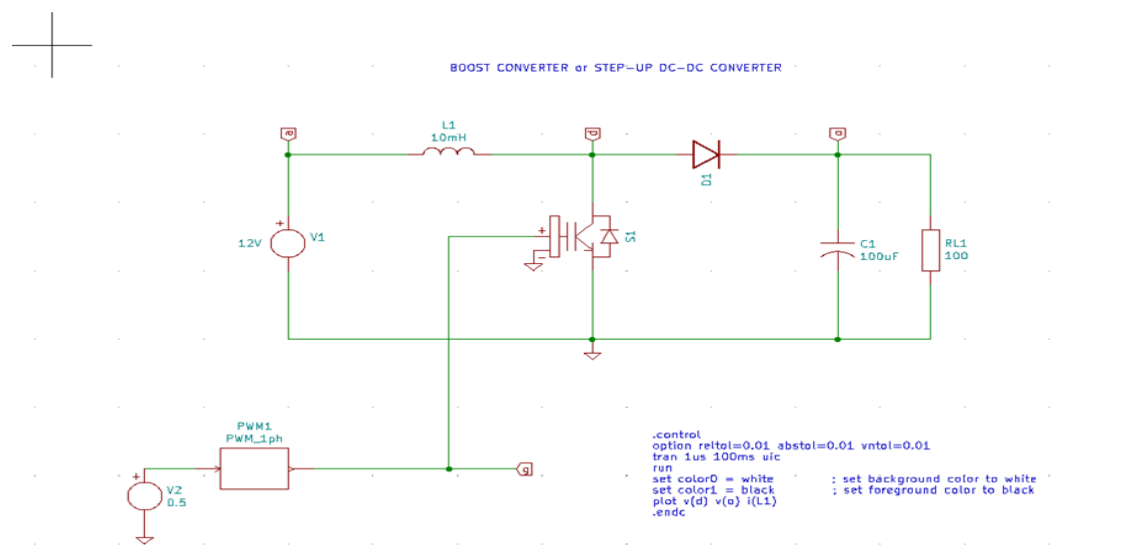


Fig. 3 Schematic of Boost Converter

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 o 10mH

XD1 0 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

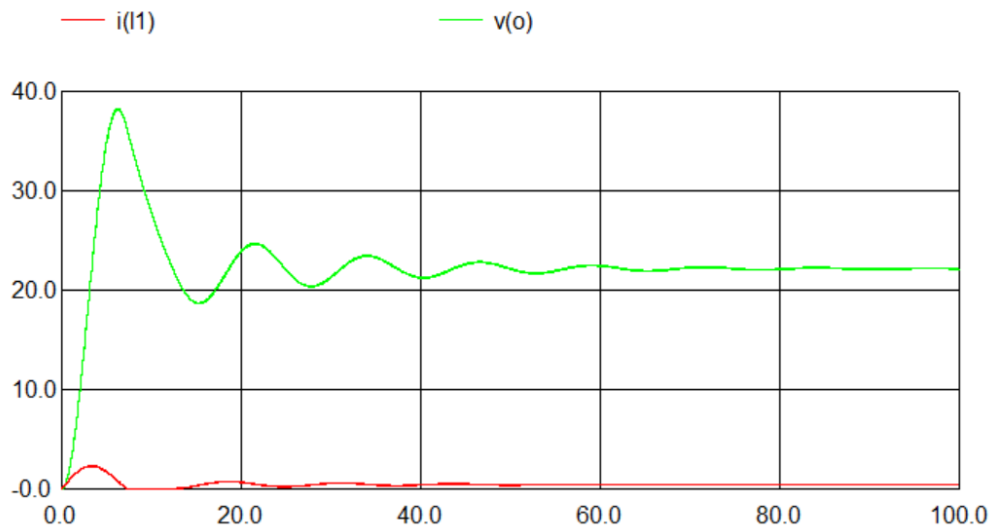


Fig. 4 Simulated waveform of Boost Converter

1.6 CONCLUSION

From the simulation results it is found that in case of the buck 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 Converter Simulation
<https://youtu.be/6pEIM2U6rek>
- Download Source Files
https://github.com/imrashmi/pecd_burla/

