INTERNSHIP REPORT

A report submitted in partial fulfillment of the requirements for the Award of Degree of

BACHELOR OF ENGINEERING

in

ELECTRICAL AND ELECTRONICS ENGINEERING

By

KESAVARAJ K
Reg No : 24EE037

Under Supervision of

DR. BIJUKUMAR B
Assistant Professor
Department of Electrical and Electronics Engineering

National institute of Technology Puducherry,
Thiruvettakudy, Karaikal, Puducherry - 609609

(Period: 12/12/2025 to 26/12/2025)

Learn Beyond

KPR INSTITUTE OF ENGINEERING AND TECHNOLOGY
(Autonomous, NAAC ‘A’)
Avinashi Road, Arasur
COIMBATORE – 641 407

JANUARY – 2026

KPR Institute of Engineering and Technology

Learn Beyond

BONAFIDE CERTIFICATE

This is to certify that the **Internship report** submitted by **KESAVARAJ K (24EE037)** is the work done by him and submitted during the academic year **2025–2026**, in partial fulfillment of the requirements for the award of the degree of **BACHELOR OF ENGINEERING in ELECTRICAL AND ELECTRONICS ENGINEERING** at **KPR Institute of Engineering and Technology**.

Department IPC Coordinator	**Head of the Department**
Dr. A. Mohamed Ibrahim	**Dr. K. Mohana Sundaram**
Department of EEE	**Department of EEE**
KPR Institute of Engineering and Technology	**KPR Institute of Engineering and Technology**

INTERNSHIP COMPLETION CERTIFICATE

ACKNOWLEDGEMENT

First and foremost, we would like to express our sincere gratitude to our respected **Head of the Department, Dr. K. Mohana Sundaram**, for her continuous support, encouragement, and valuable guidance throughout the internship. Her motivation and kind permission enabled us to gain practical exposure beyond the classroom curriculum.

We are extremely grateful to our **Chief Mentor, Dr. I. Baranilingesan**, for sharing the internship brochure, encouraging us to apply, and providing timely guidance, which helped us gain real-world experience and enhance our technical and professional skills.

We would also like to acknowledge **Dr. B. Bijukumar**, NIT Karaikal, and **Mr. Arun**, a research scholar under Dr. Bijukumar, for their valuable mentorship.

We are highly indebted to our **Chairman, Dr. K.P. Ramasamy**, our **Chief Executive Officer, Dr. A. M. Natarajan**, and our beloved **Principal, Dr. D. Saravanan**, for providing the necessary facilities and support to accomplish this internship successfully.

We sincerely thank the IIPC coordinator, **Dr. A. Mohamed Ibrahim**, faculty members, and our friends for their support, cooperation, and encouragement throughout the internship. The knowledge and experience gained during this period will be invaluable for our academic growth and future professional career.

ABSTRACT

This internship report presents the knowledge and practical experience gained in the field of **Power Electronics**. The primary objective of this internship was to understand the fundamental and advanced concepts of power electronic circuits and their applications through simulation and analysis.

Under the guidance of an expert in power electronics, practical exposure was gained in the use of industry-standard software tools such as **MATLAB** and **LTspice**. These tools were extensively used to model, simulate, and analyze various power electronic converters and control techniques. The internship emphasized understanding circuit behavior, waveform analysis, and performance evaluation under different operating conditions.

The internship also enhanced problem-solving skills by applying theoretical knowledge to real-time simulation-based tasks. Working with MATLAB helped in developing algorithmic thinking and analyzing system performance, while LTspice provided hands-on experience in circuit-level simulation and validation.

This report documents the concepts learned, tools used, and the overall experience gained during the internship, highlighting the significance of power electronics and simulation tools in modern electrical and electronic engineering applications.

TABLE OF CONTENTS

Chapter 1

About the College

The National Institute of Technology, Karaikal, Puducherry, is one of the premier institutions of higher education in India, functioning under the Ministry of Education, Government of India. The institute is well known for its strong academic foundation, research-oriented environment, and commitment to producing skilled engineers and technologists capable of meeting global industry standards.

NIT Karaikal offers undergraduate, postgraduate, and doctoral programs in various engineering and science disciplines. The institution emphasizes both theoretical knowledge and practical exposure, enabling students to gain a deep understanding of engineering concepts and their real-world applications.

The institute is equipped with advanced laboratories, modern infrastructure, and state-of-the-art computational facilities. It actively promotes hands-on learning through internships, industrial training programs, workshops, and research projects. Students are encouraged to engage in innovative problem-solving and interdisciplinary learning.

As part of its academic curriculum and industry interaction initiatives, NIT Karaikal provides internship opportunities to students from various engineering backgrounds. These internships are designed to enhance technical skills, improve analytical abilities, and offer real-time exposure to industrial and research environments. The internship programs focus on bridging the gap between academic learning and practical implementation.

During the internship period, students receive expert guidance from experienced faculty members and researchers. The institute places strong emphasis on emerging technologies, software tools, and simulation platforms such as MATLAB and LTspice, particularly in the field of power electronics and electrical engineering.

Overall, NIT Karaikal plays a significant role in shaping technically competent and professionally responsible engineers by fostering academic excellence, research innovation, and industry-oriented training.

Chapter 2

Plan of the internship program

Week 1: Study of Boost Converter and Filter Circuits

Day 1–2: Study of DC-DC Converter fundamentals in design aspect

The First day of your intenship, The given task is design the DC-DC Converter. It has two types Buck and Boost. On that day, We studied the fundamental concepts and math equations to design our own custom DC-DC Converters. After the completion study of theoretical, next we move on to desgin DC-DC Boost Converter using Math formulae.

Day 3–4: Simulation of DC-DC Boost Converter

During the next two days, we simulated our designed DC–DC Boost Converter to verify whether it was working properly. We used two simulation software tools, namely **MATLAB and LTspice**. MATLAB is an application-based software commonly preferred in academic institutions, whereas LTspice is a circuit-simulation-based software that is more efficient for circuit analysis. Hence, LTspice was preferred and recommended by Dr. B. Bijukumar for this project.

Day 5: Introduction of RC filters

After verifying the operation of our DC–DC Boost Converter and plot the graph 'Efficiency vs Output power' on MatLab, we observed significant ripple at the output side. Such pulsating DC output cannot be directly used to power electronic devices. To reduce the ripple, an output filter is required. Although several types of filters are available, we began with a basic RC filter as an introductory approach suitable for beginners.

Day 6: Study and simulate Low Pass RC filter

The first type of RC filter studied was the low-pass filter, which allows signals with frequencies lower than a selected cutoff frequency to pass while rejects higher-frequency components. To understand its behavior, the mathematical equations and derivations were studied in detail. Subsequently, the filter was simulated using LTspice to obtain frequency response graphs that closely matched the theoretical characteristics.

Day 7: Study and simulate High Pass RC filter

The RC high-pass filter permits high-frequency signals above the cutoff frequency while suppressing low-frequency components, including DC. The theoretical analysis, including cutoff frequency calculation and transfer function derivation, was carried out in detail. The designed filter was then simulated using LTspice, and the obtained frequency response closely aligned with the theoretical characteristics.

Week 2: Study of Boost Converter and Filter Circuits

Day 8: Handplot graph of RC filters

After we got the graph of low and high pass RC filters in LTspice. Sir gave the task to plot the low pass and high pass RC filter graph using 'semi-log graph(5-cycles)' to understand how to plot the graph manually, and so understand where it has the cut-off frequency and learn how to use 'semi-log graphs'.

Day 9: Full Wave Rectifier and Addition of Two Sine Waves with RC Filter

The operation of a full wave rectifier and the design of an RC filter were studied. The rectifier converts AC into pulsating DC with reduced ripple by utilizing both half cycles. Two sine waves of 50 Hz and 100 Hz were combined, and an RC low-pass filter was designed to attenuate the higher-frequency component and obtain a smooth output.

Day 10: Study of PV array and its characteristics

This day marked our introduction to renewable energy systems. We studied the fundamentals of the photovoltaic (PV) array, including its working principle and the arrangement of the p–n junctions within the PV cells. In addition, we analyzed the electrical characteristics of the PV array through the interpretation of its I–V and P–V curves.

Day 11: Design DC-DC Boost Converter with the input source of PV array

Already we designed DC-DC Boost Conveter with the input source of Constant DC source. Now, replaced the constant DC source with PV array and redesign the Boost Converter, which suitable for PV array input.

Day 12: Design a circuit with series & parallel PV array and DC-DC Boost Converter

DC–DC Boost Converter circuits were designed using both series and parallel configurations of two PV arrays. The I–V and P–V characteristics of the series- and parallel-connected PV arrays were observed and analyzed. This study helped in understanding the practical scenarios involved in selecting appropriate PV array configurations for DC–DC Boost Converter design.

Day 13: Introduction and Importance of MPPT in PV

In a series-connected PV array with a DC–DC boost converter, unequal irradiance levels cause mismatch losses, reducing the output power. To analyze this effect, the I–V and P–V characteristics of separate and series PV configurations are plotted and studied. An MPPT algorithm is then implemented to reduce these losses and ensure maximum power extraction by operating the system at the maximum power point.

Day 14: Design and Implementation of MPPT in PV array

After understanding the concept of Maximum Power Point Tracking (MPPT), the Perturb and Observe (P&O) algorithm, which is one of the commonly used MPPT techniques, is studied in detail to extract the maximum power output from the PV array. The algorithm is then implemented and simulated in MATLAB/Simulink to verify its effectiveness in tracking the maximum power point under varying operating conditions.

Day 15: Hardware Demonstration

On the final day of our internship, we had the opportunity to observe the hardware setup of the MPPT controller along with the PV array simulation instruments, helping us understand the practical implementation and real-time operation of MPPT systems.

WEEKLY OVERVIEW OF INTERNSHIP ACTIVITIES

	DATE	DAY	NAME OF THE TOPIC COMPLETED
1^{st} WEEK	12/12/25	Friday	Study of DC-DC Boost Converter
	13/12/25	Saturday	Design of DC-DC Boost Conveter
	14/12/25	Sunday	Simulation of DC-DC Boost Converter in MATLAB
	15/01/25	Monday	Simulation of DC-DC Boost Converter in LTspice
	16/12/25	Tuesday	Introduction of RC Filters
	17/12/25	Wednesday	Study and simulate Low Pass Filter
	18/12/25	Thursday	Study and simulate High Pass Filter
2^{nd} WEEK	19/12/25	Friday	Handplot graph of RC Filters
	20/12/25	Saturday	Full Wave Rectifier and Addition of Two Sine Waves with RC Filter
	21/12/25	Sunday	Study and analysis of PV array
	22/12/25	Monday	Design of PV array with Boost Converter
	23/12/25	Tuesday	Design of Boost Converter with series & parallel PV array
	24/12/25	Wednesday	Study of MPPT
	25/12/25	Thursday	Design of MPPT P&O algorithm
	26/12/25	Friday	Hardware Demonstration

Chapter 3

Training Task

Task 1: Study and design of Boost Conveter

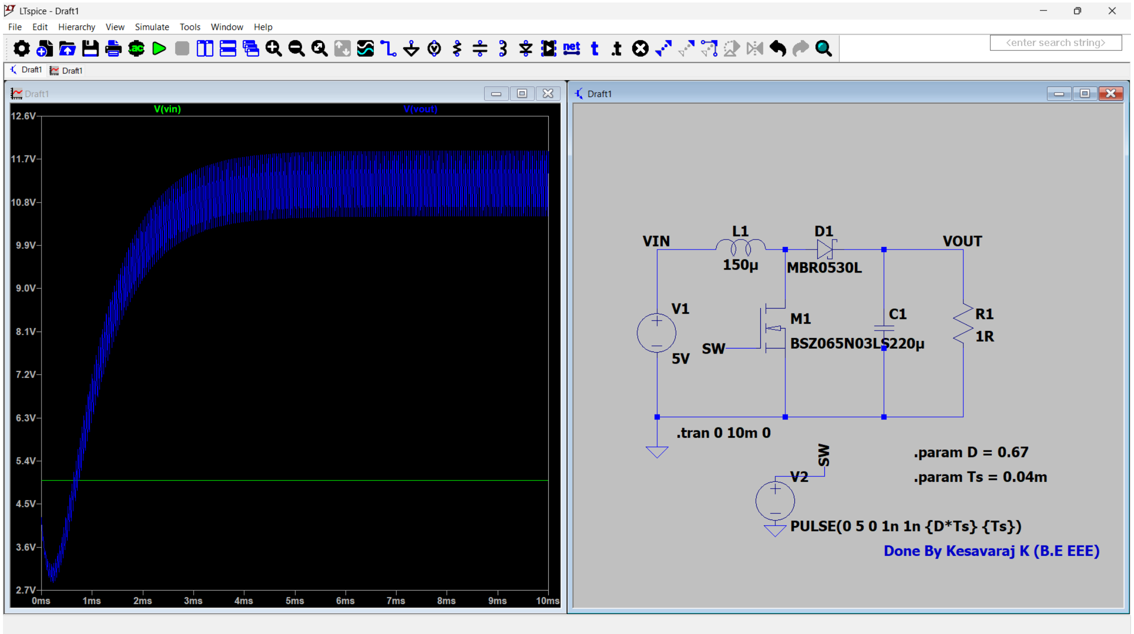


Figure 1: Boost Converter Circuit Designed in LTspice

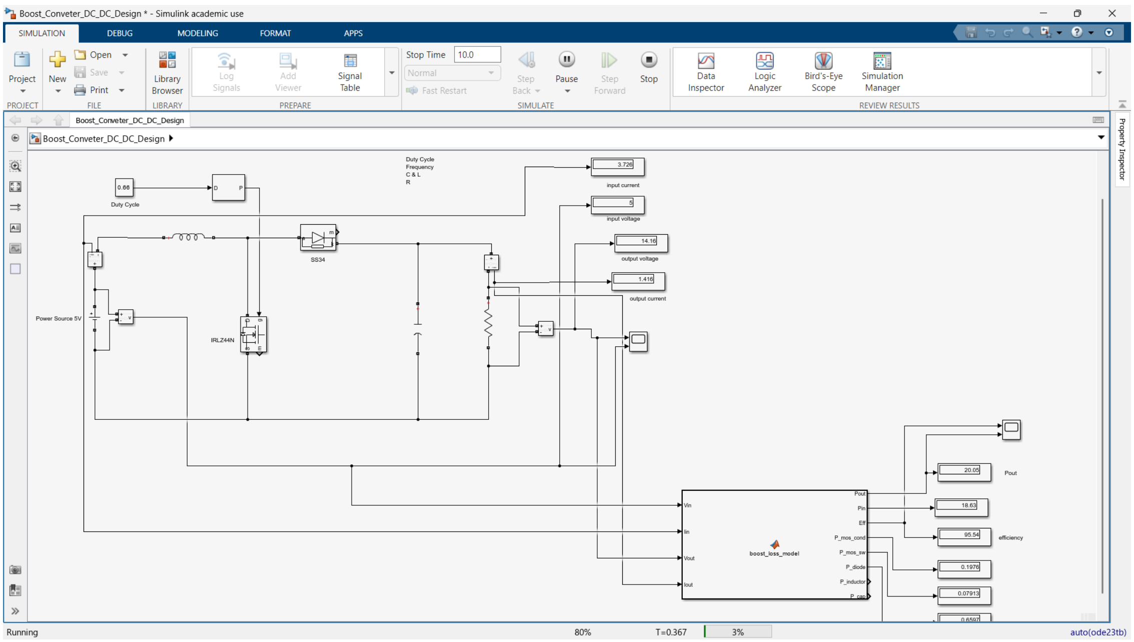


Figure 2: Boost Converter Circuit Designed in MATLAB

Calculations:

1 Specifications

- Input voltage: $V_{in} = 5\,\mathrm{V}$
- Output voltage: $V_{out} = 15\,\mathrm{V}$
- Switching frequency: $f_s = 25\,\mathrm{kHz}$
- Inductor: $L = 150\,\mathrm{mH}$
- Output capacitor: $C = 220\,\mu\mathrm{F}$
- Load resistance range: $R \in \{10, 20, 30, 40, 50, 60\}\,\Omega$

2 Design Calculations

2.1 Duty Cycle

$$D = 1 - \frac{V_{in}}{V_{out}} = 1 - \frac{5}{15} = 0.667$$

2.2 Output Current and Power for Each Load

$$I_{out} = \frac{V_{out}}{R}, \quad P_{out} = V_{out} \cdot I_{out}$$

$R\,(\Omega)$	$I_{out}\,(\mathrm{A})$	$P_{out}\,(\mathrm{W})$
10	1.500	22.500
20	0.750	11.250
30	0.500	7.500
40	0.375	5.625
50	0.300	4.500
60	0.250	3.750

Table 1: Output current and power for given loads.

3 MOSFET and Diode Loss Analysis

3.1 MOSFET Parameters

- $R_{DS(on)} = 22\,\mathrm{m}\Omega$
- Rise time $t_r = 60\,\mathrm{ns}$
- Fall time $t_f = 60\,\mathrm{ns}$

3.2 MOSFET RMS Current

$$I_{MOS,rms} = I_{in}\sqrt{D}$$

For $I_{in} = 4.5\,\mathrm{A}$:

$$I_{MOS,rms} = 4.5 \times \sqrt{0.667} = 3.64\,\mathrm{A}$$

3.3 MOSFET Conduction Loss

$$P_{cond,MOS} = I^2_{MOS,rms} \cdot R_{DS(on)}$$

$$P_{cond,MOS} = (3.64)^2 \times 22 \times 10^{-3} = 0.29\,\mathrm{W}$$

3.4 MOSFET Switching Loss

$$P_{sw,MOS} = \frac{1}{2} V_{in} I_{in} (t_r + t_f) f_s$$

$$P_{sw,MOS} = \frac{1}{2} \times 5 \times 4.5 \times (60 + 60) \times 10^{-9} \times 28 \times 10^3 = 0.033\,\mathrm{W}$$

3.5 Diode Parameters (SS34)

- Forward voltage $V_F = 0.5\,\mathrm{V}$
- Reverse recovery charge $Q_{rr} = 5\,\mathrm{nC}$

3.6 Diode Conduction Loss

$$P_{cond,D} = I_{out} \cdot V_F \cdot (1 - D)$$

For $I_{out} = 1.5\,\mathrm{A}$:

$$P_{cond,D} = 1.5 \times 0.5 \times (1 - 0.667) = 0.249\,\mathrm{W}$$

3.7 Diode Reverse Recovery Loss

$$P_{rr} = Q_{rr} \cdot V_{out} \cdot f_s$$

$$P_{rr} = 5 \times 10^{-9} \times 15 \times 28 \times 10^3 = 6 \times 10^{-18}\,\mathrm{W} \quad \text{(negligible)}$$

3.8 Total Loss

$$P_{loss} = P_{cond,MOS} + P_{sw,MOS} + P_{cond,D} + P_{rr}$$

$$P_{loss} = 0.29 + 0.033 + 0.249 + 6 \times 10^{-18} \approx 0.572\,\mathrm{W}$$

4 Efficiency

For $R = 10\,\Omega$, $P_{out} = 22.5\,\mathrm{W}$:

$$\eta = \frac{P_{out}}{P_{out} + P_{loss}} \times 100\% = \frac{22.5}{22.5 + 0.572} \times 100\% \approx 97.5\%$$

5 Efficiency for Various Loads

$R\,(\Omega)$	P_{out} (W)	η (%)
10	22.50	97.5
20	11.25	95.2
30	7.50	92.9
40	5.63	90.8
50	4.50	88.9
60	3.75	87.1

Table 2: Efficiency across different load resistances.

Ripple Factor Calculation

The output capacitor ripple voltage is given by:

$$\Delta v_c = \frac{I_o \cdot D}{f \cdot C}$$

Substituting the given values:

$$\Delta v_c = \frac{1.14 \times 0.58}{50 \times 97 \times 10^{-6}}$$

$$\Delta v_c = 0.13 \text{ V}$$

Where:

$$I_o = 1.14\,\text{A} \quad \text{(output current)}$$
$$D = 0.58 \quad \text{(duty cycle)}$$
$$f = 50\,\text{Hz} \quad \text{(switching frequency)}$$
$$C = 97\,\mu\text{F} \quad \text{(output capacitance)}$$

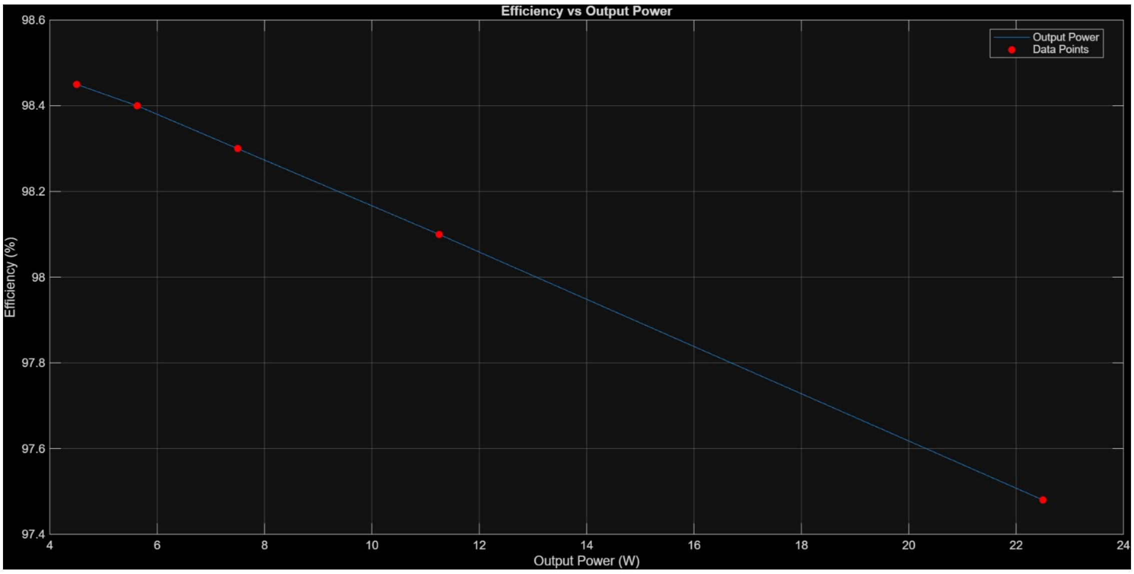


Figure 3: Efficiency vs Ouput Power graph from our calculation

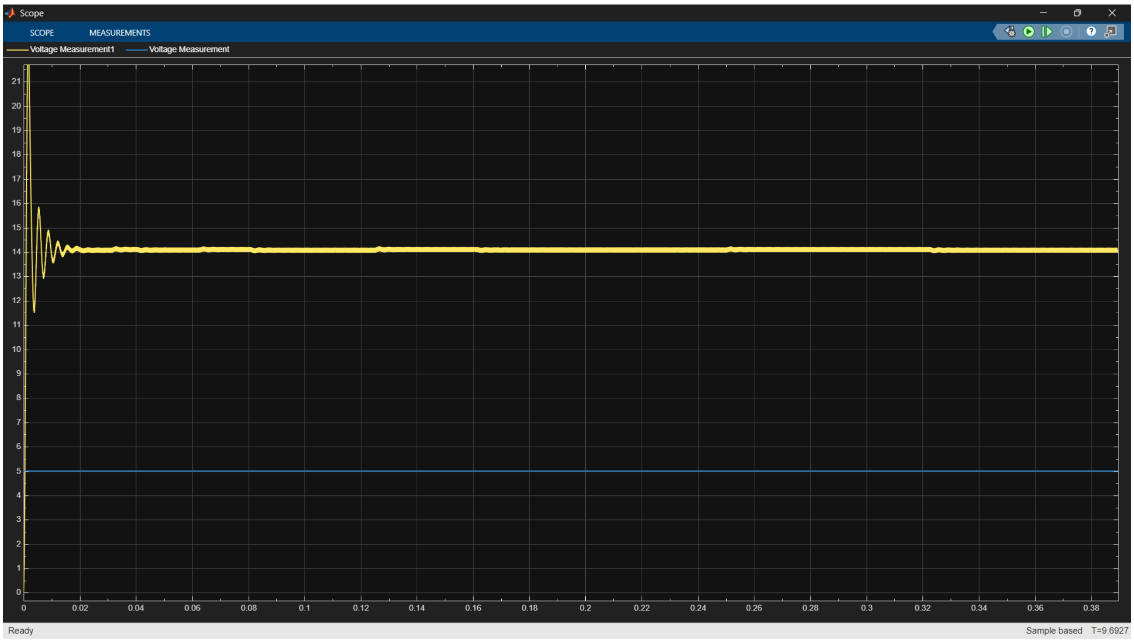


Figure 4: Boost Converter Circuit Output graph in MATLAB

Task 2: Study and design of RC Filters

After observing the output waveform of the DC–DC Boost Converter (5 V / 15 V), the ripple factor is found to be very high without an RC filter. From this, we understand the importance of an RC filter, which is one of the simplest filter circuits. There are two types of RC filters, namely:

1. Low Pass RC Filter
2. High Pas RC Filter

1. Low Pass RC Filter

This circuit allows frequencies which should be lower than cut-off frequency.

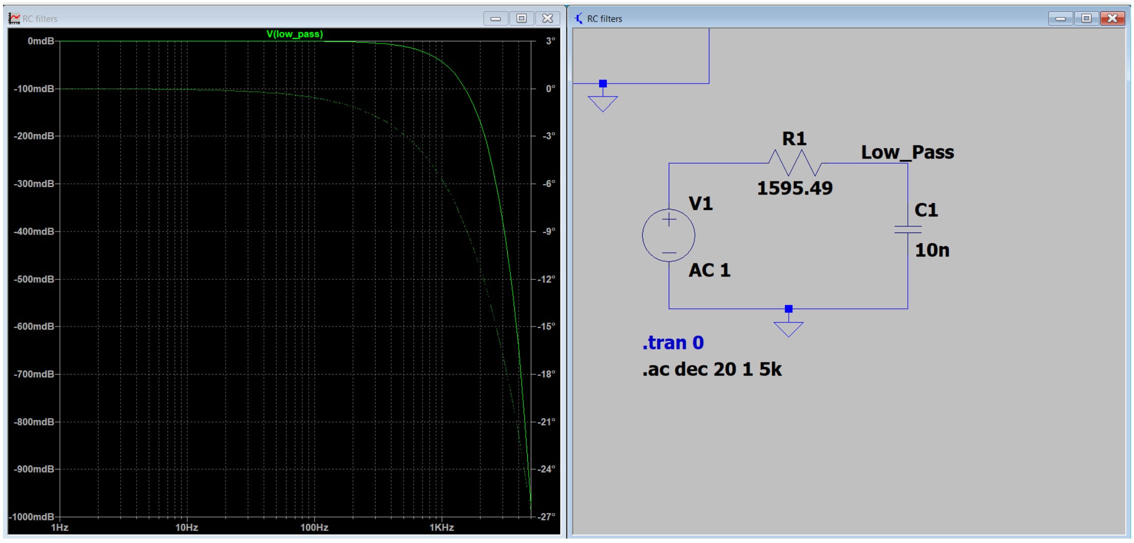


Figure 5: Low Pass RC Filter in LTspice

On the left side of Figure 5, one graph is clearly shown, which represents the characteristics of the RC filter.

Usually, the cut-off frequency occurs at -3 dB (or 70.7% of the maximum output). Therefore, the dark green line shows the frequency response where the cut-off frequency is at 1 kHz, as per our design.

2. High Pass RC Filter

This circuit allows frequencies, which should be greater than cut-off frequency.

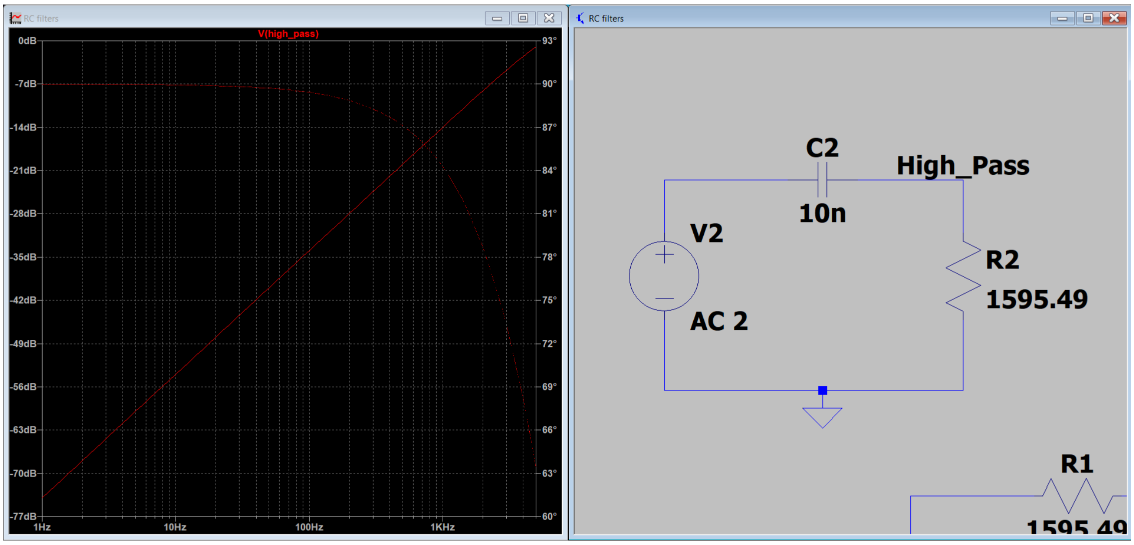


Figure 6: High Pass RC Filter in LTspice

On the left side of Figure 6, one graph is clearly shown, which represents the characteristics of the RC filter.
same as, the cut-off frequency occurs at $-3\,\mathrm{dB}$ (or 70.7% of the maximum output). Therefore, the dark red line shows the frequency response where the cut-off frequency is at $1\,\mathrm{kHz}$, as per our design.

RC Low-Pass and High-Pass Filters

6 RC Low-Pass Filter

6.1 Transfer Function Derivation (s-Domain)

- Impedance of resistor: $Z_R = R$
- Impedance of capacitor: $Z_C = \frac{1}{sC}$, where $s = \sigma + j\omega$

Applying voltage division:

$$V_{\text{out}} = V_{\text{in}} \cdot \frac{Z_C}{Z_R + Z_C} = V_{\text{in}} \cdot \frac{\frac{1}{sC}}{R + \frac{1}{sC}}$$

Multiplying numerator and denominator by sC:

$$V_{\text{out}} = V_{\text{in}} \cdot \frac{1}{1 + sRC}$$

The transfer function of a first-order RC low-pass filter is:

$$H(s) = \frac{1}{1 + sRC}$$

- DC gain: $H(0) = 1$
- Pole at: $s = -\frac{1}{RC}$, which determines the cutoff frequency

6.2 Cutoff Frequency

Substitute $s = j\omega$, where $\omega = 2\pi f$:

$$H(j\omega) = \frac{1}{1 + j\omega RC}$$

Magnitude response:

$$|H(j\omega)| = \frac{1}{\sqrt{1 + (\omega RC)^2}}$$

At cutoff frequency ω_c, $|H(j\omega_c)| = \frac{1}{\sqrt{2}}$:

$$\frac{1}{\sqrt{1 + (\omega_c RC)^2}} = \frac{1}{\sqrt{2}}$$

$$1 + (\omega_c RC)^2 = 2$$
$$(\omega_c RC)^2 = 1$$
$$\omega_c RC = 1$$
$$\omega_c = \frac{1}{RC}$$

Converting to frequency:

$$f_c = \frac{\omega_c}{2\pi} = \frac{1}{2\pi RC}$$

6.3 Step Response (Time Domain)

For a unit step input $V_{\text{in}}(s) = \frac{1}{s}$:

$$V_{\text{out}}(s) = H(s) \cdot V_{\text{in}}(s) = \frac{1}{s(1 + sRC)}$$

Partial fraction decomposition:

$$\frac{1}{s(1 + sRC)} = \frac{A}{s} + \frac{B}{1 + sRC}$$
$$1 = A(1 + sRC) + Bs$$

Solving:

$$\text{Let } s = 0 : \quad 1 = A(1 + 0) \Rightarrow A = 1$$
$$\text{For } s \text{ terms:} \quad 0 = ARC + B \Rightarrow 0 = RC + B \Rightarrow B = -RC$$

Thus:

$$V_{\text{out}}(s) = \frac{1}{s} - \frac{RC}{1 + sRC} = \frac{1}{s} - \frac{1}{s + \frac{1}{RC}}$$

Applying inverse Laplace transform:

$$V_{\text{out}}(t) = \mathcal{L}^{-1}\left\{\frac{1}{s}\right\} - \mathcal{L}^{-1}\left\{\frac{1}{s + \frac{1}{RC}}\right\}$$
$$V_{\text{out}}(t) = 1 - e^{-t/RC}$$

For general input V_m:

$$V_{\text{out}}(t) = V_m\left(1 - e^{-t/RC}\right)$$

6.3.1 Special Cases

1. At $t = 0$: $V_{\text{out}}(0) = V_m(1 - e^0) = 0$
2. At $t = \infty$: $V_{\text{out}}(\infty) = V_m(1 - e^{-\infty}) = V_m$
3. At $t = RC$ (time constant):

$$V_{\text{out}}(RC) = V_m\left(1 - e^{-1}\right) = V_m\left(1 - \frac{1}{e}\right) = V_m(1 - 0.3679) = 0.6321V_m$$

6.4 Frequency Response (Bode Plot)

6.4.1 Magnitude

$$|H(j\omega)|_{\text{dB}} = 20\log_{10}|H(j\omega)| = 20\log_{10}\left(\frac{1}{\sqrt{1+(\omega RC)^2}}\right)$$

- Low frequency ($\omega \ll \omega_c$): Gain ≈ 0 dB
- High frequency ($\omega \gg \omega_c$): Gain decreases at -20 dB/decade (or -6 dB/octave)
- At cutoff ($\omega = \omega_c$): Gain = -3 dB

6.4.2 Phase Shift

$$\angle H(j\omega) = -\tan^{-1}(\omega RC)$$

- At $f = 0$: Phase shift = $0°$
- At $f = f_c$: Phase shift = $-45°$
- At $f \to \infty$: Phase shift = $-90°$

6.5 Our Design

Design an RC LPF with cutoff frequency $f_c = 1\,\text{kHz}$.
Given $f_c = \frac{1}{2\pi RC}$, let $C = 10\,\text{nF}$:

$$R = \frac{1}{2\pi f_c C} = \frac{1}{2\pi \times 1000 \times 10 \times 10^{-9}} = \frac{1}{6.2832 \times 10^{-5}} = 1.592 \times 10^4\,\Omega$$

$$R \approx 15.9\,\text{k}\Omega$$

7 RC High-Pass Filter

7.1 Transfer Function Derivation

$$H(s) = \frac{V_{\text{out}}(s)}{V_{\text{in}}(s)}$$

Applying voltage division:

$$V_{\text{out}}(s) = V_{\text{in}}(s) \cdot \frac{Z_R}{Z_C + Z_R} = V_{\text{in}}(s) \cdot \frac{R}{R + \frac{1}{sC}}$$

Multiplying numerator and denominator by sC:

$$H(s) = \frac{sRC}{1 + sRC}$$

7.2 Frequency Response

Substituting $s = j\omega$:

$$H(j\omega) = \frac{j\omega RC}{1 + j\omega RC}$$

Magnitude:

$$|H(j\omega)| = \frac{\omega RC}{\sqrt{1 + (\omega RC)^2}}$$

Cutoff frequency (where $|H(j\omega_c)| = \frac{1}{\sqrt{2}}$):

$$\frac{\omega_c RC}{\sqrt{1 + (\omega_c RC)^2}} = \frac{1}{\sqrt{2}}$$

Solving gives $\omega_c RC = 1$, so:

$$f_c = \frac{1}{2\pi RC}$$

- Low frequency ($\omega \ll \omega_c$): Gain approaches 0 dB (after high-pass effect)
- High frequency ($\omega \gg \omega_c$): Gain approaches 0 dB
- At cutoff: Gain = -3 dB

7.3 Our design

Design an RC HPF with cutoff frequency $f_c = 1\,\text{kHz}$, using $C = 10\,\text{nF}$:

$$R = \frac{1}{2\pi f_c C} = \frac{1}{2\pi \times 10^3 \times 10 \times 10^{-9}} = 1.592 \times 10^4\,\Omega$$

$$R \approx 15.9\,\text{k}\Omega$$

7.4 Summary

Parameter	Low-Pass Filter	High-Pass Filter
Transfer function	$H(s) = \frac{1}{1 + sRC}$	$H(s) = \frac{sRC}{1 + sRC}$
Cutoff frequency	$f_c = \frac{1}{2\pi RC}$	$f_c = \frac{1}{2\pi RC}$
DC gain	1	0
High-frequency gain	0	1
Phase at f_c	$-45°$	$45°$

In design aspect, we are focusing to choose the cut-off frequency. In above table, formula for cut-off frequency is same for both. Behavior of RC filter is based on the arrangement of resistor and capacitor.

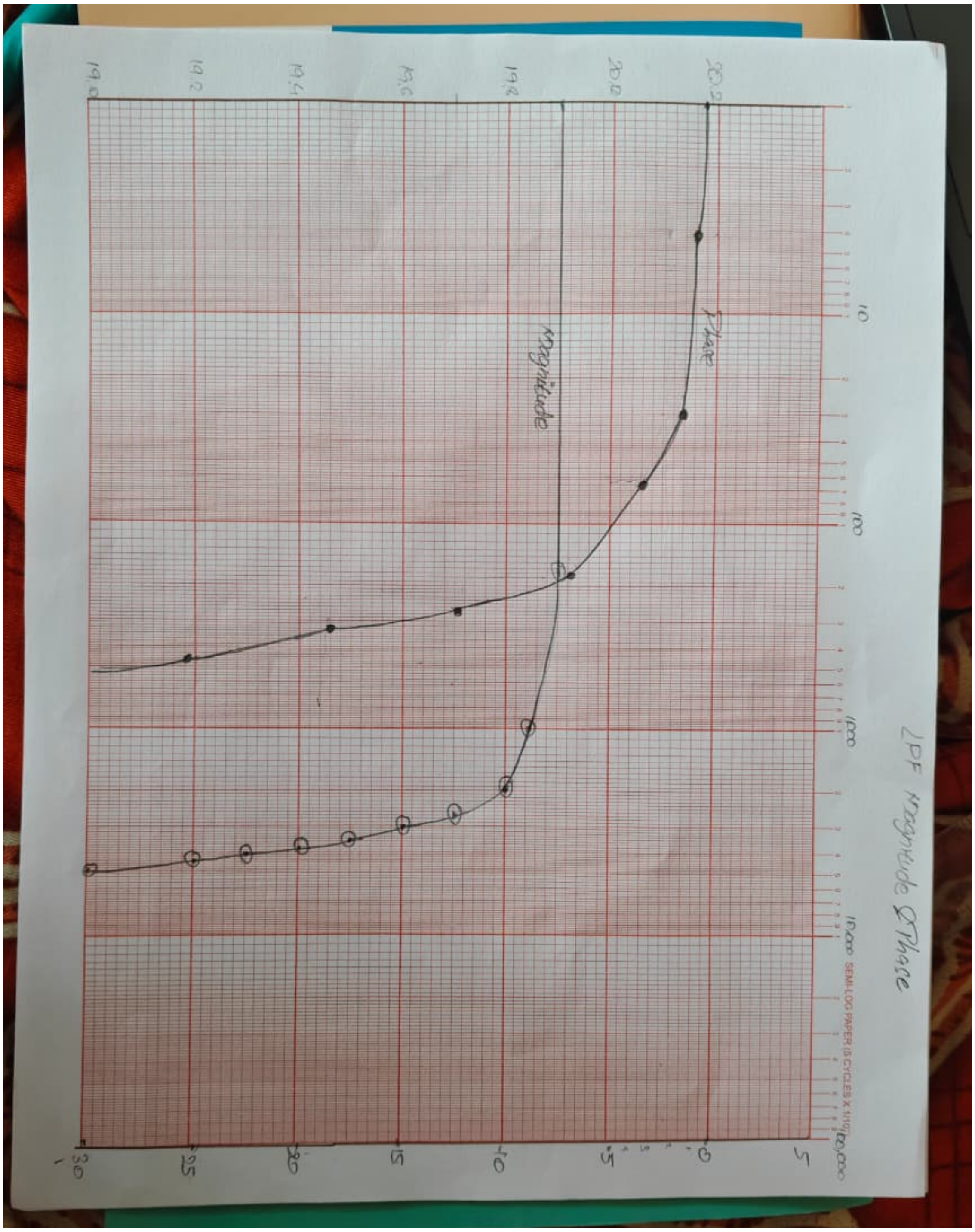


Figure 7: Hand Plotted Low Pass RC Filter characteristics graph in semi-log graph(5-cycles).

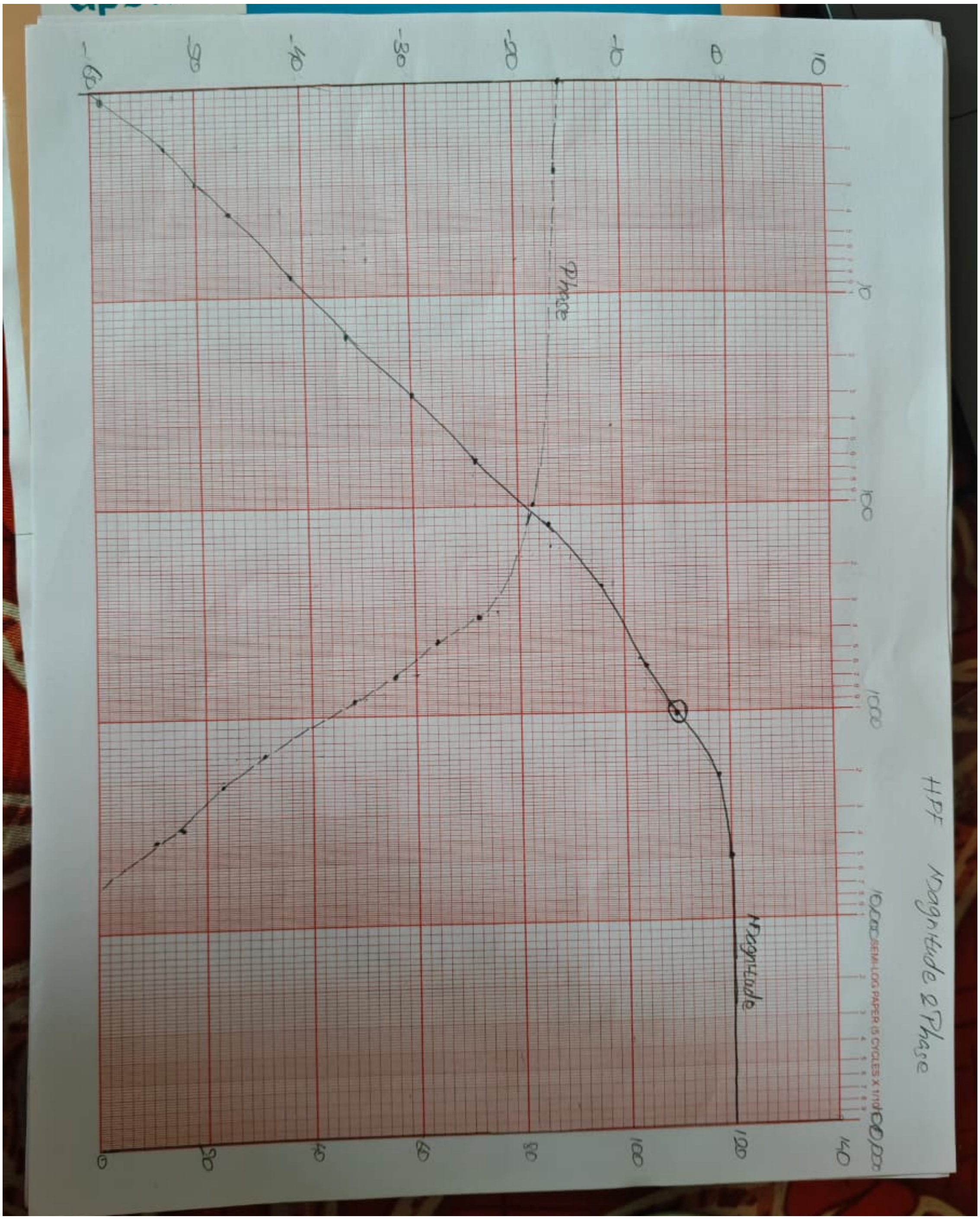


Figure 8: Hand Plotted High Pass RC Filter characteristics graph in semi-log graph(5-cycles).

RC filter practical application tasks:

1. Using RC filter with Full Wave Rectifier:

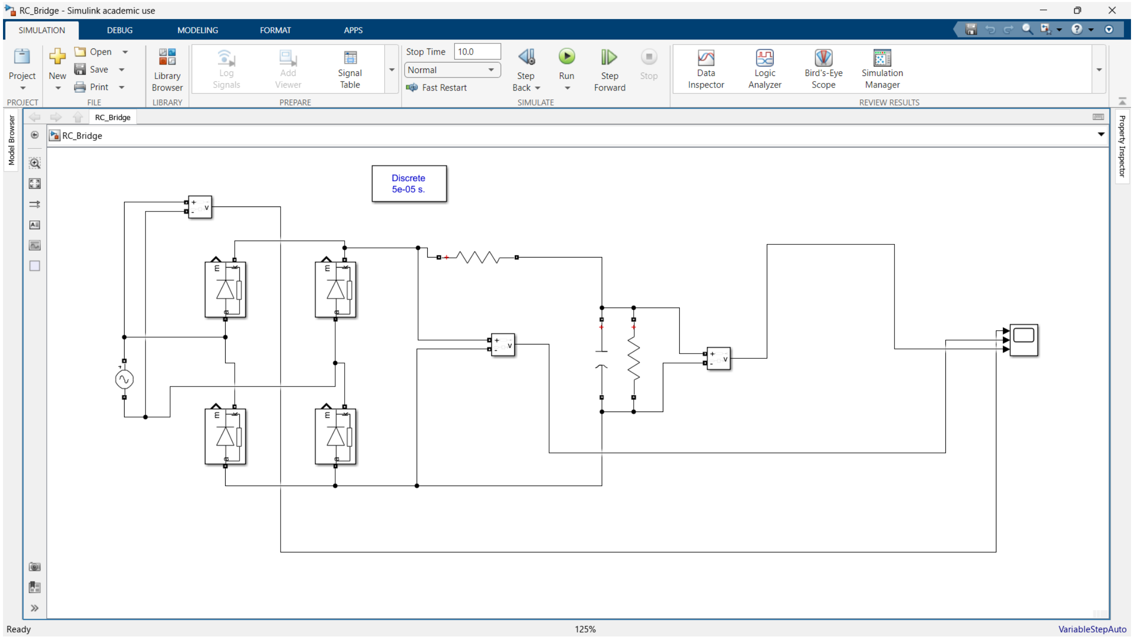


Figure 9: Full Wave Rectifier circuit with RC filter results in MATLAB

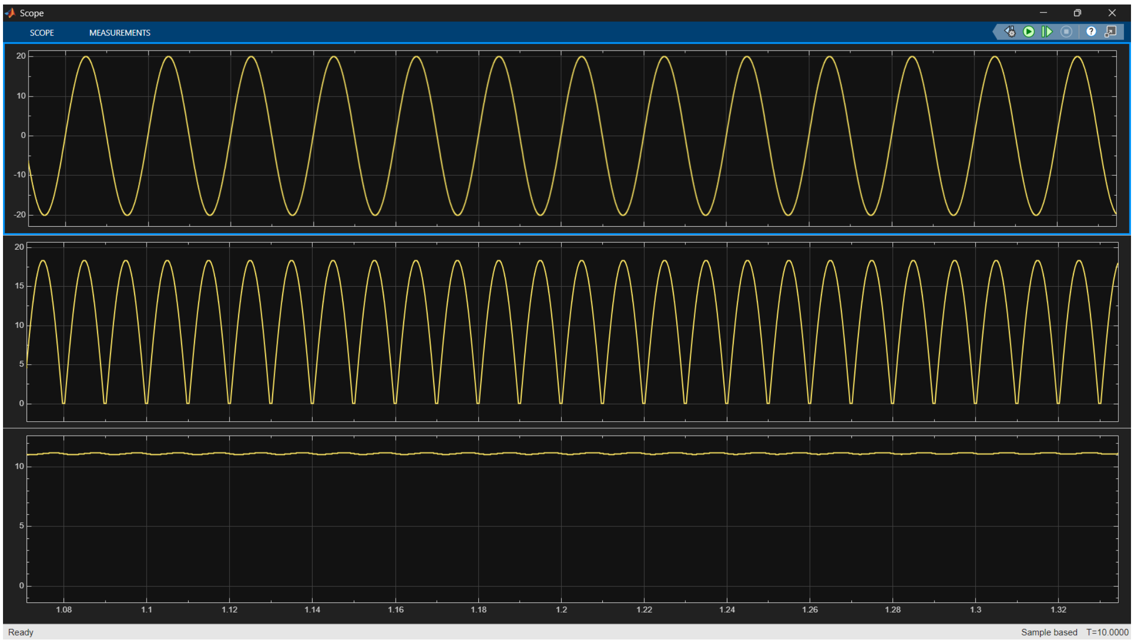


Figure 10: Full Wave Rectifier Output with RC filter results in MATLAB

2. Add two sine waves (50 Hz and 100 Hz):

In the design of a circuit to add two sine waves, the limitations of the RC filter become evident.

Although the RC filter is simple and efficient, it performs poorly when the frequencies are very close—such as 50 Hz and 100 Hz—making the filtering effect negligible.

Thus, RC filters are ineffective for separating closely spaced frequencies.

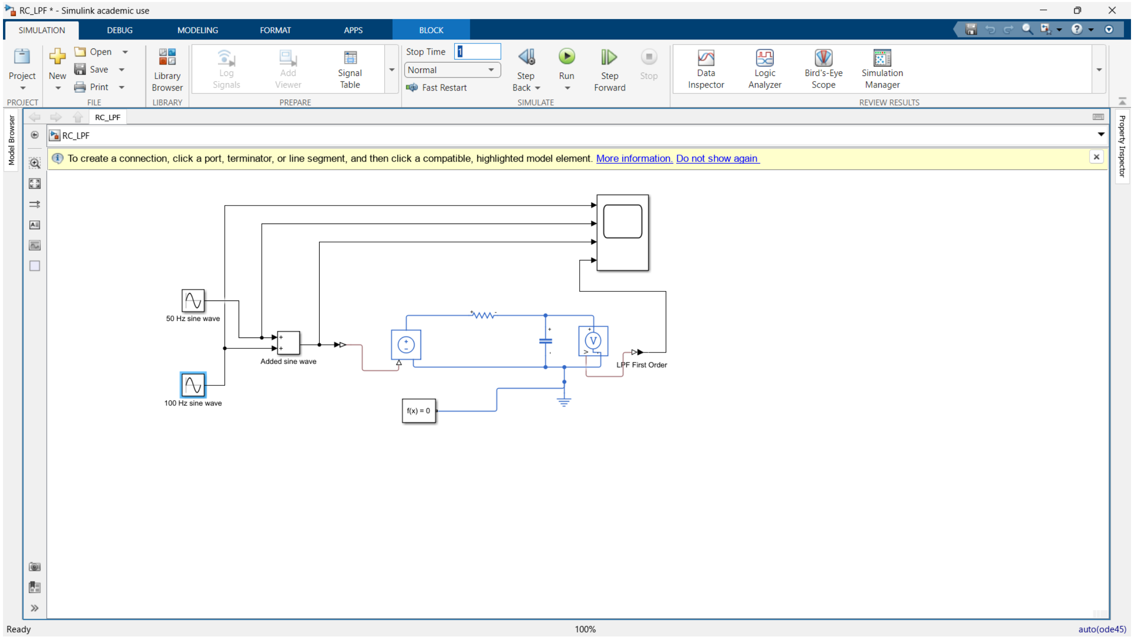


Figure 11: Design Addition of two sine wave circuit in MATLAB

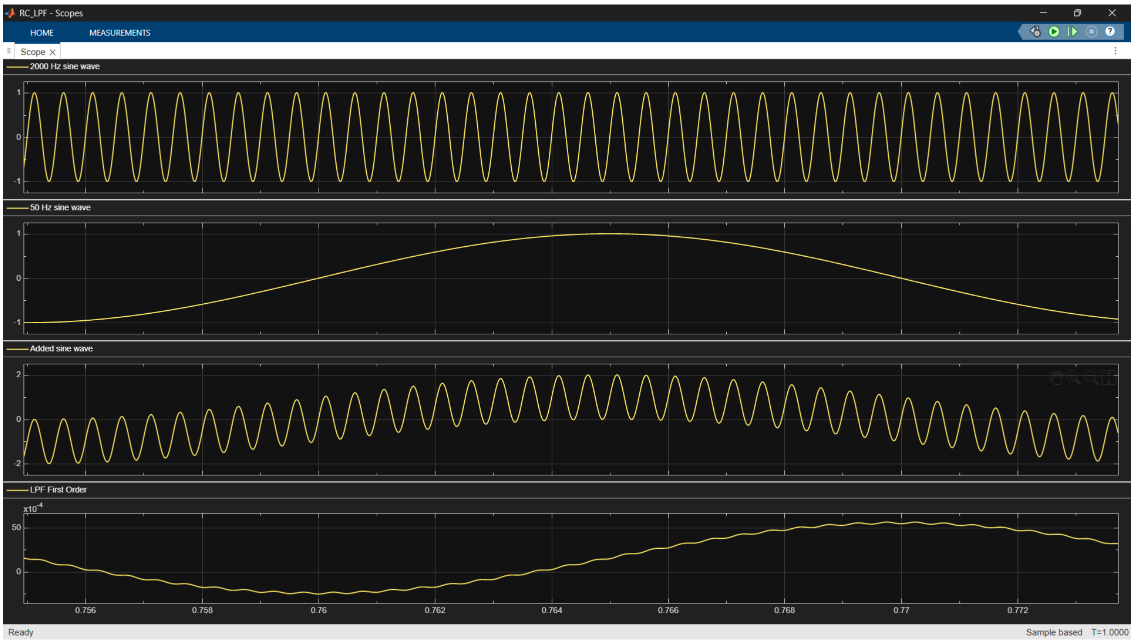


Figure 12: Added two sine wave result in MATLAB

Task 3: Study and Analysis of PV array

PV array is the most important topic in renewable energy. When the solar irradiance falls on PV array, which converts light energy into electrical energy. Basically, PV array is diode but the difference is here, it's acts as a source.

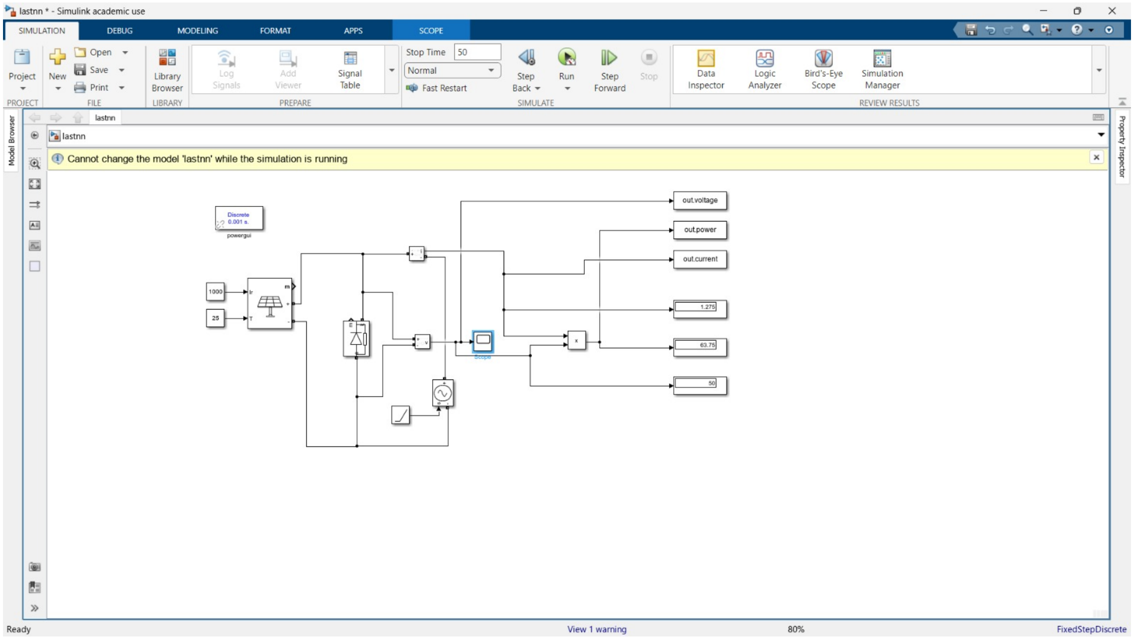


Figure 13: simple PV circuit design in MATLAB

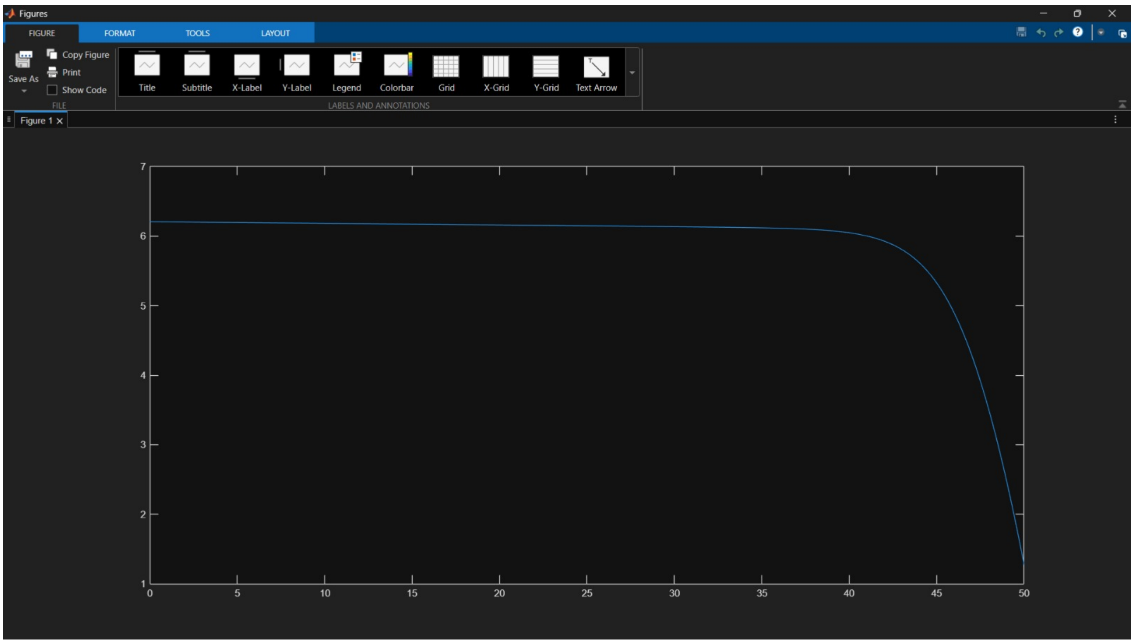


Figure 14: IV characteristics of PV array graph in MATLAB

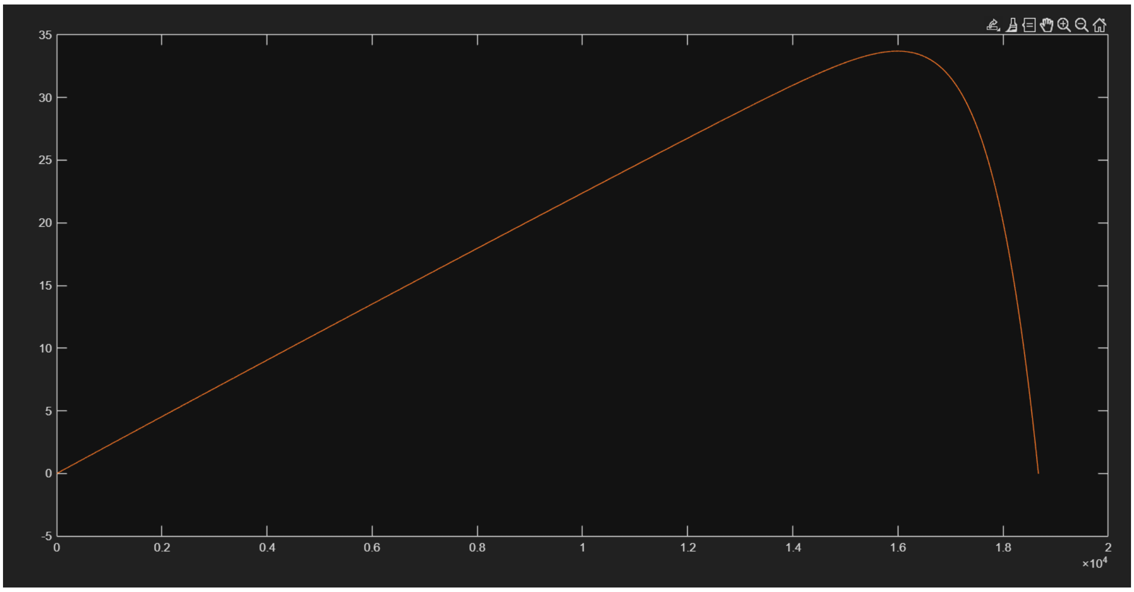


Figure 15: PV characteristics of PV array graph in MATLAB

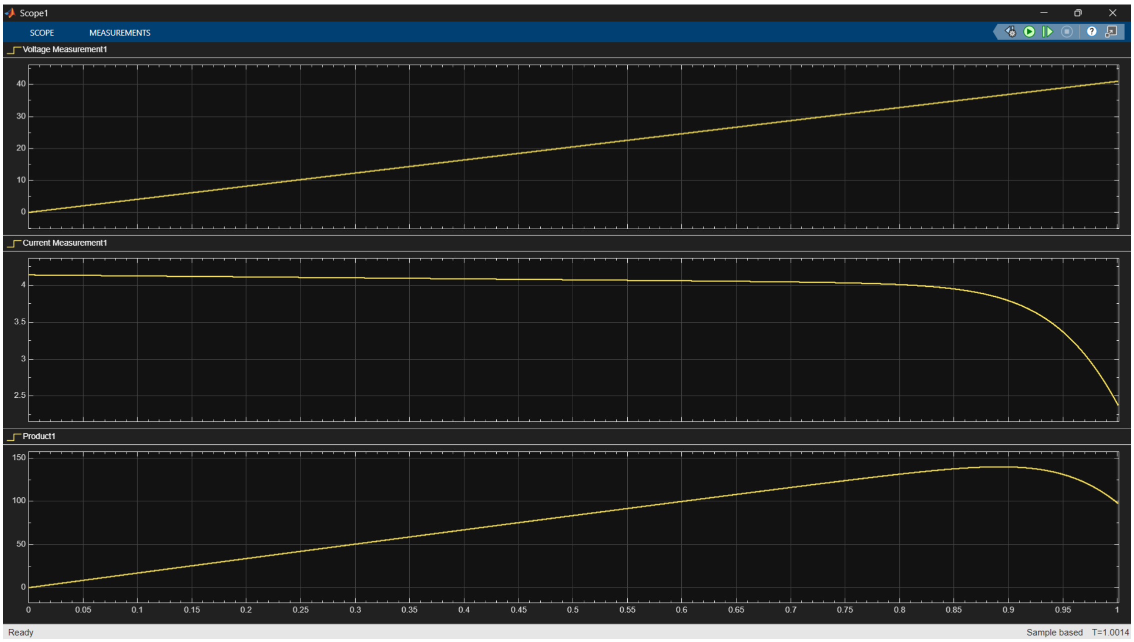


Figure 16: Voltage, Current, Power vs time in scope, MATLAB

When more than one PV array is connected—either in series or in parallel—the system behavior changes significantly under practical operating conditions.

In real-world scenarios, PV arrays often experience different irradiance levels due to shading, cloud movement, dust, or panel orientation. In series connections, mismatch in irradiance causes the array current to be limited by the lowest-irradiated panel, leading to power loss and potential hotspot issues. In parallel connections, voltage remains nearly constant, but current sharing becomes unequal, with higher-irradiance arrays supplying more current.

Therefore, variations in irradiance have a direct impact on overall power output, efficiency, and reliability of PV systems.

1. MATLAB CIRCUITS AND RESULTS

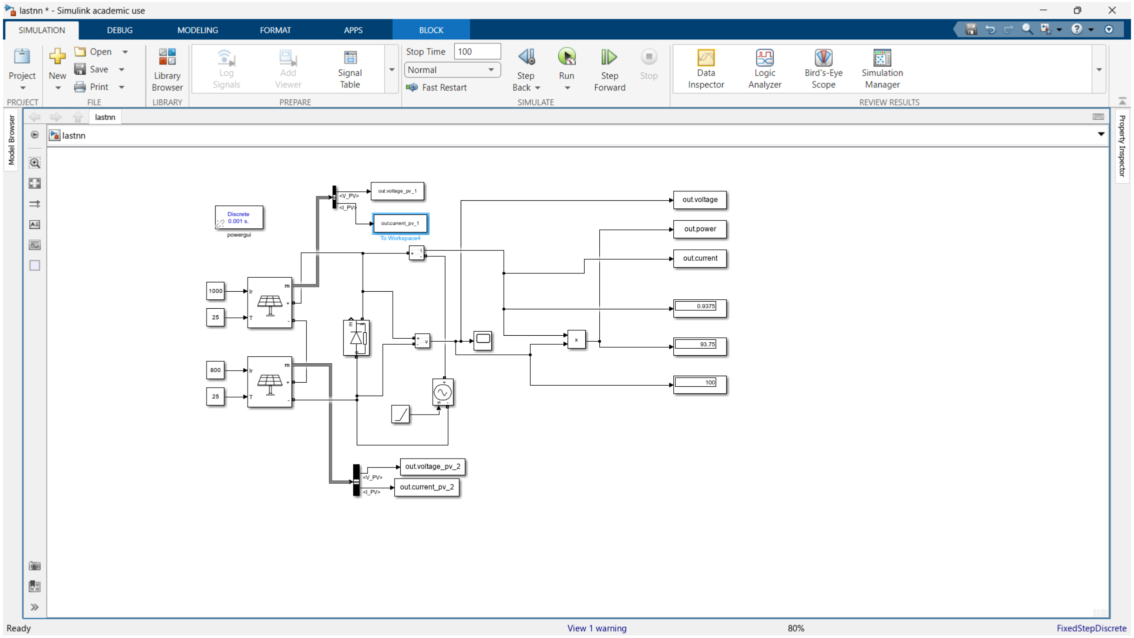


Figure 17: PV series with Boost Converter in MATLAB

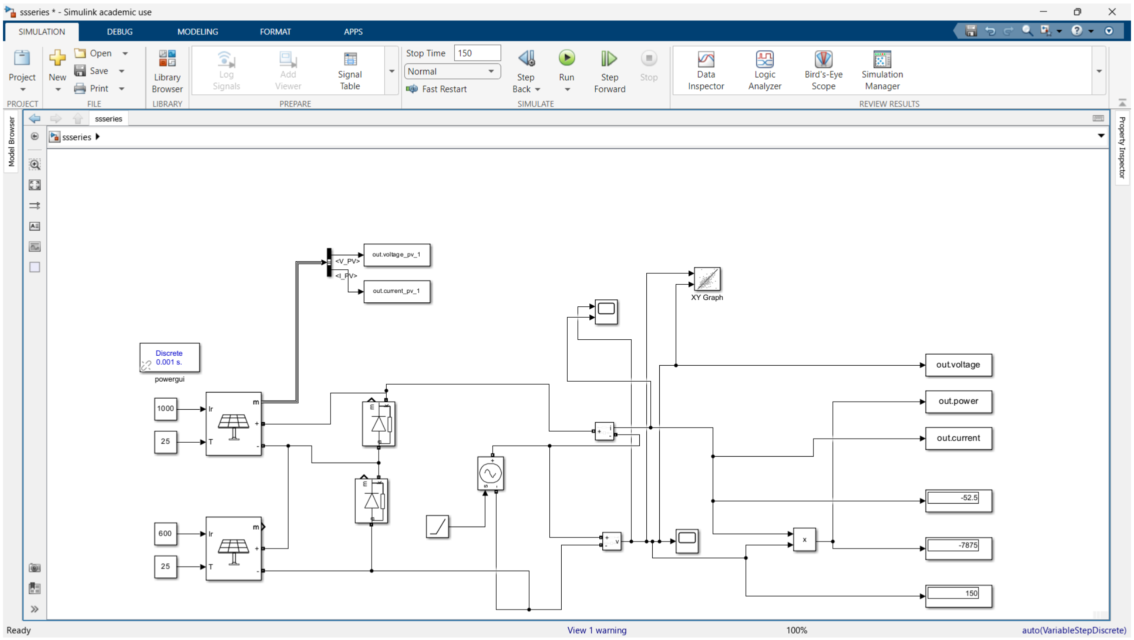


Figure 18: PV parallel with Boost Converter in MATLAB

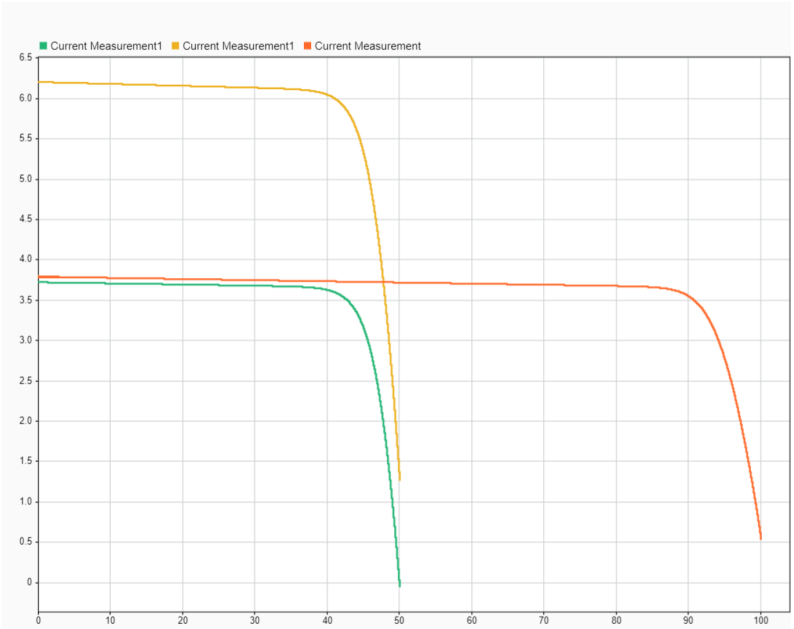


Figure 19: IV characteristics of PV series array (under different irradiance)graph in MATLAB

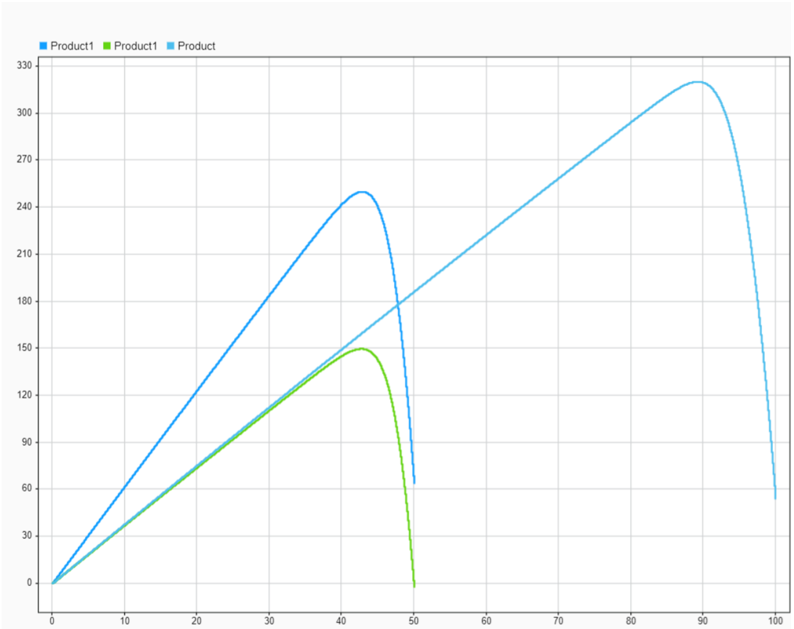


Figure 20: PV characteristics of PV series array (under different irradiance)graph in MATLAB

From Figure 19 and Figure 20, clear I–V and P–V characteristics are not observed. Under varying irradiance, series-connected PV arrays produce different voltages, which prevents achieving maximum output. In parallel connections, each PV array operates independently despite irradiance variation. Hence, the I–V and P–V characteristics remain similar to those in Figure 14 and Figure 15.

Task 4: Study and Develop MPPT algorithm

According to Ohm's law,

$$R = \frac{V}{I} \tag{1}$$

Therefore, when the load resistance is varied, the corresponding voltage and current also change.

As a result, the characteristic curve shown in Figure 21 exhibits both minimum and maximum operating points. The blue circular markers on the orange curve represent the measured data points.

Among these points, one operating point corresponds to the maximum voltage and current, indicating the optimal operating (maximum power) point of the system.

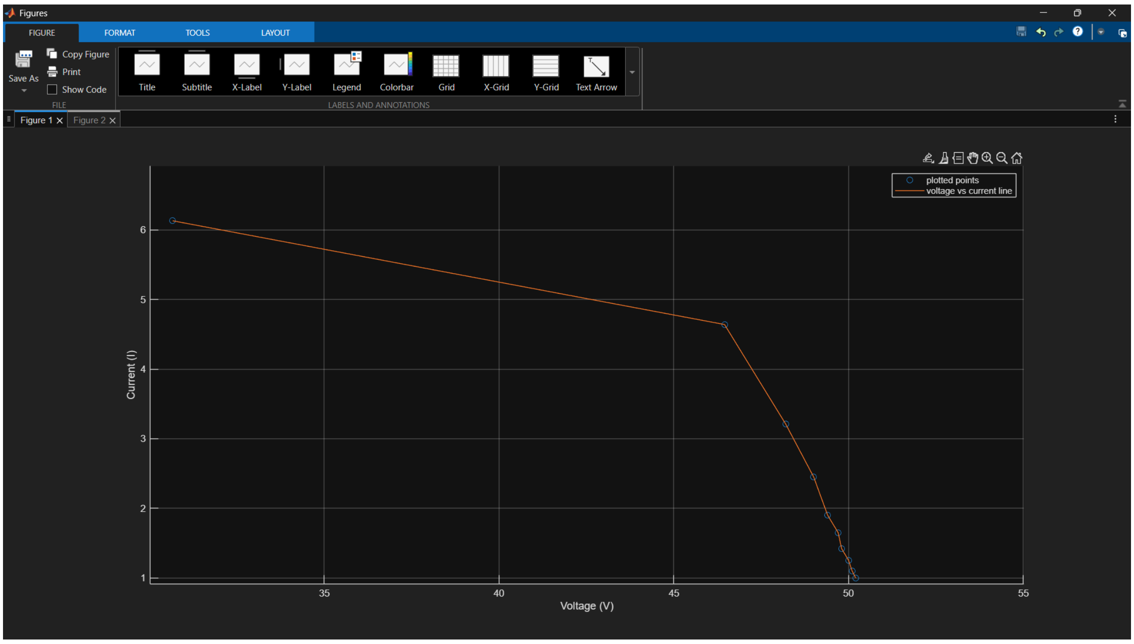


Figure 21: PV Voltage vs current with respect to changes in load graph in MATLAB

When using a PV array with a boost or buck converter, the duty cycle or gate pulse of the MOSFET (switch) is varied to achieve maximum output power.

This process is known as Maximum Power Point Tracking (MPPT). In real-world scenarios, the maximum power point is not constant, so MPPT continuously tracks this point at every time interval to maintain maximum power output.

1. Study case and understanding the logic of MPPT P&O algorithm

Perturb and Observe (P&O) MPPT Algorithm:

- P&O is a widely used **Maximum Power Point Tracking (MPPT)** method for PV systems.
- The algorithm **perturbs the duty cycle** of the DC-DC converter and **observes the effect on PV power**.
- If power **increases**, the perturbation direction is **kept the same**; if power **decreases**, the direction is **reversed**.
- This process is repeated iteratively to **track the maximum power point** under varying irradiance and temperature conditions.
- Simple, efficient, and **easy to implement in MATLAB** using measured voltage and current data.

P&O MPPT MATLAB Code for our existing data:

```matlab
% P&O MPPT simulation using given V/I arrays
% prints a neat table and plots results

clear; clc; close all;

% Given data
V_PV = [34.8 35.3 32.83 32.29 28.92 19.16 11.24 5.6 ...
  2.139 0.5142 0.1873];
I_PV = [3.4 4.2 5.26 5.29 8.324 8.45 8.47 8.49 ...
  8.50 8.513 8.515];
N = numel(V_PV);

% P&O parameters (tune as needed)
duty = 0.5;          % initial duty (0..1)
step = 0.01;         % perturbation step
dir  = 1;            % +1 or -1
min_duty = 0.0;
max_duty = 1.0;
epsilon = 1e-4;      % noise threshold (W)

% Preallocate logs
idx = (1:N)';
P   = zeros(N,1);
dP = zeros(N,1);
```

```
dV = zeros(N,1);
duty_before = zeros(N,1);
duty_after  = zeros(N,1);
dir_before  = zeros(N,1);
dir_after   = zeros(N,1);
notes = cell(N,1);

% internal previous values
v_prev = 0;
p_prev = 0;

for k = 1:N
    v = V_PV(k);
    i = I_PV(k);
    p = v * i;

    % record state before update
    duty_before(k) = duty;
    dir_before(k)  = dir;

    % deltas
    dP(k) = p - p_prev;
    dV(k) = v - v_prev;

    % P&O update
    if abs(dP(k)) <= epsilon
       notes{k} = '(noise - no change)';
    else
       if dP(k) > 0
           duty = duty + dir * step;
           notes{k} = '(power up - keep dir)';
       else
           dir = -dir;
           duty = duty + dir * step;
           notes{k} = '(power down - reverse)';
       end
     end

     % clamp duty
     duty = max(min_duty, min(max_duty, duty));

     % store after-update state
     duty_after(k) = duty;
     dir_after(k)  = dir;
     P(k) = p;

     % update previous measurements
     v_prev = v;
     p_prev = p;
end
```

```
% Build neat table
T = table(idx, V_PV', I_PV', P, dP, dV, ...
    duty_before, duty_after, dir_before, ...
    dir_after, notes, ...
    'VariableNames', {'Idx', 'V_V', 'I_A', 'P_W', ...
    'dP_W', 'dV_V', 'DutyBefore', 'DutyAfter', ...
    'DirBefore', 'DirAfter', 'Note'});

% Improve numeric display precision
T.V_V        = round(T.V_V, 3);
T.I_A        = round(T.I_A, 3);
T.P_W        = round(T.P_W, 3);
T.dP_W       = round(T.dP_W, 4);
T.dV_V       = round(T.dV_V, 4);
T.DutyBefore = round(T.DutyBefore, 4);
T.DutyAfter  = round(T.DutyAfter, 4);

% Print the table
disp('P&O MPPT simulation results:');
disp(T);

% Optional: write CSV
writetable(T, 'mppt_pno_results.csv');

% Plot power and duty
figure('Name','P&O MPPT Results', ...
      'NumberTitle','off','Position',[100 100 700 500]);

subplot(2,1,1);
plot(idx, P, '-o','LineWidth',1.2);
xlabel('Sample index');
ylabel('Power (W)');
grid on;
title('PV Power');

subplot(2,1,2);
plot(idx, duty_after, '-s','LineWidth',1.2);
xlabel('Sample index');
ylabel('Duty (0..1)');
grid on;
title('Duty After Update');

% Plot P and Duty with two y-axes
figure('Name','Power & Duty','NumberTitle','off');
yyaxis left;
plot(idx, P, '-o','LineWidth',1.2);
ylabel('Power (W)');
yyaxis right;
plot(idx, duty_after, '-s','LineWidth',1.2);
```

```
ylabel('Duty (0..1)');
xlabel('Sample index');
grid on;
legend('Power','Duty','Location','best');
```

OUTPUT:

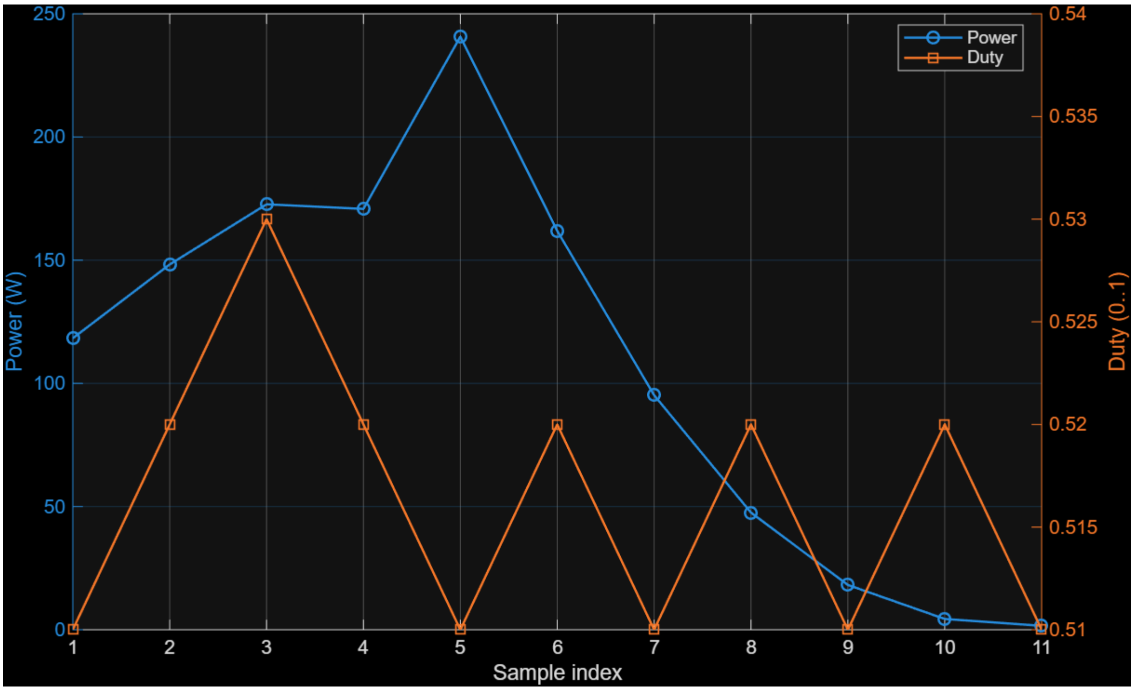
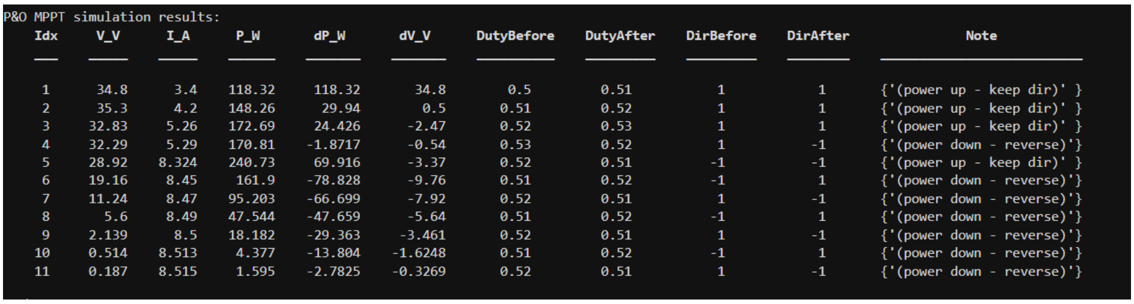


Figure 22: Duty Cycle increases and decreases respect to power changes graph



P&O MPPT simulation results:

Idx	V_V	I_A	P_W	dP_W	dV_V	DutyBefore	DutyAfter	DirBefore	DirAfter	Note
1	34.8	3.4	118.32	118.32	34.8	0.5	0.51	1	1	{'(power up - keep dir)' }
2	35.3	4.2	148.26	29.94	0.5	0.51	0.52	1	1	{'(power up - keep dir)' }
3	32.83	5.26	172.69	24.426	-2.47	0.52	0.53	1	1	{'(power up - keep dir)' }
4	32.29	5.29	170.81	-1.8717	-0.54	0.53	0.52	1	-1	{'(power down - reverse)'}
5	28.92	8.324	240.73	69.916	-3.37	0.52	0.51	-1	-1	{'(power up - keep dir)' }
6	19.16	8.45	161.9	-78.828	-9.76	0.51	0.52	-1	1	{'(power down - reverse)'}
7	11.24	8.47	95.203	-66.699	-7.92	0.52	0.51	1	-1	{'(power down - reverse)'}
8	5.6	8.49	47.544	-47.659	-5.64	0.51	0.52	-1	1	{'(power down - reverse)'}
9	2.139	8.5	18.182	-29.363	-3.461	0.52	0.51	1	-1	{'(power down - reverse)'}
10	0.514	8.513	4.377	-13.804	-1.6248	0.51	0.52	-1	1	{'(power down - reverse)'}
11	0.187	8.515	1.595	-2.7825	-0.3269	0.52	0.51	1	-1	{'(power down - reverse)'}

Figure 23: Numerical data of duty cycles changes with respect to power changes

MPPT P&O Algorithm:

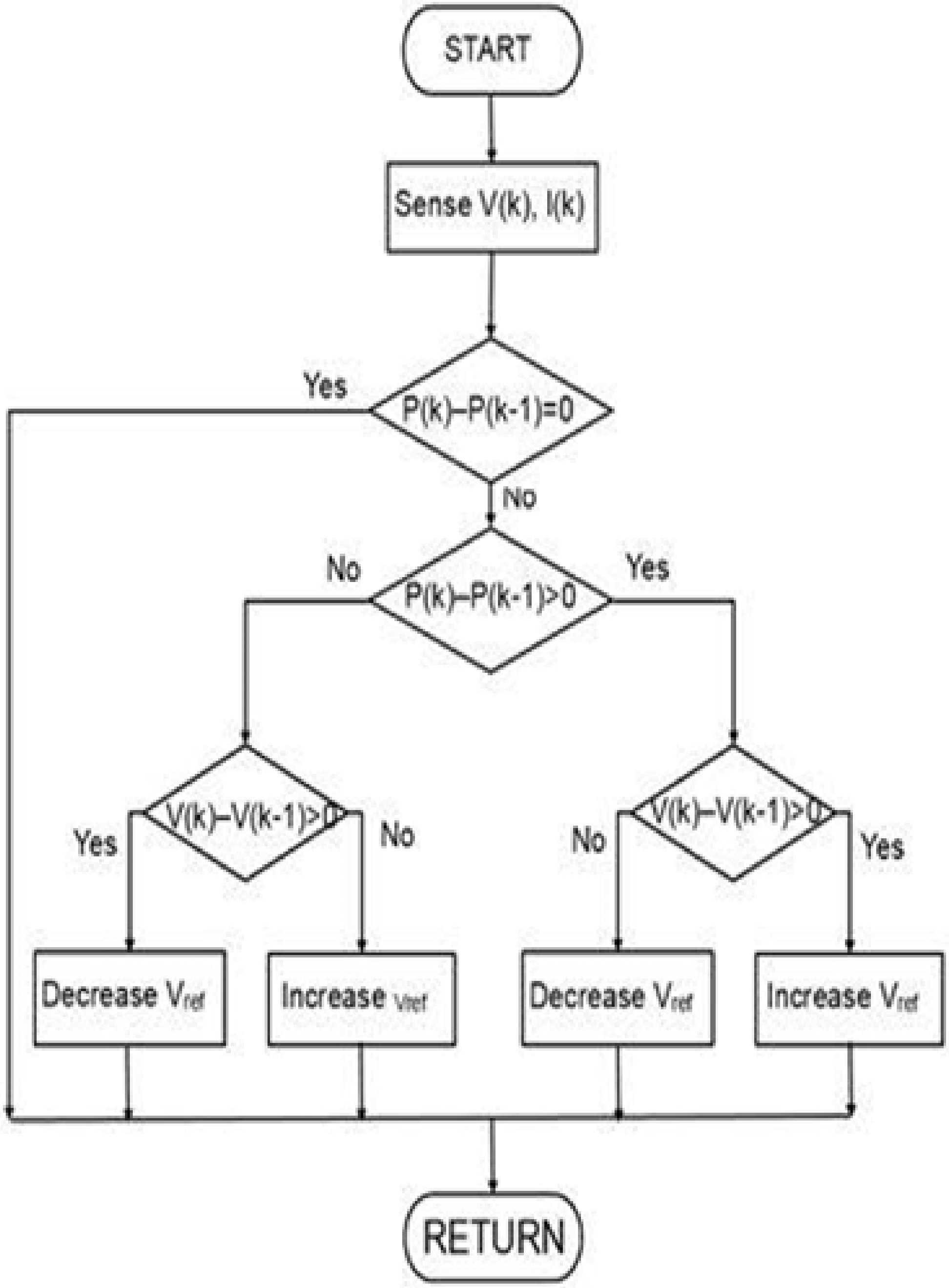


Figure 24: Flowchart of Perturb & Observe Algorithm

MATLAB Circuit & Results:

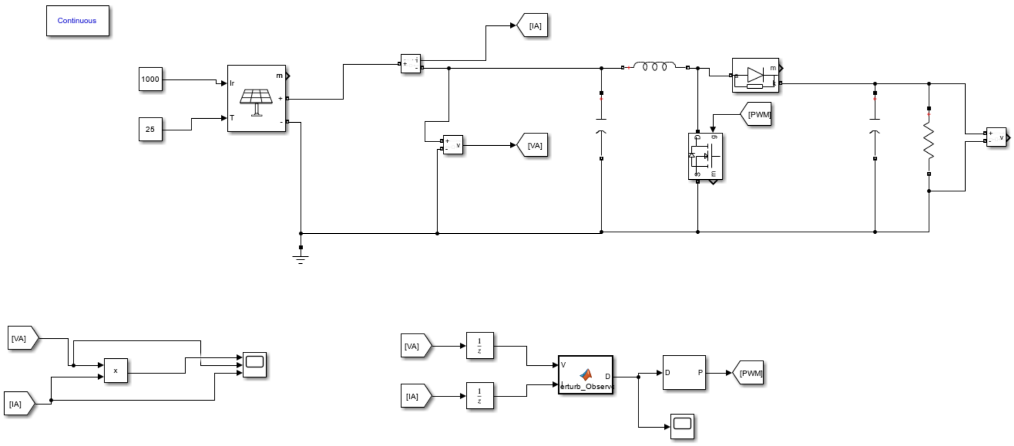


Figure 25: PV array circuit with MPPT P&O Algorithm in MATLAB

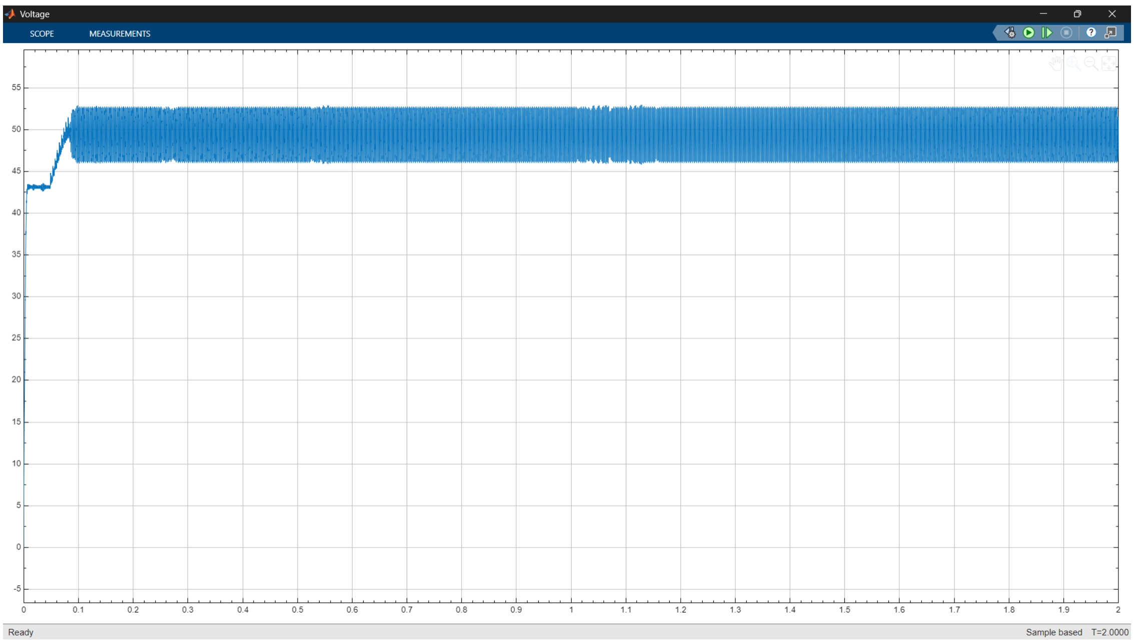


Figure 26: MPPT P&O Algorithm Output Graph in MATLAB

Our MPPT P&O Algorithm as per logic of Figure 24:

```
%----MPPT P&O Algorithm inside the matlab function block...
% on Figure 25 MATLAB CIRCUIT-----

function D = Perturb_Observe(V,I)

persistent V_p P_p D_p
if isempty(V_p)
     V_p = 0; % Previous voltage = 0
     P_p = 0; % Previous power = 0
     D_p = 0; % Previous Duty cycle = 0
end
% Measure power
P = V*I;

if (P-P_p == 0) % Check the condition for MPP
    D = D_p;
end
if (P-P_p > 0) % Compare powers
   if (V-V_p > 0)  % Compare Voltages
       D = D_p-0.005;
       else
           D = D_p+0.005;
       end
    else
          if (V-V_p >0)  % Compare Voltages
              D = D_p+0.005;
          else
              D = D_p-0.005;
          end
    end

P_p = P;
V_p = V;
D_p = D;
end
```

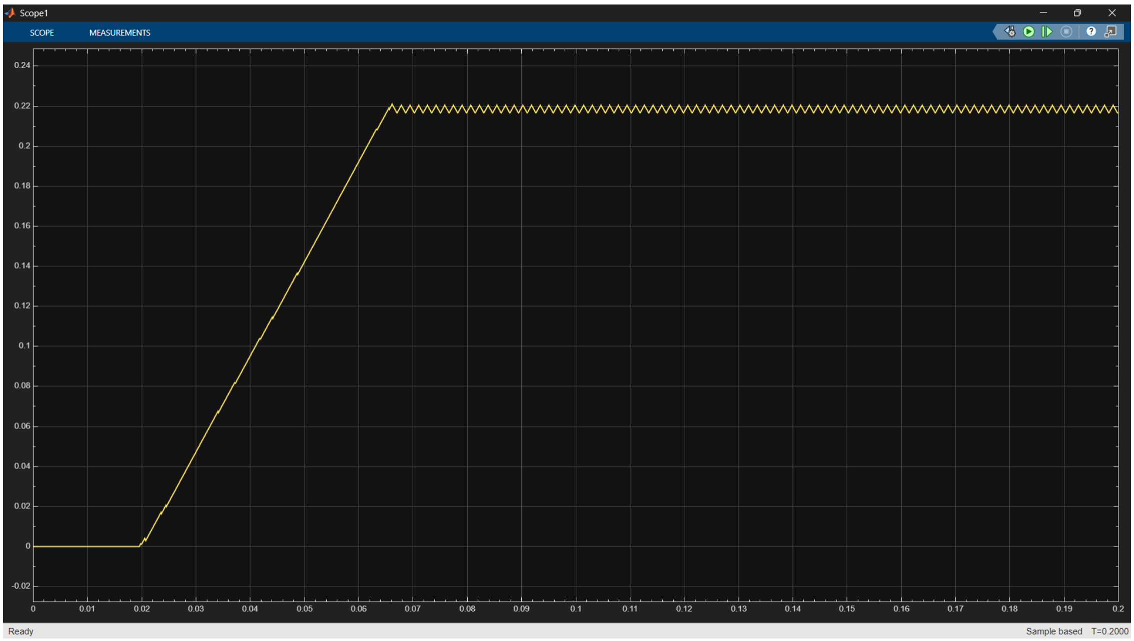


Figure 27: MPPT P&O Algorithm working on scope in MATLAB

On Figure 27, the MPPT P&O algorithm starts at 20 ns due to the chosen sample time, but ideally it should start at 0 sec or very close to 0 sec. This indicates that the algorithm is working properly, and the start time can be adjusted by reducing the sample time.

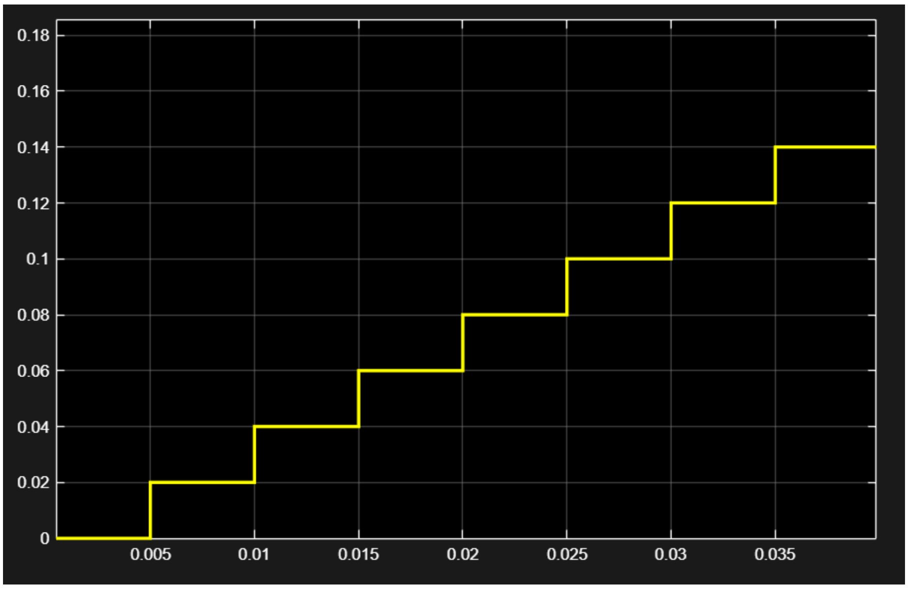


Figure 28: MPPT P&O Algorithm working on scope in MATLAB

Hardware Demonstration:

We are greatly thankful to Mr. Arun, a research scholar under Dr. B. Bijukumar at NIT Karaikal, Pondicherry. He demonstrated and explained both theoretical and practical scenarios, providing deep knowledge of the components in a Boost Converter circuit with a PV array as the input source. He also introduced a simulation tool to model the PV array output, explained the required software, discussed the hardware design, and taught us how to read datasheets—skills that are extremely helpful to us.

Figure 29: Hardware setup of Boost converter with PV simulation instrument in simulation lab

Figure 30: Live working demonstration of hardware setup in simulation lab

Figure 31: BLDC motor hardware demonstration

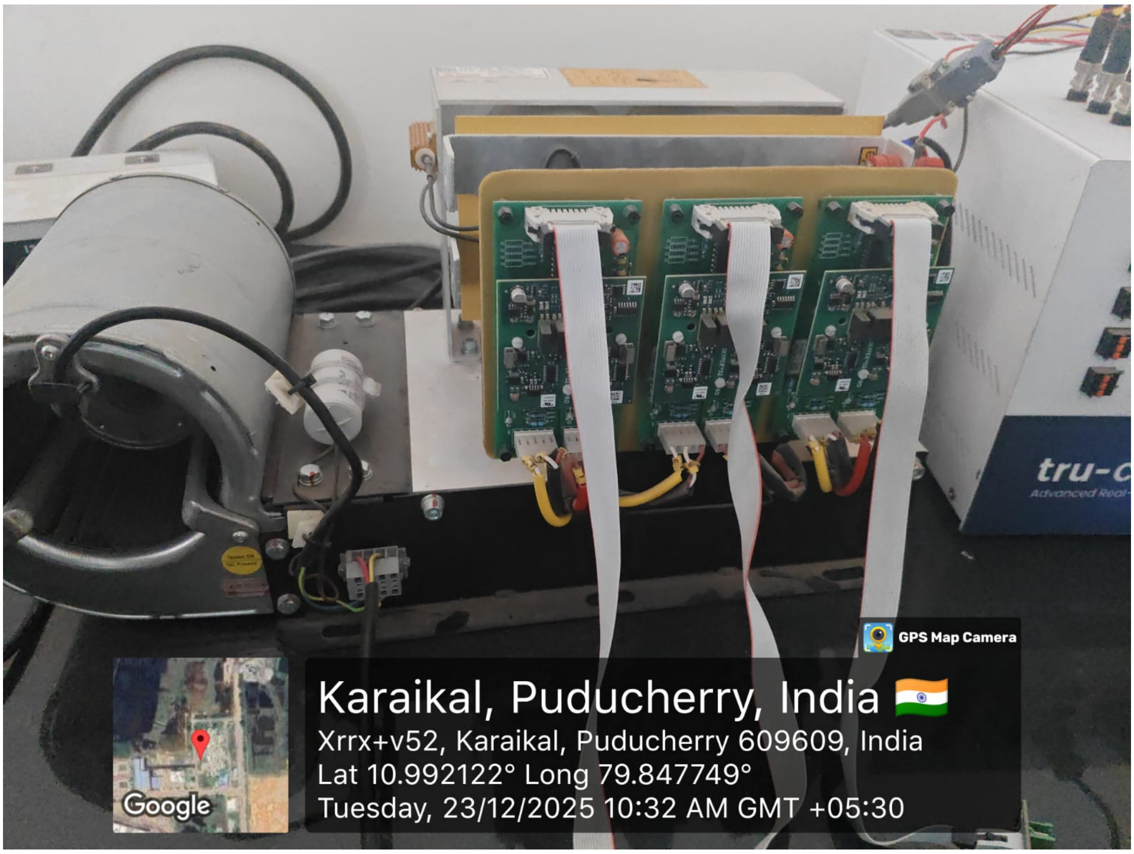


Figure 32: BLDC motor with driver circuit hardware setup

Conclusion

This two-week internship at the National Institute of Technology, Karaikal, proved to be a highly enriching academic and technical experience, offering valuable exposure to the practical aspects of power electronics and renewable energy systems. The program successfully bridged the gap between theoretical concepts and real-world applications through structured learning, simulations, and hardware demonstrations.

During the internship, I gained hands-on experience in the design, simulation, and performance analysis of a DC-DC Boost Converter using MATLAB and LTspice. This exercise strengthened my understanding of converter operation, efficiency evaluation, and the influence of component losses under different operating conditions. The study of RC low-pass and high-pass filters further enhanced my knowledge of signal conditioning and frequency-domain behavior in power electronic circuits.

A major focus of the internship was photovoltaic (PV) systems. I analyzed PV array characteristics, examined I-V and P-V curves, and studied the effects of series and parallel configurations under non-uniform irradiance conditions. The implementation and simulation of a Maximum Power Point Tracking (MPPT) system using the Perturb and Observe (P&O) algorithm in MATLAB/Simulink provided practical insight into optimizing power extraction from PV systems.

Additionally, the hardware demonstrations involving boost converters, PV simulators, and BLDC motor drives offered real-time exposure to industrial-grade systems. These sessions reinforced the relationship between simulation results and physical system behavior, improving my understanding of design constraints and practical considerations.

Overall, this internship significantly enhanced my technical competence in power electronics, simulation tools, and renewable energy systems. It also improved my analytical thinking, problem-solving skills, and appreciation for the role of power electronics in sustainable energy solutions. I am sincerely thankful to the faculty and researchers at NIT Karaikal for their guidance and support. The knowledge and skills gained through this internship will serve as a strong foundation for my future academic pursuits and professional career in electrical engineering.