Laboratory Manual

EE-153L – Introduction to Electrical Engineering

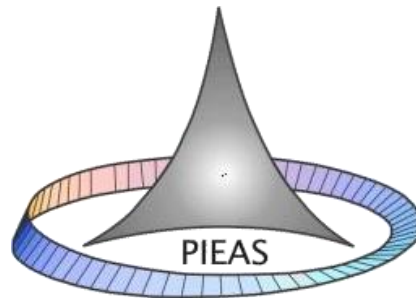### Fall 2025

Instructor

Engr. Noman Khan

Lab Engineers

Engr. Bilal Haider
Engr. Yasir Kamal

Department of Electrical Engineering
Pakistan Institute of Engineering & Applied Sciences
Islamabad, Pakistan

EE-153 Introduction to Electrical Engineering Lab Manual

Experiment 1

Introduction to LTspice

1.1 Objective

The main objective of this lab is to introduce students to LTspice, a widely used simulation tool for electrical and electronic circuit design. By the end of this experiment, students will be able to:

- Understand the LTspice interface and its basic tools.
- Create simple circuits and run simulations.
- Compare theoretical calculations with simulation results.
- Develop confidence in using software-based simulation for circuit verification.

1.2 Introduction

LTspice is a free SPICE-based simulation software developed by Analog Devices. It is a powerful tool for analyzing circuits without physically building them. SPICE stands for Simulation Program with Integrated Circuit Emphasis, and it is the standard for electronic circuit simulation.

Using LTspice, students can:

- Build schematics of circuits quickly and easily.
- Analyze the behavior of circuits under different conditions.
- Visualize voltage and current waveforms in real time.
- Perform parameter sweeps, transient analysis, and frequency analysis.

Why use LTspice in academics?

Simulation saves time, cost, and resources. Instead of buying physical components and breadboards, students can test circuits virtually and correct design mistakes before moving to hardware implementation. This makes it a valuable skill for future electronics engineers.

1.3 Software Requirements

LTspice XVII or Later – Free download at: https://www.analog.com/en/resources/design-tools-and-calculators/ltspice-simulator.html

Operating System: Windows 10/11 or macOS

EE-153 Introduction to Electrical Engineering Lab Manual

1.4 LTspice Simulation Procedure

i. Open LTspice: Launch the LTspice application and select File → New Schematic to start a new design.

ii. Placing Components: Press F2 or click the Component Icon to open the component library. Select the required components like resistors, capacitors, voltage sources, etc., and place them on the schematic.

iii. Wiring the Circuit: Press F3 or click the Wire Tool to connect the components with virtual wires.

iv. Setting Component Values: Right-click on a component to set its value, e.g., a resistor value of 1 kΩ or a voltage source of 5 V.

v. Adding Power and Ground: Place a voltage source using F2 → Voltage. Every circuit must have a Ground node, added using F2 → Ground.

vi. Defining Simulation Type: Go to Simulate → Edit Simulation Cmd and select: - DC Operating Point
- Transient Analysis
- AC Sweep Analysis depending on the experiment type.

vii. Running the Simulation: Click the Running Man icon (or press Ctrl + R). The waveform viewer opens, showing voltages and currents.

viii. Viewing Results: Click on wires or nodes to probe voltages. Click on components to measure current.

Tip: Save the schematic regularly using File → Save As to avoid losing your work.

1.5 LAB Tasks

Task 1

How to place components and run a simple simulation?
Instructions:
1. Build a simple circuit:
   - Voltage Source = 5 V
   - Resistor = 1 kΩ connected to ground.
2. Run a DC Operating Point Analysis.
3. Measure the current through the resistor and voltage across it.

Task 2
How total resistance affects current in series vs. parallel resistor configurations.
Instructions:

1. **Series Circuit:**

   - Voltage Source V=10V

- Two resistors:

    - R1=2kΩ
    - R2=3kΩ

  - Connect the resistors in series with the voltage source.
  - Run a DC Operating Point analysis and record the total current.

2. **Parallel Circuit:**

  - Use the same resistors, but connect R1 & R2 in parallel across the voltage source.
  - Run the simulation and record the total current.

Task 3

Analyze a circuit that combines both series and parallel connections.

Instructions:

1. Build the following circuit:

    - Voltage Source V=12V
    - R1=3kΩ
    - R2=4kΩ
    - R3=5kΩ

Configuration:

- R2 & R3 are connected in parallel.
- This parallel combination is in series with R1.

2. Run a DC Operating Point analysis.
3. Measure:

  - Total circuit current.
  - Voltage across the parallel branch.
  - Current through each parallel resistor.

1.6 Lab Report

EE-153 Introduction to Electrical Engineering Lab Manual