# EXPERIMENT 01

## 1.0 AIM OF THE EXPERIMENT:

Simulation of Buck Converters using ngSPICE/Pspice

## 1.1 SOFTWARE/APP REQUIRED:

**NgSpice 35, KiCAD 5.1**

## 1.3 THEORY:

### 1.3.1 Buck Converter

A buck converter is a step-down dc-dc converter consisting primarily of inductor and two switches (generally a transistor switch and diode) for controlling inductor. It fluctuates between connection of induction to source voltage to mount up energy in inductor and then discharging the inductor's energy to the load. When the switch pictured above is closed (i.e., On-state), the voltage across the inductor is *VL= Vi- Vo.* The current flowing through inductor linearly rises. The diode doesn't allow current to flow through it, since it is reverse-biased by voltage. For Off case (i.e., when switch pictured above is opened), diode is forward biased and voltage is *VL= - Vo* (neglecting drop across diode) across inductor. The inductor current which was rising in ON case now decreases.
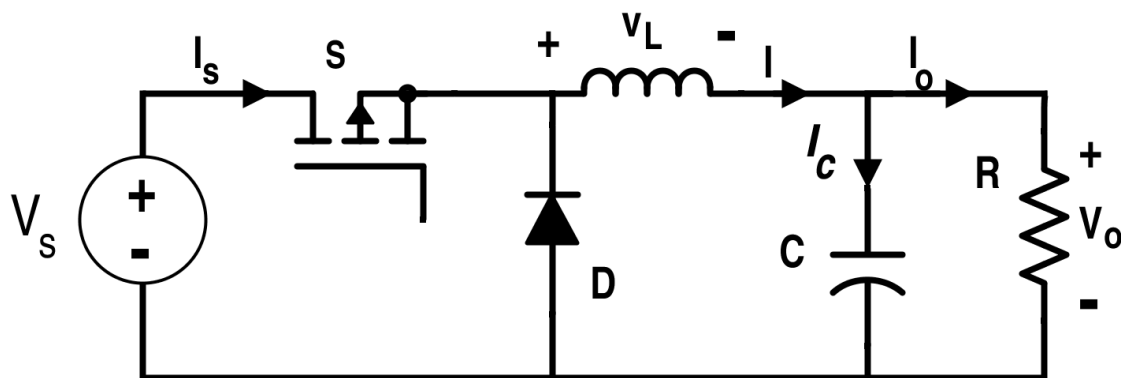


**Fig. 1** Circuit diagram of Ideal Buck Converter

### 1.3.2 Circuit Operation

In above diagram, the average output *Va* is less than the input voltage, *Vs.* The circuit diagram of a buck regulator has shown below and this is like a step-down converter.

The freewheeling diode $D$ conducts due to energy stored in the inductor; and the inductor current continues to flow through inductor ($L$), capacitor ($C$), load and diode ($D$). The inductor current falls until transistor $S$ is switched on again in the next cycle.
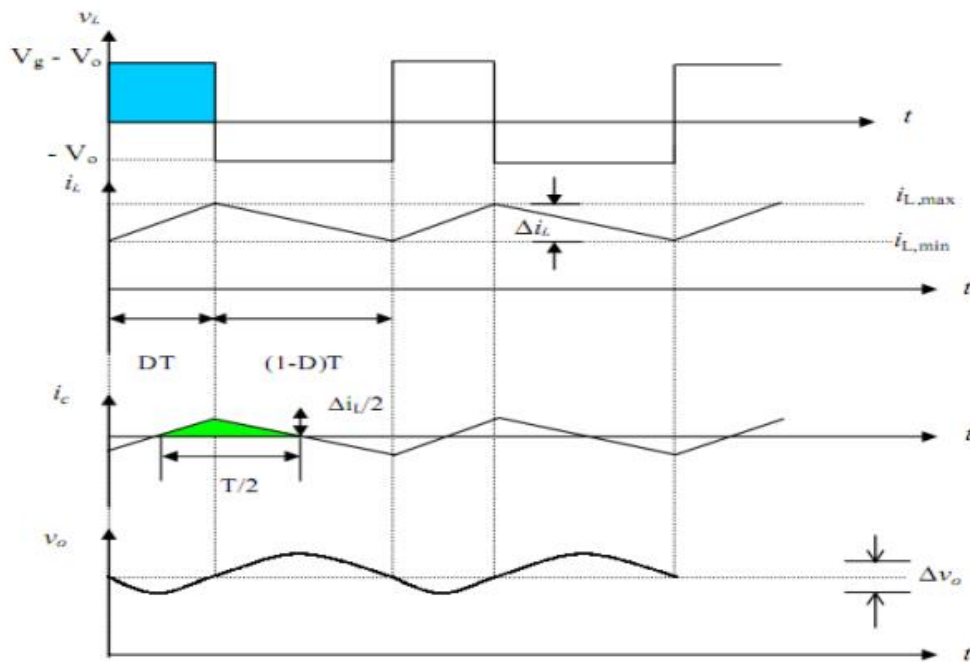


Fig. 2 Waveform of Ideal Buck Converter

The waveforms for voltage and current are shown in for continuous load current assuming that the current rises or falls linearly. For a constant current flow in the inductor $L$, it is assumed that the current rises and falls linearly. In practical circuits, the switch has a finite, nonlinear resistance. Its effect can generally be negligible in the most applications depending on the switching frequency, filter inductance, and capacitance, the inductor current could be discontinuous.

### 1.3.3 Input Output Relationship

Input output relationship determine the value of the desired output based on given input. The relationship can be calculated by following equation;

$$Vo = Vs * d$$

$$Is = Io * d$$

Where   Vo = Output Voltage

Vs = Source Voltage

Io = Output Current

Is = Source Current

d = Duty Cycle

## 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 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 [symbol] 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 [cursor] 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 [wire] 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 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:

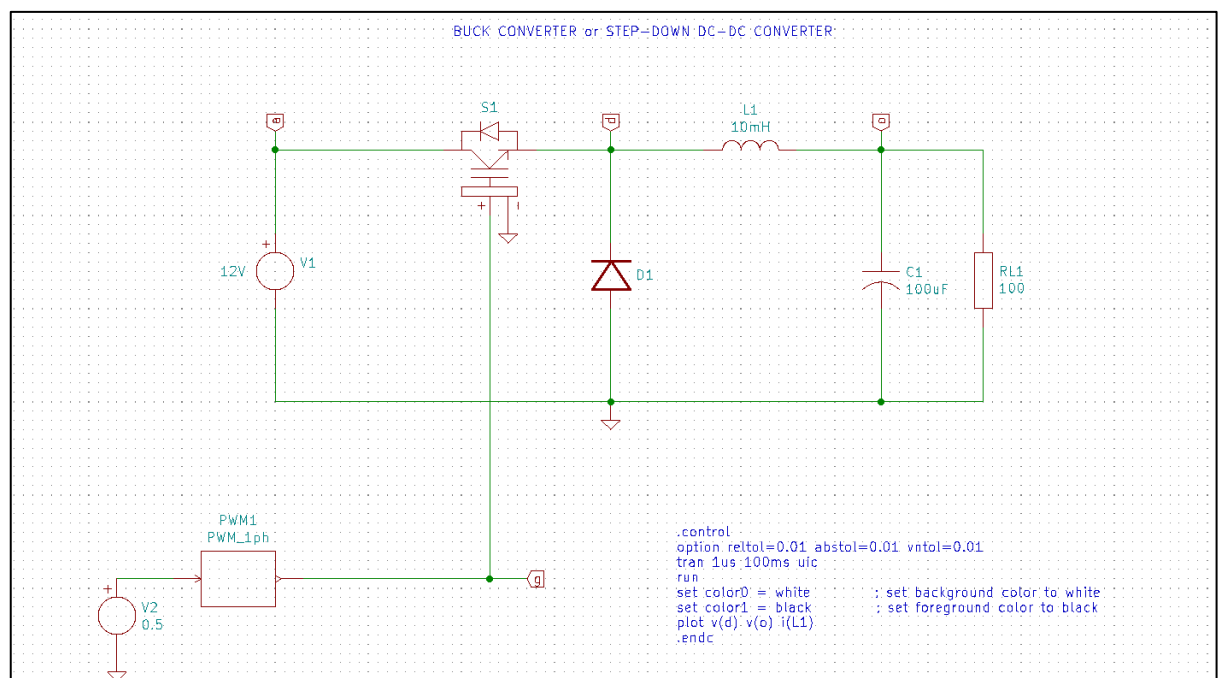### 1.5.1 SCHEMATIC CAPTURE USING KiCAD



**Fig. 3** Schematic of Buck 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 a d 10mH

XD1 d o diode_pwr

XS1 d 0 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
```
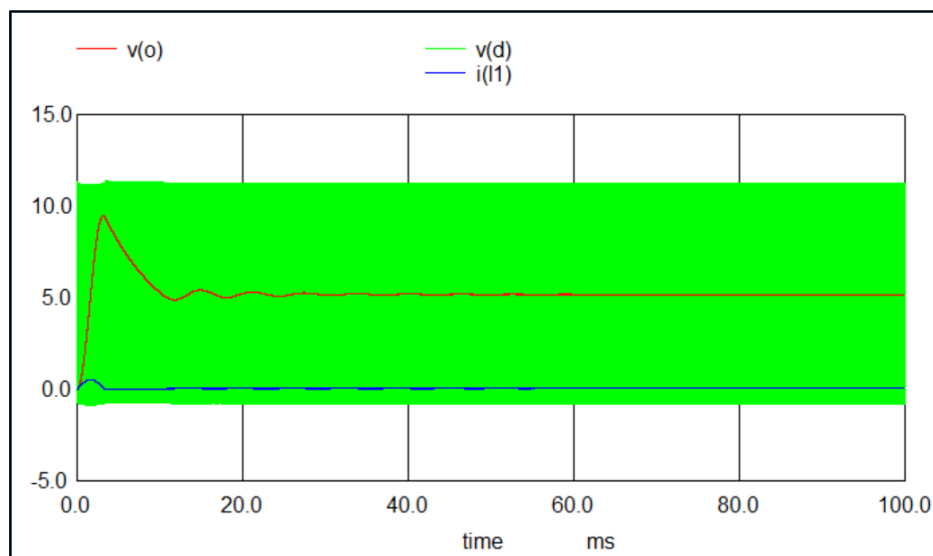
### 1.5.3 Results



**Fig. 4** Simulated waveform of Buck 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/

❖