

California State University, Northridge

College of Engineering & Computer Science

**Department of Electrical and Computer
Engineering**

**ECE 240L Electrical Engineering Fundamentals
Laboratory Report**

By Mr. Zachary Ramos and Mr. Steven Ovanessian



Summer 2025

Instructor: Sequare Daniel-Berhe, Ph.D.

California State University, Northridge
College of Engineering and Computer Science
Electrical and Computer Engineering Department

ECE 240L Electrical Engineering Fundamentals Laboratory Reports Title

{Design Electrical Engineering Fundamentals circuit to meet a desired specification!}

No	Main Lab Topics	Design Specifications are given in Lab Manual/ it will be given during lab sessions
1	Laboratory Instruments and Reports	<u>Combine Lab # 1 and #2</u> Being Familiar with DC Power Supply, AC Function Generator, Oscilloscope and all other Laboratory Instruments as well as Lab Report Writing
2	Oscilloscopes,	
3	DC Circuits Design, Experimental Test and Analysis	
4	Computer Simulations Design on DC & AC Circuits, Simulation and Experimental Test as well as Analysis	
5	Design Experiment – Circuit I - Implementing Mesh Analysis – Design, Simulation and Experimental Test as well as Analysis	
6	Application of Network Theorems (Thevenin's, Norton's & Superposition Theorems) Circuit Design, Simulation and Experimental Test as well as Analysis	
7	Design Experiment – Circuit II - Implementing Maximum Power Transfer – Design, Simulation and Experimental Test as well as Analysis	
8	Operational Amplifiers Design, Simulation and Experimental Test as well as Analysis	
9	First Order Circuits Design, Simulation and Experimental Test as well as Analysis	
10	Design Experiment – Circuit III - Implementing RC Circuits, Simulation and Experimental Test as well as Analysis	
11	Second Order Circuits Design, Simulation and Experimental Test as well as Analysis	
12	Impedance and Admittance Circuits Design, Simulation and Experimental Test as well as Analysis	
13	Frequency Response AC Circuits Design, Simulation and Experimental Test as well as Analysis	
14	Passive Filters Design, Simulation and Experimental Test as well as Analysis	
15	Diode Circuits Design, Simulation and Experimental Test as well as Analysis	<u>We will combine and write one lab report for Lab# 15 & Lab# 16 with a lab title:</u> Simple Diodes & Transistors DC Biasing with Amplification Design, Simulation and Experimental Test as well as Analysis
16	Transistor DC Biasing Circuits Design, Simulation and Experimental Test as well as Analysis	

Practical Application - Final Lab Project Titles:

[1] DC Motor

[4] RF tag

[2] Wireless Charger

[5] Radio Tuner

[3] Smoke Detector

[6] Night Light

ECE 240L Electrical Engineering Fundamentals Laboratory

Lab Course Description

The ECE 240 course deals with introduction to the theory and analysis of electrical circuits; basic circuit elements including the operational amplifier; circuit theorems; DC circuits; forced and natural responses of simple circuits; sinusoidal steady state analysis and the use of a standard computer-aided circuit analysis program. The course also focuses on power, energy, impedance, phasors, frequency response calculations and their use in circuit design. The corresponding ECE 240L lab includes laboratory instruments and reports, oscilloscopes, DC circuits, computer simulations, design experiments – circuit I, network theorems, design experiments – circuit II, operational amplifiers, first order circuits, design experiments – circuit III, second order circuits, impedance and admittance, frequency response and passive filters. This lab will pursue extensive simulation exercises and practical experiments with the help of discrete components and/or IC's circuits by applying Computer Aided Design (CAD) simulation programs/software like PSpice. In addition, analytical and graphical tools will be used to examine and explain the basic design and analysis of fundamental electrical engineering circuits. The main focus is on design and analysis of Electrical Engineering and Computer Science problems and solutions.

Preparations for this Lab are:

- Revision of basic physics (Electricity and Magnetism) and mathematics.
- Be familiar with PSpice & MATLAB. Introduce yourself also to other industrial simulation programs like Spice, Micro-CAP 9, MultiSim, Quartus II, Altera Max Plus II whenever you have time.

Texts and Materials

- ECE240 Lab Manual and
- Introduction to Electric Circuits, 8th ed., by Svoboda JA, DORF RC, John Wiley & Sons, Inc., 2010.
- PSPICE, by Cadence Corporation: <http://www.cadence.com/>
- PSPICE by OrCAD MicroSim Corporation <http://www.microsim.com/>
- In addition to the textbook, main lecture notes and supplemental lecture materials will be offered to provide additional materials for the class.

Main Labs Outline

In this lab the students will learn how to design, analyze & experimentally test the following main labs.

- Laboratory Instruments and Reports,
- Oscilloscopes,
- DC Circuits,
- Computer Simulations,
- Design Experiments – Circuit I,
- Network Theorems,
- Design Experiments – Circuit II,
- Operational Amplifiers,
- First Order Circuits,
- Design Experiments – Circuit III,
- Second Order Circuits,
- Impedance and Admittance,
- Frequency Response and
- Passive Filters.

Student Learning Outcomes and Lab Course Learning Objectives

Student Learning Outcomes

After completing this laboratory course the students should have an:

- Ability to apply laboratory equipment relating to basic circuit analysis and design.
- Ability to design, analyze and experimentally test DC circuits.
- Ability to design, analyze and experimentally test various types of circuits based on given specification designs (Design Experiments – Circuit I, II & III).
- Ability to design, analyze and experimentally test network theorems.
- Ability to design, analyze and experimentally test operational amplifiers.
- Ability to design, analyze and experimentally test first order circuits.
- Ability to design, analyze and experimentally test second order circuits.
- Ability to design, analyze & experimentally test any circuits related to impedance & admittance.
- Ability to design, analyze and experimentally test circuits with frequency responses and
- Ability to design, analyze and experimentally test Passive filters.

Lab Course Learning Objectives

- The objective of Electrical Engineering Fundamentals lab is to have the student design, analyze and build basic circuits using discrete components and/or IC's.
- The lab will help students to how to solve D.C. circuit problems with independent and dependent sources, op-amps and resistors using nodal analysis, mesh analysis, superposition, source transformations and Thevenin/Norton equivalent circuits.
- The lab will help students to how to find the complete response for first and second-order circuits to input signals modeled by waveforms that are DC, step, window, ramp, decaying exponential and sinusoidal.
- The lab will help students to how to apply phasors and the concept of impedance to analyze circuits with sinusoidal input under steady-state conditions and to find the frequency response of linear, time-invariant circuits.
- The lab will help students to how to design 1st & 2nd-order filters given specifications in terms of 3-db bandwidth & center freq.
- The lab will help students to how to use PSpice for the design & analysis of aforementioned fundamental electrical ct.'s simulations.
- To accomplish this, the student will:
 - Learn how to analyze basic circuits using concepts learned in Electrical Engineering Fundamentals course
 - Learn basic circuits and network theorems.
 - Learn basics of operational amplifiers, first order circuits and second order circuits.
 - Learn how to design basic circuit to meet a desired specification
- More emphasis will be given on their applications, and extensive hardware experimental designs and simulation exercises will be examined with the help of discrete components and/or IC designs as well as using simulation programs.
- Hardware experimental circuit designs, Computer Aided Design (CAD) simulation programs/software, analytical, graphical, and instrumentation tools will be used to examine and explain the basic building blocks of advanced electronics circuit design, analysis and test.
- The objective of this lab is also to introduce and be familiar with current industrial and educational simulation programs/software like PSpice/Spice, Micro-CAP, MultiSim, Quartus, Altera Max Plus, and MATLAB.
- Students will learn applications and troubleshooting of basic circuits and systems. They will be exposed to hands on practice and/or computer simulation and analysis of fundamental circuits & systems.
- The main focus is on design, analysis and test of Engineering and Computer Science problems and solutions.
- This basic circuit design, analysis and test lab provides the student with the basic knowledge necessary to understand the operation and application of fundamental circuits using discrete components and/or IC's.
- Following the completion of this lab the student should be well versed in electronics circuit design, analysis, and test and should be able to continue with advanced courses.

Contribution of Lab Course to Meeting the Professional Component

- This advanced lab contributes primarily to the students' knowledge of Electrical, Electronics and Computer Engineering topics, and does provide fundamental design experience.
- The following statement indicates which of the following considerations are included in this lab primarily: economic, environmental, ethical, political, societal, health and safety, manufacturability, sustainability.
 - Solution: Issues of manufacturability, economics, environmental, and health and safety are primarily relevant in the context of electronic design.
- Thus, the contribution of ECE 240L course and lab to meeting the professional component becomes
 - Engineering Design 2.5 (with the Laboratory)
 - Engineering Science 1.5 (with the course)

Relationship of Lab Course to Undergraduate Degree Program Objectives and Outcomes

This course supports the achievement of the following program objectives and outcomes:

- *An ability to apply knowledge of mathematics, science, and engineering to the analysis of electrical and electronic engineering problems.*
- *An ability to identify, formulate and solve electrical engineering problems. An ability to design and conduct scientific and engineering experiments, as well as to analyze and interpret data.*
- *An ability to design systems that include hardware and/or software components within realistic constraints such as cost, manufacturability, safety and environmental concerns.*
- *An ability to communicate effectively through written reports and oral presentations.*
- *An ability to use modern engineering techniques for analysis and design*
- *An ability to analyze & design complex devices and/or systems containing hardware and/or software components. This leads to recognition of the need for & an ability to engage in life-long learning.*
- *An ability to apply knowledge of mathematics including differential equations, linear algebra, complex variables and discrete math to the analysis of electrical engineering problems.*
- *An ability to be competitive in the engineering job market and/or to continue their studies at the graduate level.*

In addition, the following information describes how the lab course contributes to the undergraduate program objectives and supports the achievements of the following expected lab outcomes.

- This lab provides a basic understanding of the methods in the analysis and design of basic circuits using IC's and/or discrete components. It gives each student a solid knowledge based in the fundamentals of electrical and computer engineering.
- This lab develops in each student the basic skills of problem solving and critical thinking.
- Project works, labs and examinations require students to think critically based on engineering and science concepts, and at the same time, let students practice and develop problem solving techniques.
- This lab provides extensive hardware experimental designs and simulation exercises that require students to put methods learned in lectures into practice. This lab develops in each student the team-working skills necessary to perform effectively as an engineer. The project work allows students to discuss in groups at least in pairs but each student will do his/her own work.
- This lab develops in each student good writing skills so that they are able to communicate technical material effectively and clearly. The lab project works have formal reports that require student to document theory, experimental outputs and simulated results as well as their interpretations in an organized manner. These reports are graded based on writing skill (Clarity, Format, Completeness, Mechanics, Appearance, etc.), Technical Presentations/Descriptions, Procedures, Data and Figures, as well as Discussions and Conclusions.
- This lab imparts to each student a sense of ethical and professional responsibility. An understanding or professional and ethical responsibility is obtained through the strict enforcement of the Academic Honesty Policy of the University, which applies to all labs and projects.
- This lab develops basic skill in methods of design and analysis across a broad range of electrical and computer engineering areas. This lab provides knowledge and abilities to apply various techniques and theories to the operation and design of fundamental circuits.

Assessment of Student Progress Toward Lab Course Objectives

- Design oriented lab project works will be provided including extensive use of computer aided simulation and design techniques especially using PSPICE.
- Students' progress is measured in weekly based lab project work activities, demonstrations and presentations that exercise the analysis and design skills developed throughout the semester, and based on the overall final project work reports.

Lab Grading Standards:

A final letter grade is to be awarded to each enrolled student based on the grading system shown below.

Letter Grade	Percent of Total Points	Grade Points
A	90% to 100%	4.00
A-	85% to 89%	3.70
B+	80% to 84%	3.30
B	75% to 79%	3.00
B-	70% to 74%	2.70

Letter Grade	Percent of Total Points	Grade Points
C+	66% to 69%	2.00
C	60% to 64%	2.00
C-	55% to 59%	1.75
D	50% to 54%	1.00
F	49% & below	0.00

Note: 100% can be achieved by doing extra credit hardware experimental designs and/or simulation exercises, etc.

Grading Weights: The final grade will be determined by the below percentages.

Professionalism/Participation	10%
Weekly Lab Work Progress & Activities	35%
➤ Based on Hardware Experimental Designs Preparation	
➤ Based on Simulation Program Exercises Preparation (if necessary)	
Final Lab Project Work Demonstration & Presentation	15%
Design & Analysis of any Practical Application of this Lab Course	
➤ Based on Hardware Experimental Design Outcome	
➤ Based on Simulation Program Exercise Outcome (if necessary)	
Overall Lab Project Work Reports	40%
➤ Based on Simulation & Hardware Experimental Designs Outcome	
➤ Based on Technical Writing Skill (Clarity, Format, Completeness, Mechanics, Appearance, Technical Presentations/Descriptions, etc.)	
➤ Based on Procedures, Data & Figures, Discussions & Conclusions	
<hr/>	
Total	100%

❖ **Extra Credit (10%)** If you design & analyze any of the extra credit lab works you can get extra credits.

❖ **Professionalism/Participation** – is based on:

- ◆ **Reliability** - Attended lab; brought required texts and materials to lab; reading textbook, handouts/ assigned materials; met deadlines for graded assignments, kept instructor informed of future absences.
- ◆ **Punctuality** – Came to lab on time & returned from break(s) on time. **Please do not miss lab sessions.**
- ◆ **Positive Attitude** – Showed intent to learn, staying focused on task; participated actively in lab, discussions and group activities;
- ◆ **Respect for others** – Turned off audible signal of cell phone, pager while in lab room; remained for duration of each lab session; etc.
- ◆ **Teamwork** – Encouraged responsible behavior by group members; attended & contributed to group meetings; completed own group assignment(s) on time; worked to keep group focused on project goals; helped group members understand & complete their assigned tasks;
- ◆ **Trustworthiness** – Avoided all dishonest acts including plagiarism; refrained from injuring others...

To be successful –

- You need to work hard - show an effort to study;
- You need to work smart – follow guidelines, methods, theorems, order of operations, etc.;
- You need to have positive attitude – think positively or open your mind to learn new methods;
- You need to study for two hours for every one hour lecture before you come to the next session;
- You need to study 70% of your study time alone and 30% in a group of three or five on weekly bases at a library.

Required Reading Sources

- Introduction to Electric Circuits, 8th ed., by Svoboda JA, DORF RC, John Wiley & Sons, Inc., 2010.
- PSPICE, by Cadence Corporation: <http://www.cadence.com/>
- PSPICE, by OrCAD Corporation (www.orcad.com) or refer to www.microsim.com/
- Main Lecture Notes, and Handouts

Supplemental Reading Sources/References

- Electric Circuits Analysis, 3rd ed., David E. Johnson and et al, John Wiley & Sons, Inc., 1999.
- Fundamentals of Electric Circuits, 3 ed. Charles Alexander et al.
- Electric Circuits, 3rd Ed., by Alexander and Sadiku, 2007.

Software and Demos

- PSpice Simulation Program – that provides fully interactive mixed analog and digital simulations.
- MATLAB is also required to study frequency response of analog circuits.
- For downloading PSpice refer to
 - PSPICE, by Cadence Corporation: <http://www.cadence.com/>
 - PSPICE by OrCAD MicroSim Corporation <http://www.microsim.com/>
- Introduce yourself to other industrial simulation programs like Spice, Micro-CAP, MultiSim, Quartus, Altera Max Plus whenever you have time.

BEING FAMILIAR WITH DC POWER SUPPLY, AC FUNCTION GENERATOR, OSCILLOSCOPE AND ALL OTHER LABORATORY INSTRUMENTS AS WELL AS LAB REPORT WRITING

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT: The objective of this lab is to learn the basic set-up of circuits and how to use the breadboard, DC power supplies, AC function generators, and lab equipment. The DC or AC power supply was used to power up the circuit. The circuit consisted of $1\text{k}\Omega$ and $2\text{k}\Omega$ resistors connected in series. The measures were made in different DC supply 6V, 5V, 3V, 1V. The voltages were measured using a digital multimeter. With the Oscilloscope, two different waveforms that were displayed with a frequency generator set at 1.26 kHz showed the square wave, sine wave, ramp wave, and triangle wave.

KEYWORDS: Direct Current (DC) Circuits, Alternating Current(AC) Circuits, DC Power Supply, AC Function Generator, Schematic, Breadboard, Grounding, Digital Multimeter, Oscilloscopes, Ohm's Law, Frequency, Period.

1-2.1 INTRODUCTION

This experiment is the introduction of essential tools and practices used in electrical laboratories, including power supplies, digital multimeters and breadboards. By investigating the basic principles of voltage, current, and resistance measurement, while applying Ohm's Law to real-world circuit components. The expected outcome is to become proficient in circuit assembly and measurement techniques. Setting a strong foundation for future experiments involving more complex electrical circuits.

1-2.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

Using a breadboard, two resistors of $1\text{k}\Omega$, $2\text{k}\Omega$ and an AC function generator. An experimental circuit of two resistors in series was constructed on a breadboard (Figure 1-2.1). An Oscilloscope was used to measure the input voltage of the circuit was measured on Channel 1 and the voltage across R_2 on Channel 2. Once the circuit and oscilloscope were constructed. Using the AC function generator the voltage was set to a peak-to-peak voltage of 3 and adjusted to a frequency of 1.26 kHz. Measurements were documented for a sine function, square function, triangular function, ramp function and a constant DC function.

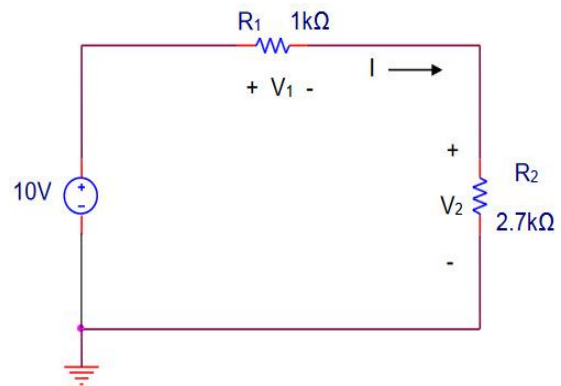


Fig 1-2.1 Simple Resistive Series circuit 1

1-2.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

AC Supply Vpp (volts)	V(in) (volts)	V(R2) (volts)
6	6.08	3.92
6	6	4
6	6	4
1	5.92	3.92

Table 1-2.1: AC Circuit Numeric Data

Equations

$$I \text{ (calc)} = V \text{ (in)} / [R1 + R2] \quad (1-2.1)$$

$$\% \text{Error} = (| \text{Measured} - \text{Calculated} | / \text{Measured})$$

$$* 100\% \quad (1-2.2)$$

$$V = I * R \quad (1-2.3)$$

$$F = 1/T \quad (1-2.4)$$

$$\text{Conversion Factor (K)} = V_{\text{peak}} / V_{\text{RMS}} \quad (1-2.5)$$

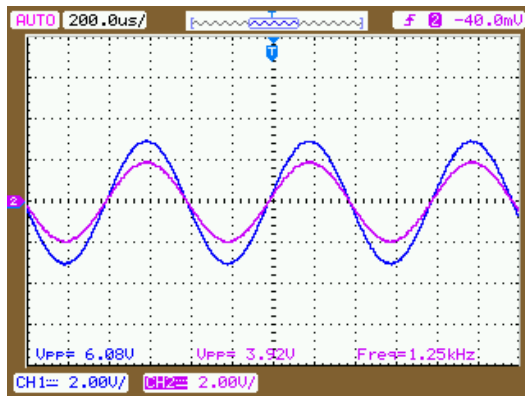


Fig. 1-2.2 Sine Waveform from first Simple Resistive Series Circuit for AC Analysis

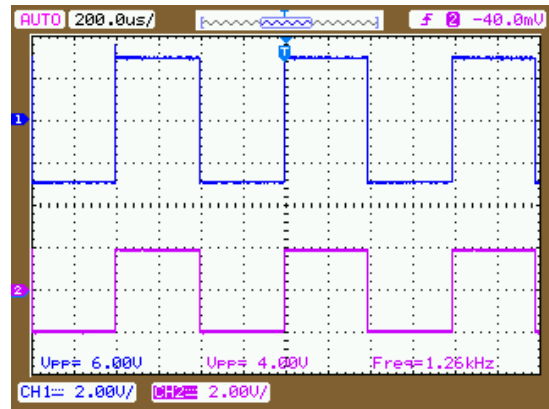


Fig 1-2.3 Square Waveform from first Simple Resistive Series Circuit for AC Analysis

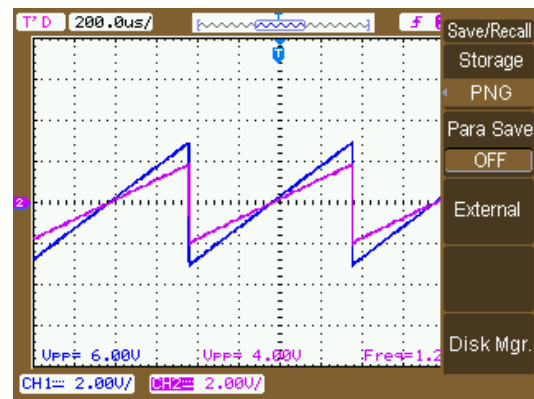


Fig. 1-2.4 Ramp Waveform from first Simple Resistive Series Circuit for AC Analysis

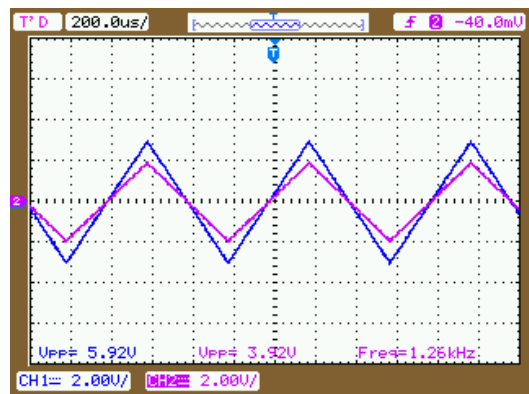


Fig. 1-2.5 Triangular Waveform from Simple Resistive Series Circuit for AC Analysis

1-2.4 DISCUSSION & CONCLUSION

In Labs 1 and 2, we explored the foundational use of essential lab instruments including the DC power supply, function generator, oscilloscope, and digital multimeter. Using PSpice

simulations and physical circuit assembly, we analyzed how different waveforms—sine, square, triangular, and ramp—affect voltage behavior in simple resistor circuits.

The recorded data showed consistent voltage readings across R2 for all waveform types, with minimal variation. For instance, with a 6V peak-to-peak sine wave at 1.26 kHz, we observed 6.08V at the input and 3.92V across R2, confirming theoretical expectations. Similar agreement was observed for other waveforms, affirming our grasp of voltage division and measurement techniques.

These experiments allowed us to:

- Apply laboratory equipment in basic circuit analysis and design.
- Design, simulate (via PSpice), and experimentally test DC circuits.
- Troubleshoot measurement discrepancies and compare theoretical, simulated, and actual results.
- Practice professional documentation for reporting results, essential for real-world engineering tasks.

By completing these labs, we achieved key Student Learning Outcomes (SLOs) such as the ability to experimentally test and analyze circuits, and key Lab Course Learning Objectives including the use of PSpice for simulation, hands-on circuit assembly, and waveform analysis. This foundation will support more advanced studies and design challenges in electrical engineering.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

DC CIRCUITS DESIGN, EXPERIMENTAL TEST & ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT: The objective of this lab is to calculate theoretical voltage values across various resistors wired either in parallel or series as well as currents and to compare the calculations with the measured result as well as the percentage error. This was done by using a DC power supply, ohmmeter, breadboard and resistors. The theoretical values were calculated using various techniques including Ohm's law, Kirchhoff's voltage law (KVL), Kirchhoff's current law (KCL), voltage division, current division. Lastly, we had to find the error percentage in each resistor, voltage, current. Using the percent error formula.

KEYWORDS: DC Power Supply, Resistors, DMM, ELVIS, Kirchhoff's Current Law, Kirchhoff's Voltage Law, Branches, Conductance, Series Circuits, Parallel Circuits and Ohmmeter

3.1 INTRODUCTION

The objective of this Lab was to analyze the electrical behavior of series-parallel resistor networks using both theoretical calculations and experimental validation. This experiment aimed to reinforce fundamental concepts such as Ohm's Law, Kirchhoff's Voltage Law (KVL), and Kirchhoff's Current Law (KCL), which are essential for circuit analysis in both academic and practical settings. By constructing specific resistor configurations, we measured voltage drops and current distributions using digital multimeters, comparing these values against our calculated expectations.

Through this lab, we also became more proficient in using standard lab instrumentation and improved our understanding of how electrical energy is divided across components in complex networks. The combination of predictive analysis, circuit construction, and measurement verification provided a comprehensive approach to validating core electrical engineering principles and prepared us for more advanced circuit design and testing.

3.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

To begin Lab 3, all resistors were measured with a digital multimeter to verify their nominal values before constructing the circuits. The primary circuits tested was Figure 3.4 and 3.5. Figure 3.5 consisted of five resistors in a series-parallel configuration powered by a 10V DC source. The circuit was assembled on a breadboard using standard jumper wires and resistors with values: $R_1=1.1\text{ k}\Omega$, $R_2=4.7\text{ k}\Omega$, $R_3=2.7\text{ k}\Omega$, $R_4=3.3\text{ k}\Omega$, and $R_5=1.1\text{ k}\Omega$. The positive terminal of the power supply was connected to R_1 , and the circuit was grounded at the junction of R_3 , R_4 , and R_5 to establish a common reference point. Voltage measurements were taken across each resistor using a digital multimeter, ensuring that all leads shared a common ground reference. In parallel, the same circuit was simulated in PSPICE, where DC sweep analysis was used to measure voltages and currents at key nodes and components. The experimental and simulation results were compared to verify

the application of Ohm's Law, Kirchhoff's Laws, and voltage division across series-parallel branches. All data was recorded for analysis and percent error calculations.

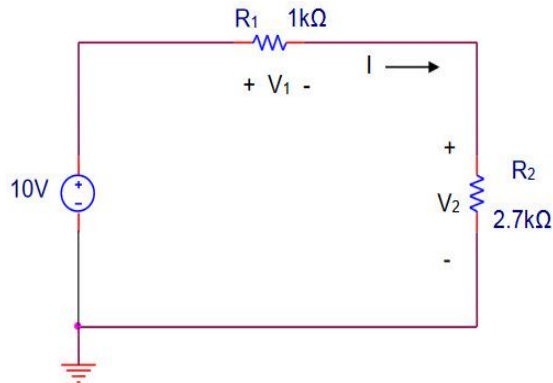


Fig 3.1 Simple Resistive Series Circuit 1

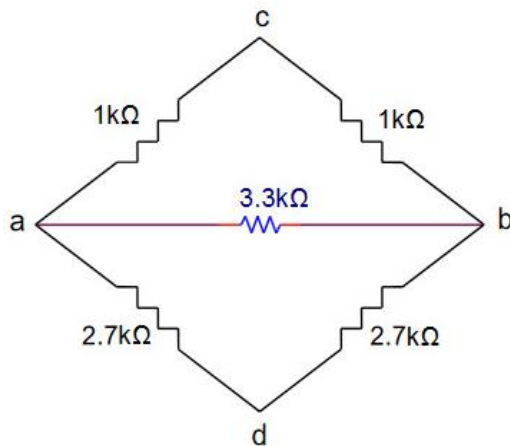
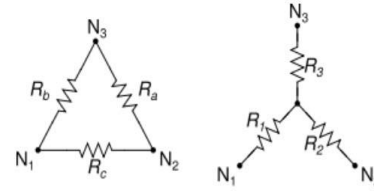


Fig 3.2 Circuit used for Y-Delta Conversion



$$\begin{aligned} R_1 &= \frac{R_b R_c}{R_a + R_b + R_c} & R_a &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_1} \\ R_2 &= \frac{R_a R_c}{R_a + R_b + R_c} & R_b &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_2} \\ R_3 &= \frac{R_a R_b}{R_a + R_b + R_c} & R_c &= \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_3} \end{aligned}$$

Fig.3.3 Delta → Wye Conversion Formulas

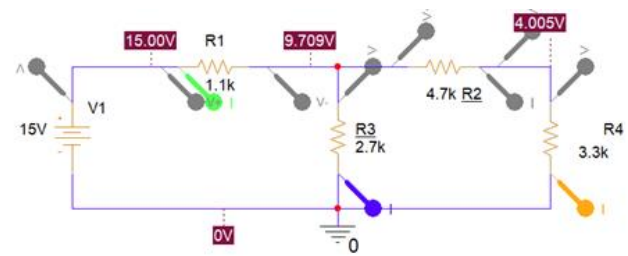


Fig 3.4 Circuit used for parallel circuit

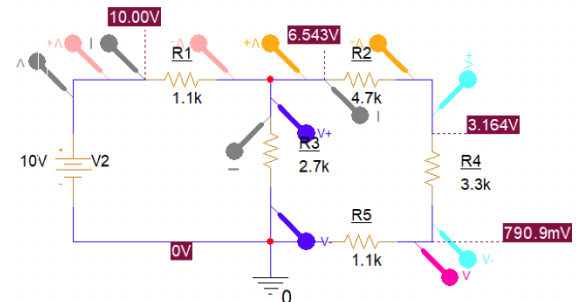


Fig 3.5 Circuit with parallel and series resistors

3.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

Resistance	Calculated (Ω)	Measured (Ω)	Percent Error
R_{ab}	1.05k	910	13.33%
R_{ac}	1.1k	779	36.61%
R_{cd}	1.9k	1.79	5.79%

Table 3.1 Values for measured resistance and calculated from circuit in Fig 3.2

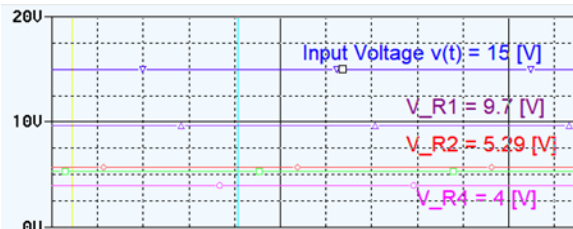


Fig 3.5 Voltage simulations for parallel circuit in Fig 3.4

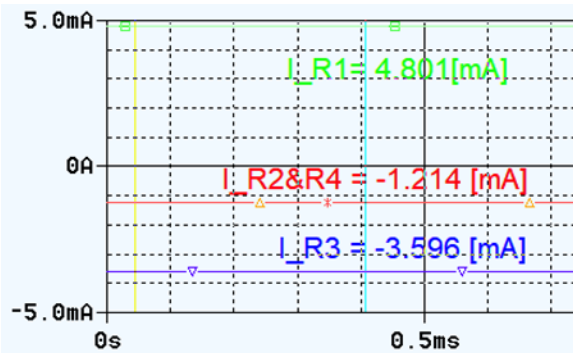


Fig 3.6 Current simulations for parallel circuit in Fig 3.4

Trace Color	Trace Name	Y1
	X Values	407.289u
CURSOR 1,2	I(R1)	4.8097m
	I(R3)	-3.5960m
	-I(R4)	-1.2137m
	I(R2)	-1.2137m
	V(V1:+,R1:2)	5.2907
	V(R1:2,R2:1)	5.7042
	V(V1:+)	15.000
	V(R3:2)	9.709
	V(R4:1)	4.0051

Fig 3.7 Simulation table for parallel circuit

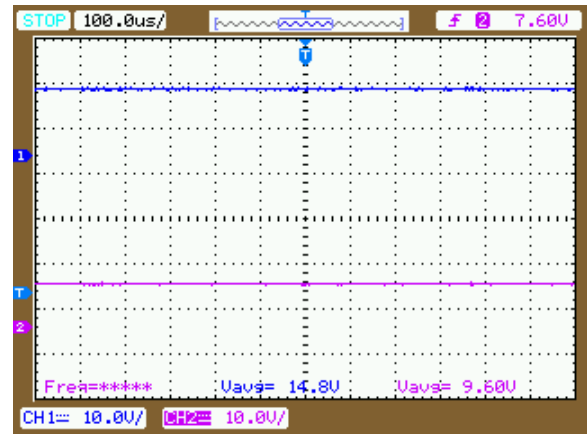


Fig 3.8 Experimental results for parallel circuit Voltage input and V_3

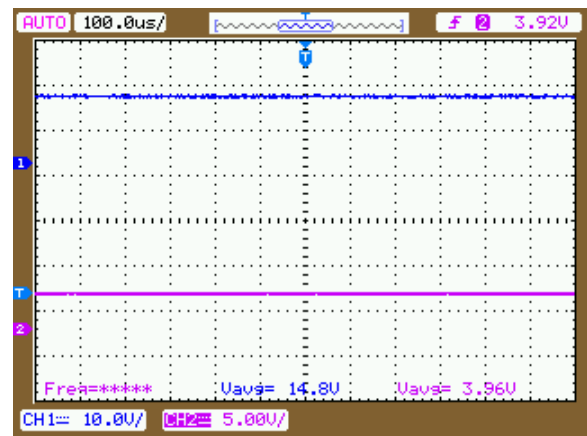


Fig 3.9 Experimental results for parallel circuit Voltage input and V_4

	V_3	V_4
Calculated	9.7	3.99
Measured	9.6	3.95
Percent Error	1.03%	1.15%

Table 3.2 Values of voltage measured and calculated in circuit from Fig 3.4

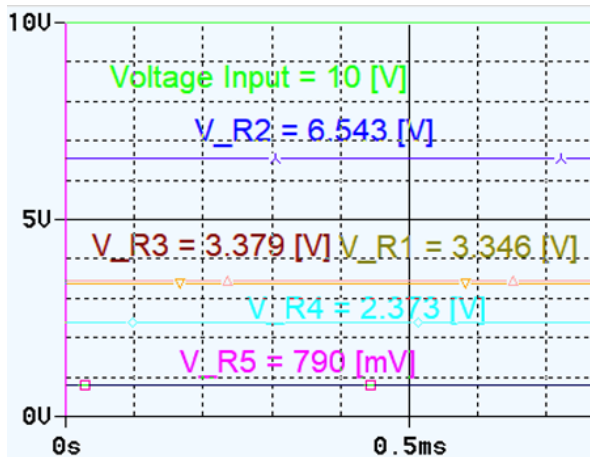


Fig 4.1 Simulated Voltage of circuit in Fig 3.5

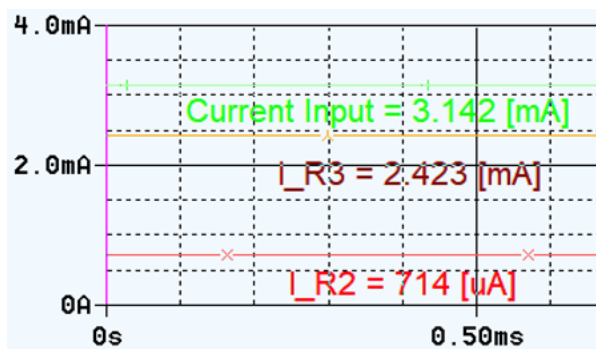


Fig 4.2 Simulated Current of Circuit in Fig 3.5

	X Values	10.000n
CURSOR 1,2	V(R4:1)	790.945m
	V(R2:2,R4:1)	2.3728
	V(R3:2,R2:2)	3.3795
	V(R1:1,R3:2)	3.4567
	V(R3:2,0)	6.5433
	V1(V2)	10.000
	I(R1)	3.1425m
	I(R2)	719.041u
	I(R3)	-2.4234m
	I(R4)	-719.041u
	I(R5)	-719.041u

Table 4.1 Simulated Current and voltage of each resistor in Fig 3.5

Equations

$$\text{Ohm's Law: } V = I \cdot R \quad (3.1)$$

$$\text{Voltage Divider: } V_R = (R/R_{eq}) \cdot V_{\text{supply}} \quad (3.2)$$

for resistors in series

$$\text{Kirchhoff's Voltage Law: } \sum V = 0 \quad (3.3)$$

$$\text{Kirchhoff's Current Law: } \sum I = 0 \quad (3.4)$$

3.4 DISCUSSION & CONCLUSION

In Lab 3, we focused on analyzing and verifying the behavior of a complex series-parallel DC circuit, specifically Figure 3.13 from the 2024 ECE240L lab manual. The experiment involved both PSPICE simulation and physical circuit testing using discrete components. The circuit was carefully constructed on a breadboard using five resistors, and voltage drops across each resistor were measured using a digital multimeter. The simulation and experimental results were compared with theoretical values derived from Ohm's Law and Kirchhoff's Laws. The outcomes demonstrated close alignment, with minor percent errors attributed to component tolerances and measurement limitations.

Throughout this lab, we applied multiple core skills, including reading schematics, setting up circuits on a breadboard, using lab equipment correctly, and performing voltage analysis across complex nodes and loops. The PSPICE simulations allowed for efficient verification of current paths and voltage distributions before conducting hands-on measurements. This not only streamlined the process but also served as a strong learning tool for confirming analytical predictions.

This lab successfully reinforced several **Student Learning Outcomes**, particularly:

- The ability to apply lab equipment in basic circuit analysis and design.
- The ability to design, simulate, and experimentally test series-parallel resistor networks.

- The ability to troubleshoot circuit construction issues and interpret voltage behavior in complex branches.

In terms of **Course Learning Objectives**, this lab achieved:

- The development of skills to analyze D.C. circuits using Ohm's Law, KVL, and KCL.
- The use of PSpice as a CAD simulation tool for analyzing voltage and current distributions.
- Hands-on experience in building and testing a series-parallel circuit using discrete components.
- The analysis and comparison of theoretical, simulated, and experimental data to validate circuit performance.

Additionally, this lab experience contributed to building professional skills such as report writing in a technical format, critical thinking in analyzing errors, and collaborative troubleshooting—skills essential for academic success and professional engineering practice. By completing Lab 3, we built a deeper understanding of voltage division and current paths in multi-branch circuits, preparing us for future labs involving network theorems and more advanced circuit behavior.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.

- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.

- [3] S. Daniel-Berhe, "ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus," Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

COMPUTER SIMULATIONS DESIGN ON DC & AC CIRCUITS, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

**Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu**

ABSTRACT:

The objective of this lab combined PSPICE simulation and hands-on testing to analyze DC and AC resistor circuits. In Fig. 4.1, we measured input voltage, node voltages, and branch currents to verify Ohm's law and KCL, comparing analytical, simulated, and experimental results. In Fig. 4.2, we conducted two studies: a square-wave input and a sine-wave input, recording voltages across R1, R2, and total circuit current. Simulated and experimental data showed strong agreement, with minor deviations due to component tolerances and instrument loading.

KEYWORDS: DC circuit analysis, AC circuit analysis, PSPICE simulation, transient response, square-wave input, sine-wave input, node voltage, branch current, Ohm's law, Kirchhoff's Current Law (KCL), and Kirchhoff's Voltage Law (KVL)

4.1 INTRODUCTION

This Lab focuses on applying PSPICE simulation and practical measurements to analyze the behavior of basic resistor networks under both DC and AC excitation. The objective is to model and evaluate two circuits, Fig. 4.1 for DC operation and Fig. 4.2 for transient AC response by measuring input voltages, node voltages, branch currents, and component voltage drops. The lab reinforces fundamental circuit laws such as Ohm's law, Kirchhoff's Voltage Law (KVL), and Kirchhoff's Current Law (KCL) while developing proficiency in simulation

tools, laboratory instrumentation, and systematic comparison of theoretical, simulated, and experimental results.

4.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

In this lab PSPICE was used to simulate Fig. 4.1 and Fig. 4.2 from the lab manual. For Fig. 4.1, exact resistor values and a DC source were assigned, with voltage and current markers placed to measure the input voltage, three node voltages, branch currents, and voltage drops. A Bias Point Analysis provided steady-state DC results. For Fig. 4.2, two transient profiles were created: a square-wave (VPULSE) and a sine-wave (VSIN) input, both matching lab specifications, with probes across R1, R2, and the input, plus a loop current probe. Transient Analysis was run over multiple cycles. Experimentally, both circuits were built on a breadboard using a DC supply, function generator, oscilloscope, and DMM. Measurements for input voltage, resistor voltages, and loop current were taken for each case, matching simulation conditions. Data were compiled in tables to compare theoretical, simulated, and experimental results, enabling verification of circuit behavior under both DC and AC excitation.

4.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

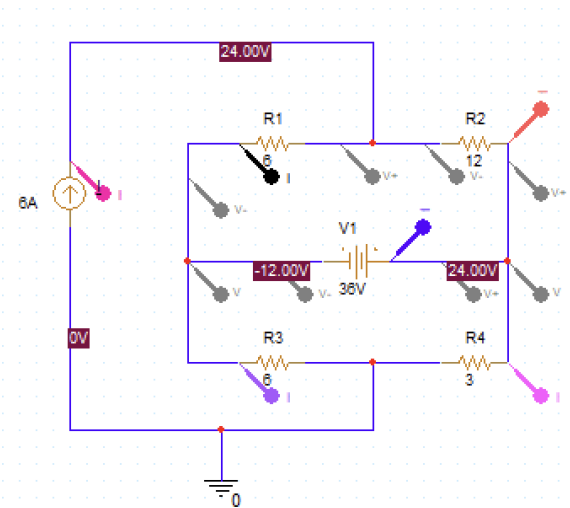


Figure 4.1

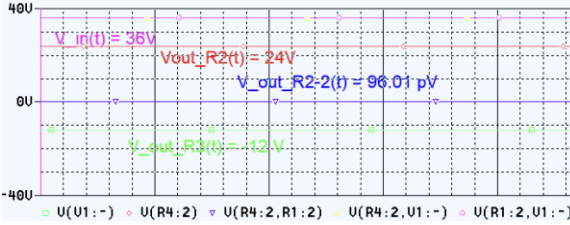


Figure 4.11

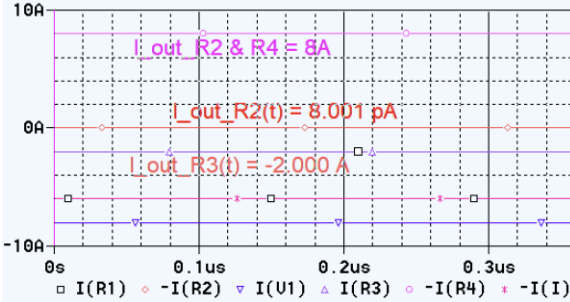


Figure 4.12

Trace Color	Trace Name	Y1
	X Values	0.000
	I(R1)	-6.0000
	-I(R2)	8.0001p
	I(V1)	-8.0000
	I(R3)	-2.0000
	-I(R4)	8.0000
	-I(I)	-6.0000
	CURSOR 1,2	V(V1:-)
		-12.000
		V(R4:2)
		24.000
		V(R4:2,R1:2)
		96.001p
		V(R4:2,V1:-)
		36.000
		V(R1:2,V1:-)
		36.000

Table 4.1

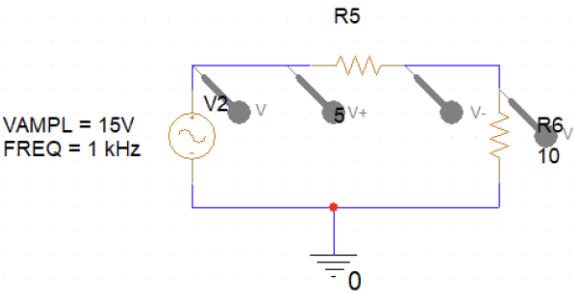


Fig 4.2

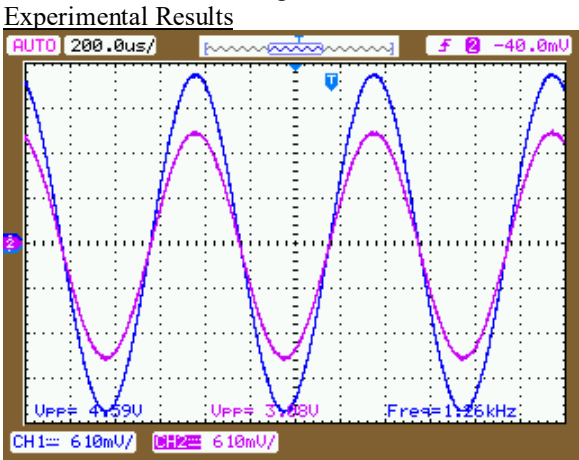
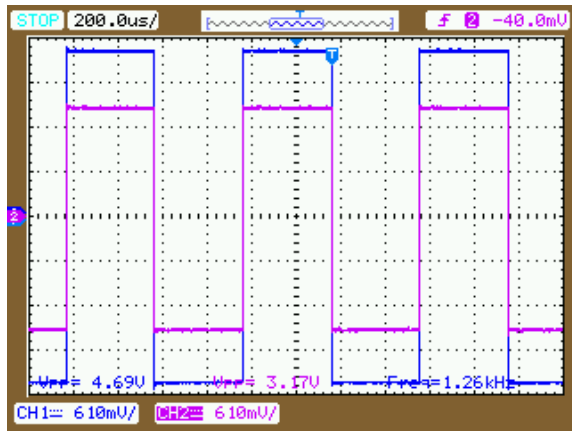


Figure 4.21

	Measured Voltage (V)
Input (1.26kHz)	15
R1	4.59
R2	3.08

Table 4.2



	Measured Voltage (V)
Input (1.26kHz)	15
R1	4.69
R2	3.17

Table 4.3

4.4 DISCUSSION & CONCLUSION

In this lab, we successfully applied PSpice simulations and experimental methods to analyze both DC and AC circuits as outlined in Figures 4.1 and 4.2. For the DC circuit (Fig. 4.1), simulations confirmed theoretical expectations of current flow and voltage drops across individual resistors and at specific node points. This validated the principles of Ohm's Law and Kirchhoff's Laws, reinforcing foundational concepts in DC circuit analysis.

For the AC case (Fig. 4.2), a 1.26 kHz sine wave input was applied both in simulation and through experimental setup. Voltage measurements across R1 and R2, and the overall input voltage, showed strong agreement between simulation and hands-on results, with only minor variances likely due to component tolerances and instrumentation limits. The consistency in data indicates an accurate implementation of AC circuit behavior, especially in resistive networks under sinusoidal excitation.

Overall, this lab reinforced our ability to use simulation software to predict circuit behavior and validated those predictions through physical measurement. It

strengthened our practical skills in voltage and current measurement, while also building confidence in using simulation as a design and verification tool in circuit analysis.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, "ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus," Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

DESIGN EXPERIMENT – CIRCUIT I - IMPLEMENTING MESH ANALYSIS – DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

**Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu**

ABSTRACT:

This lab applied mesh analysis to a multi-loop DC circuit to determine branch currents and component voltages, integrating theoretical calculations, PSPICE simulation, and experimental testing. The circuit from Design Experiment – Circuit I was first analyzed using mesh equations derived from Kirchhoff's Voltage Law (KVL) to obtain predicted values. The same circuit was modeled in PSPICE to simulate operating conditions. A breadboard prototype was then constructed, and voltages were measured using a multimeter. This exercise strengthened core skills in circuit analysis, simulation software application, laboratory measurement, and troubleshooting, fulfilling Student Learning Outcomes related to designing, analyzing, and experimentally testing DC circuits.

KEYWORDS:

Direct Current (DC) Circuits, Alternating Current (AC) Circuits, DC Power Supply, AC Function Generator, Digital Multimeter, Oscilloscopes, Ohm's Law, Frequency, Period, Resistors, DMM, Kirchhoff's Current Law, Kirchhoff's Voltage Law, Branches, Series Circuits, Parallel Circuits, DC circuit analysis, AC circuit analysis, transient response, square-wave input, sine-wave input, node voltage, branch current, and mesh analysis

5.1 INTRODUCTION

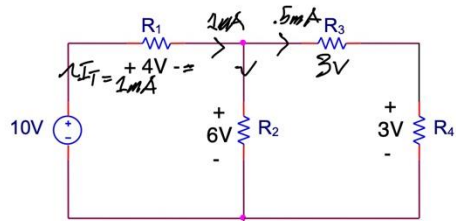
Lab 5, Design Experiment – Circuit I: Implementing Mesh Analysis, focused on applying mesh current methods to analyze a multi-loop DC circuit. Using both PSPICE simulation and physical circuit construction, we determined theoretical voltages and currents, then verified them through measurement. The process involved calculating mesh currents from circuit parameters, setting up the design in PSPICE for simulation, and assembling the circuit on a breadboard for experimental testing. This lab emphasized the integration of analytical methods, simulation tools, and experimental verification to reinforce core electrical engineering principles and laboratory skills.

5.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The circuit for Design Experiment – Circuit I was first analyzed theoretically using mesh analysis, applying Kirchhoff's Voltage Law (KVL) to form and solve simultaneous equations for the loop currents. For the simulation, the circuit was modeled in PSPICE with the exact resistor values and DC voltage sources specified in the lab manual. Voltage and current probes were placed across each resistor and in each loop to verify analytical results through a Bias Point Analysis. For the experimental setup, the circuit was built on a breadboard using precision resistors, a DC power supply, and a digital multimeter (DMM) for voltage measurements. The input voltage, as well as voltage drops across each

resistor, were recorded, ensuring the same configuration as the simulated circuit. All results analytical, simulated, and experimental were compiled in a comparative table to evaluate accuracy and identify any measurement deviations.

5.3 EXPERIMENTAL & SIMULATION DATA & RESULTS



$$R_T = \frac{10}{1mA} = 10k$$

$$R_1 = \frac{4}{1mA} = 4k$$

$$R_2 = \frac{6}{5mA} = 12k$$

$$R_3 = \frac{3}{5mA} = 6k$$

$$R_4 = \frac{3}{5mA} = 6k$$

Figure 5 Resistant value calculations

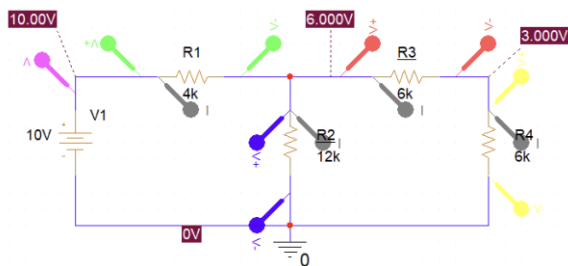


Figure 5.1

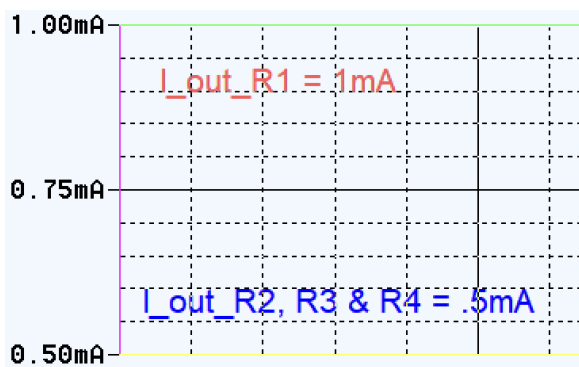


Figure 5.11

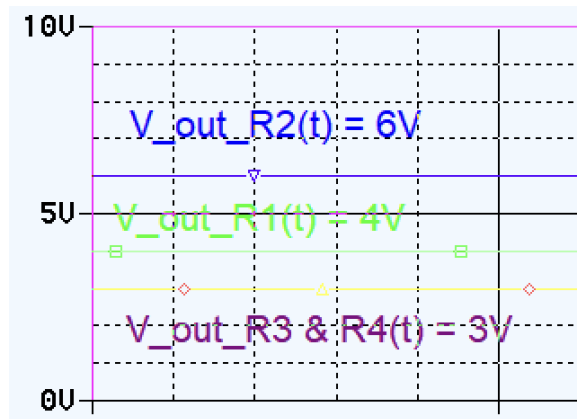


Figure 5.12

Trace Name	Y1
X Values	10.000n
V(V1:+,R1:2)	4.0000
V(R1:2,R2:2)	3.0000
V(R3:2,0)	6.0000
V(R4:2,R4:1)	3.0000
V(V1:+)	10.000
I(R1)	1.0000m
-I(R3)	500.000u
I(R2)	500.000u
-I(R4)	500.000u

Table 5.1

	Resistance (Ω)	Theoretical Voltage (V)	Measured Voltage (V)	% error
R_1	4k	4	3.99	-0.25
R_2	12k	6	6.04	0.67
R_3	6k	3	3.01	0.33
R_4	6k	3	3.01	0.33

Table 5.2

5.4 DISCUSSION & CONCLUSION

The implementation of Mesh Analysis in Design Experiment Circuit I successfully demonstrated the ability to design, analyze, and experimentally test DC circuits in alignment with the stated Student Learning Outcomes. PSPICE simulations accurately predicted node voltages and branch currents, which closely matched the measured experimental values, with all percentage errors remaining under 1%. This high level of agreement validated both the theoretical mesh current calculations and the simulation results.

Through this lab, skills were reinforced in applying laboratory equipment for circuit analysis, troubleshooting circuit builds, and interpreting data from multiple sources. The exercise strengthened proficiency in using PSPICE to model circuits, applying fundamental electrical laws in practice, and professionally documenting technical work.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.

- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.

- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

APPLICATION OF NETWORK THEOREMS (THEVENIN'S, NORTON'S & SUPERPOSITION THEOREMS) CIRCUIT DESIGN, SIMULATION & EXPERIMENTAL TEST AS WELL AS ANALYSIS

Steven Ovanessian and Zachary Ramos
Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

steven.ovanessian.199@my.csun.edu, zachary.ramos.438@my.csun.edu

ABSTRACT:

This lab explored the application of fundamental network theorems Thevenin's, Norton's, Maximum Power Transfer, and Superposition. Using PSpice and laboratory instruments, equivalent circuits were derived and validated by comparing load voltages and currents from both original and reduced networks. The Maximum Power Transfer theorem was confirmed by varying load resistance to produce a parabolic power curve peaking at the theoretical value. Superposition was verified by comparing single-source responses with the combined circuit. Overall, results demonstrated strong agreement between theory, simulation, and experiment, reinforcing the practical importance of circuit simplification, power optimization, and linearity in electrical engineering analysis.

KEYWORDS:

Thevenin's Theorem, Norton's Theorem, Superposition Theorem, Maximum Power Transfer, Load Resistance, PSpice Simulation, Experimental Measurements, Voltage, Current, Power Dissipation

6.1 INTRODUCTION

Thevenin's and Norton's equivalents demonstrate circuit reduction techniques that maintain identical load behavior, while the Maximum Power Transfer theorem emphasizes optimal load selection for efficiency. Superposition illustrates the linearity of electrical systems by analyzing contributions from multiple independent sources. Through PSpice simulations and hands-on measurements, this lab reinforces theoretical concepts by comparing calculated, simulated, and experimental results. By validating these theorems, students gain essential skills in circuit simplification, power optimization, and experimental verification—key foundations for advanced electrical engineering problem solving.

6.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The experiments were performed using both PSpice simulations and bench measurements with DC power supplies, digital multimeters, and oscilloscopes. For Thevenin and Norton equivalents (Fig. 6.4), the original circuit was analyzed, then reduced into its equivalent forms for comparison of load voltage and current. The Maximum Power Transfer setup (Fig. 6.5) involved varying the load resistance around the calculated optimal value and recording voltages, currents, and power to confirm the parabolic relationship. For Superposition (Fig. 6.6), circuits were tested by activating sources individually and jointly, then comparing the measured voltages and currents to verify linearity. All unique currents and voltages were measured, with results checked against KVL, KCL, and Ohm's Law to ensure consistency between theory, simulation, and experimental outcomes.

6.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

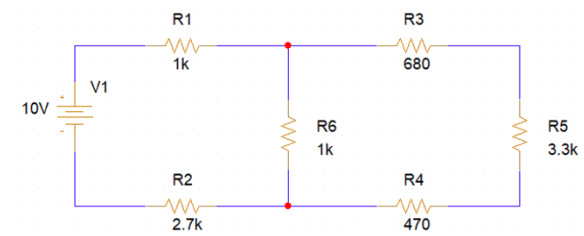


Figure 6.4

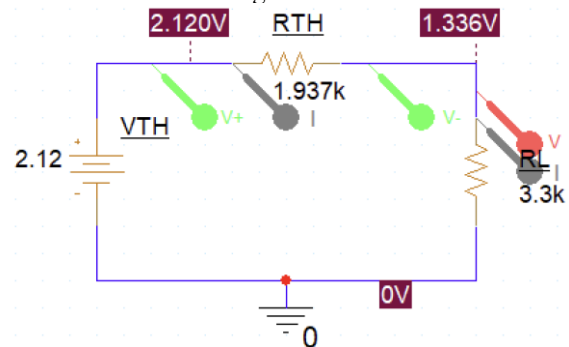


Figure 6.5

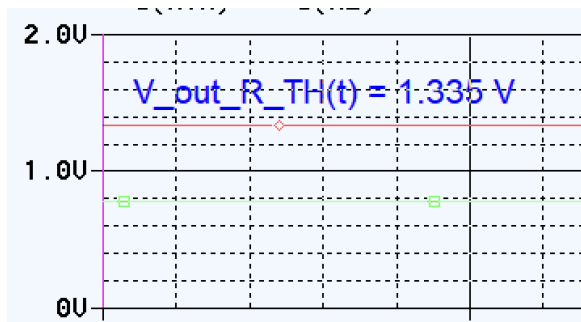


Figure 6.51

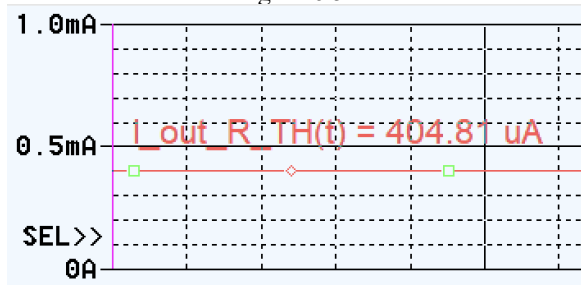


Figure 6.52

Trace Name	Y1
X Values	10.000n
V(RTH:1,RTH:2)	784.121m
V(RTH:2)	1.3359
I(RTH)	404.812u
-I(RL)	404.812u

Table 6.5

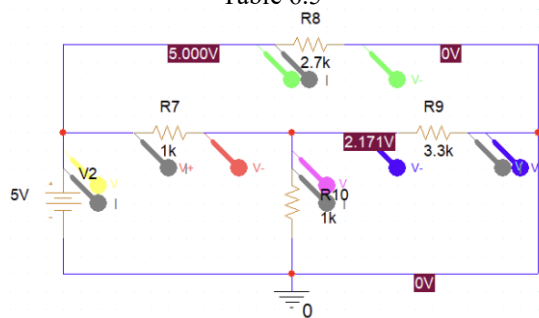


Figure 6.61

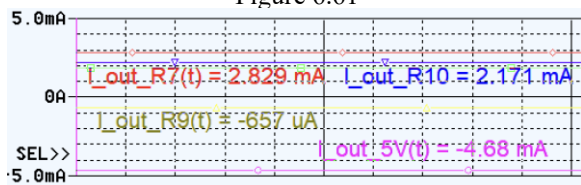


Figure 6.62

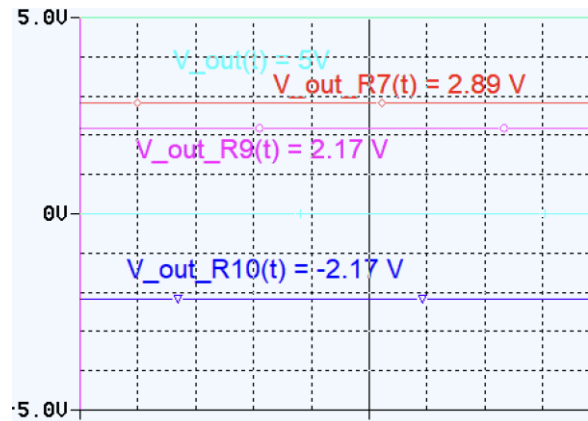


Figure 6.63

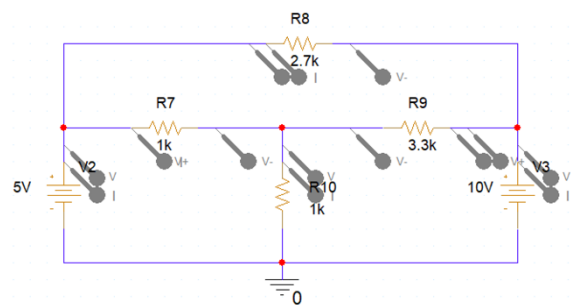


Figure 6.64

Trace Name	Y1
X Values	0.000
V(R8:1,0)	5.0000
V(R7:1,R7:2)	2.8290
V(0,R9:1)	-2.1711
V(R8:1)	5.0000
V(R9:1)	2.1711
V(R9:2)	0.000
I(R8)	1.8519m
I(R7)	2.8290m
-I(R10)	2.1711m
-I(R9)	-657.895u
I(V2)	-4.6808m

Figure 6.65

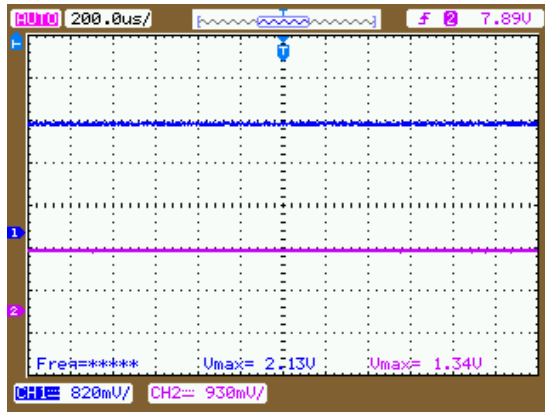


Figure 6.53

Thevenin Measurements	
Voltage	1.34 V
Current	4 uA

Table 6.5

R(Ω)	480	1.1k	1.75k	2.3k	3.3k	4.62k	5.5k
P(w)	2.03	1.39	1.22	4.18	1.58	0.713	0.621

Table 6.7

	10V Short	5V short
R1	1.02 V	6.02 V
R2	0.99 V	-3.98 V

6.4 DISCUSSION & CONCLUSION

In the Thevenin and Norton equivalent circuit analysis (Fig. 6.4), we validated both the simulated and experimental responses by comparing the original circuit's output to its simplified Thevenin and Norton equivalents. The load voltage and current were consistent across both configurations, confirming the correctness of our derived equivalents. The small deviation between theoretical and experimental data is attributed to minor component tolerances and measurement uncertainties.

In the Maximum Power Transfer experiment (Fig. 6.5), we varied the load resistance around the

calculated $R_{max} = R_{Th}$ to observe changes in power dissipation. Both simulation and experimental data produced a parabolic curve, peaking precisely at $R = R_{Th}$, confirming the maximum power transfer theorem. This validated the theoretical prediction and emphasized the importance of impedance matching in circuit design for optimal power.

Finally, the Superposition Theorem (Fig. 6.6) was verified by comparing the summed response of two single-source circuits (5V and 10V individually) with the dual-source circuit. The algebraic summation of voltages V_1 and V_2 matched the direct measurement from the combined-source setup, in both simulation and experimental domains. This reinforced the linearity principle of electrical circuits.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, "ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus," Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

DESIGN EXPERIMENT – CIRCUIT II - IMPLEMENTING MAXIMUM POWER TRANSFER – DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Steven Ovanessian and Zachary Ramos

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
steven.ovanessian.199@my.csun.edu, zachary.ramos.438@my.csun.edu

ABSTRACT:

This lab investigated the Maximum Power Transfer Theorem through design, PSpice simulation, and experimental testing. Resistors R_1 , R_2 , and R_3 were chosen to yield an effective Thevenin resistance near 2.7 k Ω . A range of load resistances above and below this value were tested, producing a parabolic power curve peaking at 2.7 k Ω with a maximum power of approximately 2.315 mW. Results confirmed theoretical predictions and validated KVL, KCL, and Ohm's Law, with close agreement between simulation and experiment.

KEYWORDS:

Maximum Power Transfer, Thevenin Resistance, Load Resistance, Power Dissipation, PSpice Simulation, Experimental Verification, Voltage, Current, Ohm's Law, Parabolic Power Curve

7.1 INTRODUCTION

The purpose of Lab #7 is to implement and verify the Maximum Power Transfer Theorem through circuit design, PSpice simulation, and experimental testing. This theorem states that maximum power is delivered to a load when the load resistance equals the Thevenin resistance of the source circuit. To test this principle, resistors R_1 , R_2 , and R_3 were selected to yield an equivalent source resistance near 2.7 k Ω . A range of load resistors above and below this value were analyzed, with voltages, currents, and power dissipation measured. By comparing simulation and experimental data, the lab confirmed the parabolic relationship of power versus resistance and validated Ohm's Law, KVL, and KCL.

7.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The lab setup consisted of designing a DC circuit with resistors R_1 , R_2 , and R_3 chosen to yield a Thevenin resistance of approximately 2.7 k Ω . Using PSpice, the circuit was simulated to measure input voltage, load voltage, current, and power for varying load resistances. Experimentally, the circuit was constructed on a breadboard, with measurements taken using a DC power supply, digital multimeter, and oscilloscope. Load resistors both below and above 2.7 k Ω were tested, and their corresponding power was calculated. The data were plotted to confirm the parabolic power curve and to validate that maximum power transfer occurred at $R_L = 2.7$ k Ω .

7.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

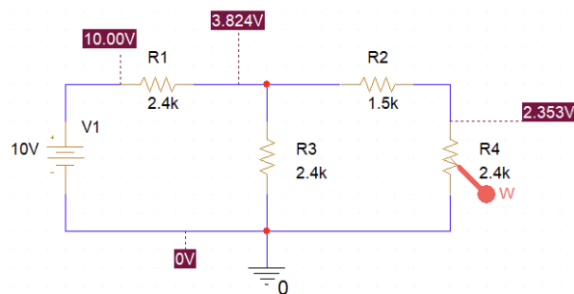


Figure 7.1

Resistance (kOhms)	Power (mW)
900	1.736
1.1	1.904
1.5	2.125
1.9	2.245
2.3	2.300
2.7	2.315

3.1	2.304
3.5	2.276
4.2	2.205
5	2.1

Table 1

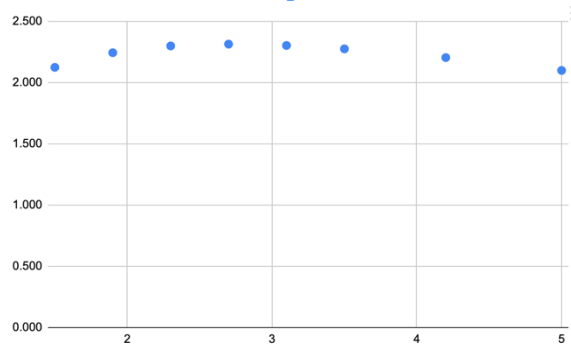


Figure 7.2

7.4 DISCUSSION & CONCLUSION

In this lab, the Maximum Power Transfer Theorem was tested through simulation and experimental verification using a designed DC circuit. The objective was to identify the conditions under which a resistive load receives the maximum amount of power from a source, specifically confirming that maximum power is transferred when the load resistance equals the Thevenin resistance of the source circuit. Resistors R_1 , R_2 and R_3 were selected to construct a source with an effective Thevenin resistance near 2.7 kΩ. A range of load resistors, both below and above this value, were tested. The output power across each resistor was measured and recorded in both simulation and physical experiments. The resulting data formed a smooth,

parabolic power curve, with a clear peak at $R_L = 2.7\text{k}\Omega$, yielding a maximum power of approximately 2.315 mW. This strongly supports the theoretical prediction that optimal power delivery occurs when the load matches the source resistance. The experiment validated Ohm's Law and conservation principles (KVL and KCL), as measured voltages and currents across all elements aligned with theoretical expectations. The PSPICE simulation also confirmed the shape of the power curve and matched closely with the experimental results, confirming the accuracy of both the design and methodology.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, "ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus," Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

OPERATIONAL AMPLIFIERS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

8.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

ABSTRACT:

This lab involved the design, simulation, and experimental testing of operational amplifier circuits using PSpice and hardware implementation. Simulations were first conducted to verify theoretical designs, focusing on parameters such as gain, signal relationships, and configuration effects. The experimental results closely matched the simulations, confirming the accuracy of the design process and providing insight into real-world op-amp performance.

KEYWORDS:

Operational Amplifier, PSpice simulation, Gain, Phase Response, Function Generator, Oscilloscope, Signal Amplification

8.1 INTRODUCTION

Operational amplifiers (op-amps) are fundamental building blocks in analog electronics, widely used for signal amplification, filtering, and mathematical operations such as addition, subtraction, integration, and differentiation. In this lab, we explore the design, simulation, and experimental testing of op-amp circuits to understand their theoretical operation and practical performance. Using PSpice, simulations were conducted to validate circuit behavior and hardware implementation. The experimental phase involved building the circuits on a breadboard, applying test signals, and measuring outputs to compare against theoretical predictions and simulation results.

The op-amp circuits were first simulated in PSpice using ideal components and resistor values specified in the lab. A sinusoidal input was applied, and transient and AC sweep analyses were performed to observe gain, phase, and waveform behavior. For the experimental setup, the same circuits were built on a breadboard using a standard op-amp IC, ± 15 V power supply, and verified resistor values. A function generator provided the input signal, while a digital oscilloscope and multimeter recorded measurements. Experimental results were then compared to simulations and theory, noting close agreement with minor differences due to real-world component limitations.

8.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

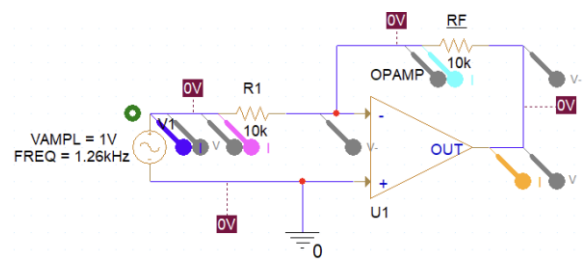
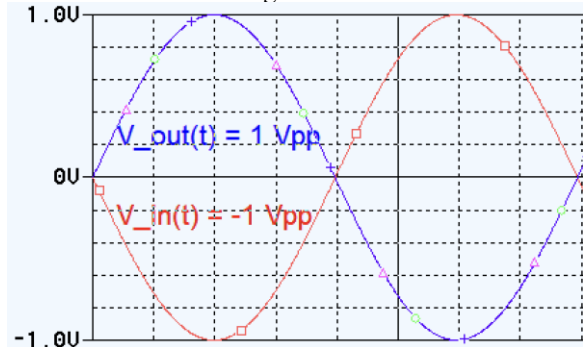
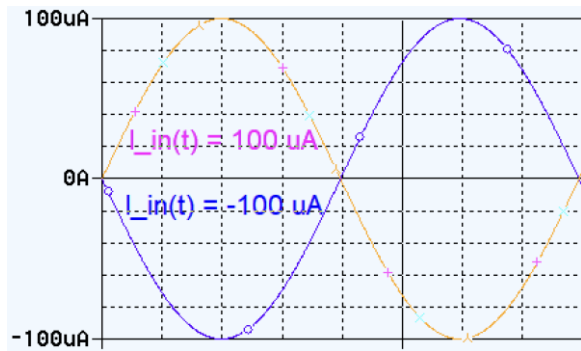
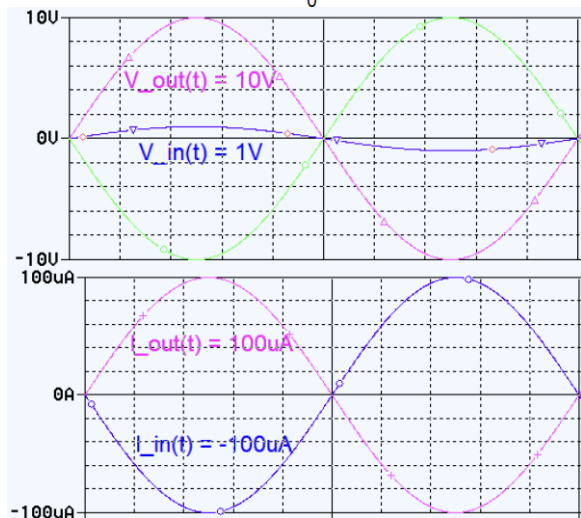
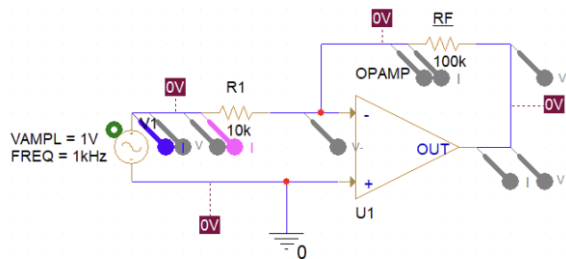


Figure 8.1

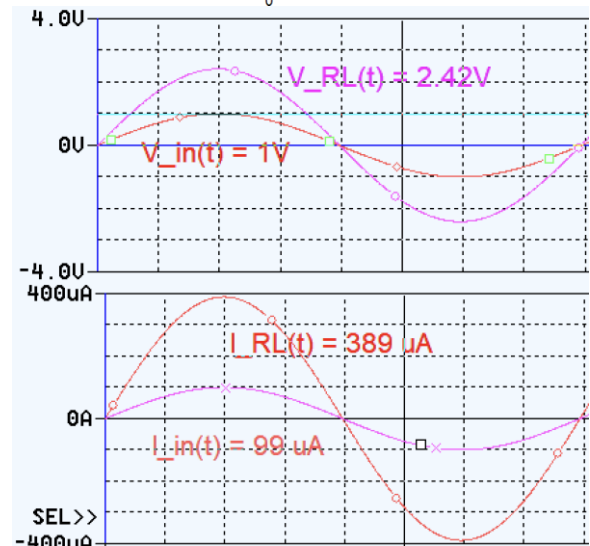
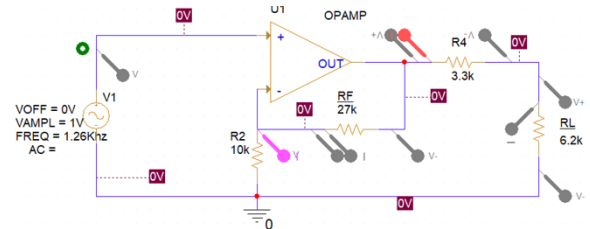




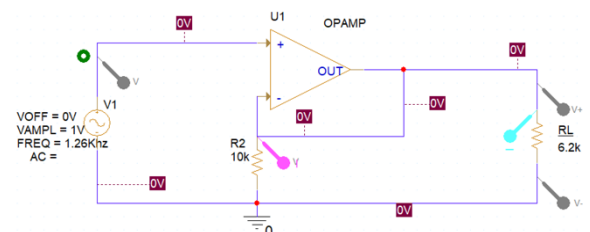
Trace Name	Y1	Y2
X Values	202.285u	0.000
I(V1)	-99.974u	1.3699u
I(R1)	99.974u	7.9168n
I(RF)	99.974u	7.9168n
I(U1)	99.974u	7.9168n
V(U1:OUT)	-0.9997	0.000
V(R1:2,U1:OUT)	0.9997	79.168u
V(R1:1)	0.9997	79.168u
V(R1:1,R1:2)	0.9997	79.168u

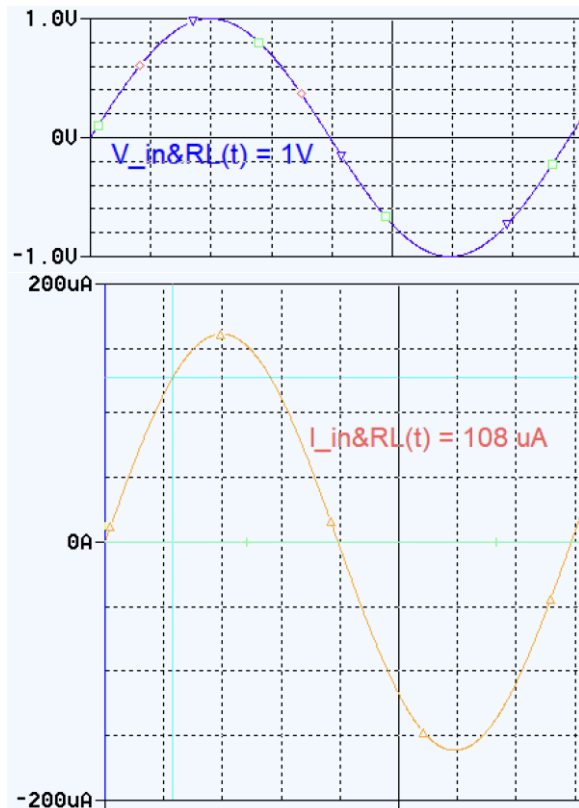


X Values	Y1	Y2
I(V1)	-99.999u	-6.2831n
I(R1)	99.999u	6.2831n
V(R1:1)	1.0000	0.000
V(R1:1,RF:1)	1.0000	62.831u
V(RF:1,U1:OU)	10.000	628.312u
V(U1:OUT)	-10.000	-628.312u



Trace Color	Trace Name	Y1	Y2
	X Values	993.303u	10.000n
	I(R4)	389.454u	30.834n
	-I(R2)	99.995u	7.9168n
CURSOR 1,2	V(R2:2)	1.0000	79.168u
	V(V1:~)	1.0000	79.168u
	V(R4:2,0)	2.4146	191.170u





Trace Name	Y1	Y2
X Values	198.285u	0.000
V(R2:2)	1.0000	0.000
V(V1:+))	1.0000	79.168u
-I(R2)	100.000u	7.9168n
-I(RL)	161.290u	12.769n
V(U1:-,0)	1.0000	79.168u
I(V1)	0.000	0.000

8.4 DISCUSSION & CONCLUSION

In this lab, we successfully designed, simulated, and experimentally tested operational amplifier circuits using PSpice and hardware implementation. The

results from both simulation and experimental measurements closely matched the theoretical calculations, confirming the accuracy of our design process and the validity of the underlying op-amp principles. The process of moving from theory to simulation to hardware provided a clear understanding of how op-amp circuits perform under real conditions. Through testing and observation, the lab reinforced important concepts such as gain, input and output signal relationships, and the impact of circuit configuration on performance. Overall, the experiment provided valuable insight into both the strengths and practical limitations of operational amplifier designs.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

FIRST ORDER CIRCUITS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This laboratory focused on the design, simulation, and experimental validation of first-order circuits using both resistive-capacitive (RC) and resistive-inductive (RL) configurations. Using PSpice CAD tools, theoretical predictions for transient and steady-state responses were developed and compared against measured experimental data obtained from physical circuit builds. Key parameters, including time constants, natural and forced responses, and frequency-dependent behaviors, were analyzed to verify theoretical behavior.

KEYWORDS:

First-order circuits, RC circuit, RL circuit, Time Constant, Transient response, Steady-state response, Power analysis

9.1 INTRODUCTION

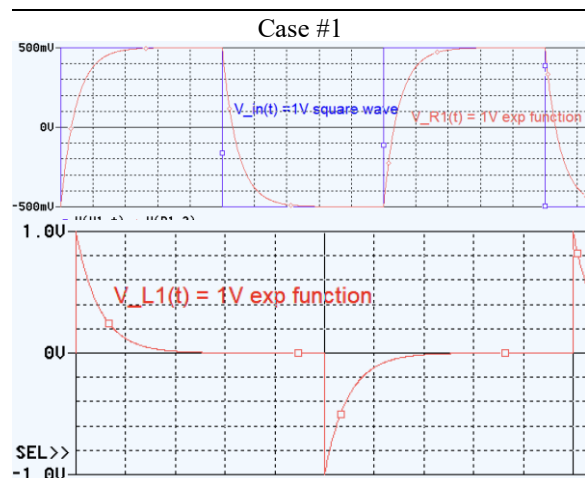
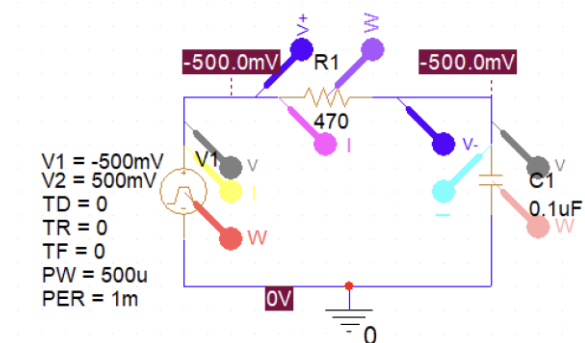
This lab focuses on the design, simulation, and experimental analysis of first-order RC and RL circuits to study their transient and steady-state responses. Using PSpice, each circuit is modeled with specified component values and driven by a 1 V square wave, with pulse width and period chosen based on the circuit's time constant to capture the full charging and discharging cycles. Measurements of voltages, currents, and power are obtained from both simulations and physical experiments, then compared to theoretical predictions.

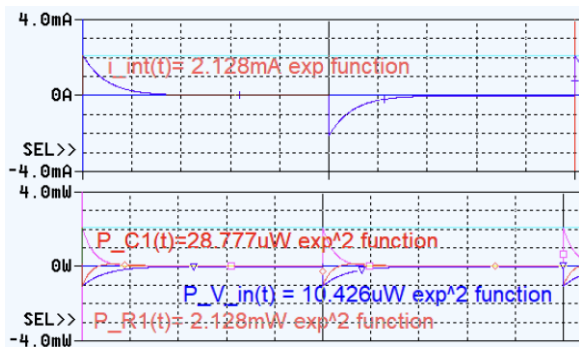
9.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The lab setup utilized a DC power supply, function generator, digital oscilloscope, and standard resistors, capacitors, and inductors as specified for

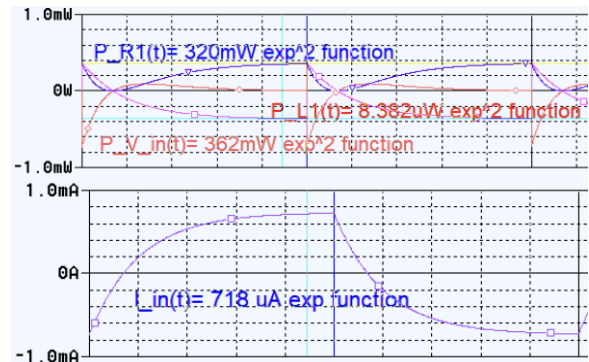
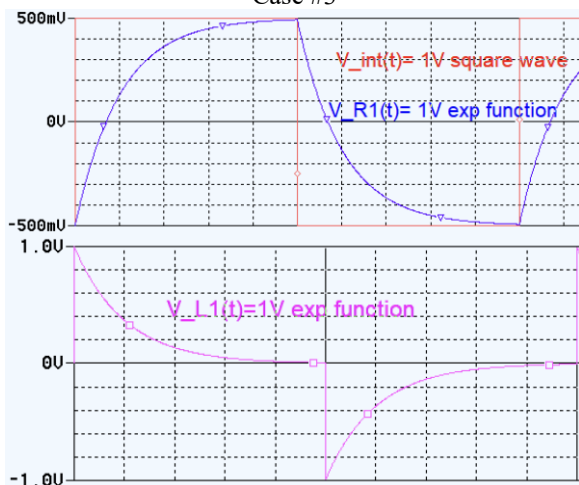
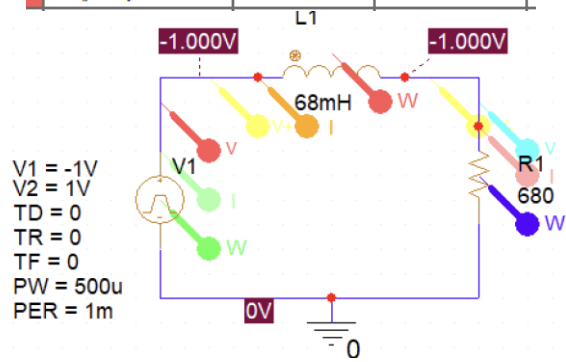
each case. For each RC and RL configuration, components were assembled on a breadboard according to the lab manual diagrams. Input square waves were generated with carefully selected pulse widths and periods based on calculated time constants. PSpice simulations were created to mirror the experimental circuits, enabling direct comparison of voltage, current, and power waveforms. Measurements were taken using oscilloscope channels to overlap input and output signals, ensuring accurate observation of transient and steady-state behaviors.

9.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

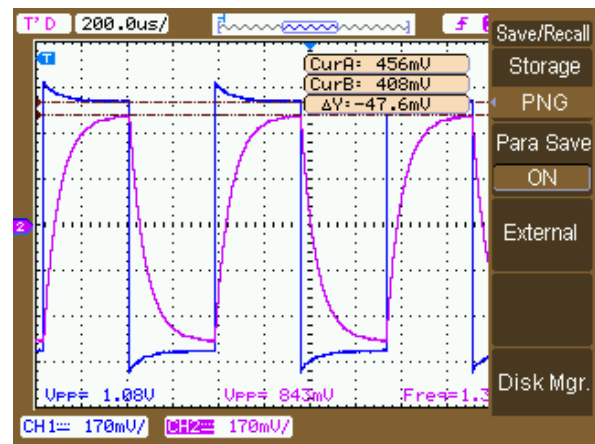
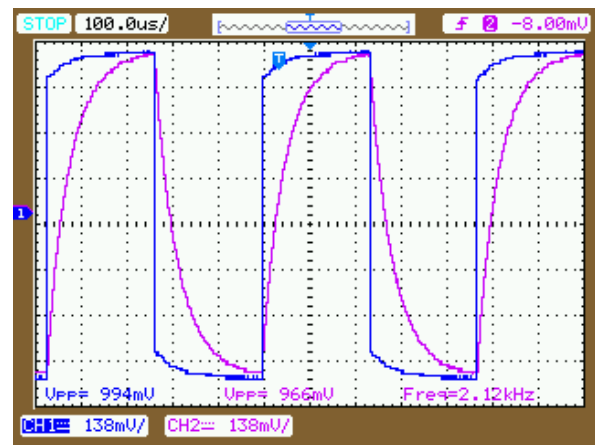




Trace Name	Y1	Y2
X Values	1.5000m	0.000
V(C1:2)	499.976m	-500.000m
V(V1:~)	499.994m	-498.433m
V(V1:~,R1:2)	18.013u	3.1348m
I(V1)	-38.324n	3.1348m
I(R1)	38.324n	21.277u
-I(C1)	38.324n	21.277u
W(R1)	690.319f	212.766n
W(C1)	19.161n	3.1348m
W(V1)	-19.162n	10.426u



X Values	445.060u	500.001u
V(L1:1,R1:2)	11.671m	6.7317m
V(V1:~)	500.000m	499.994m
V(R1:2)	488.329m	493.262m
-I(R1)	718.131u	725.386u
I(V1)	-718.131u	-725.386u
W(V1)	-359.065u	-362.689u
W(L1)	8.3815u	4.8831u
W(R1)	350.684u	357.806u
I(L1)	718.131u	725.386u



9.4 DISCUSSION & CONCLUSION

In this lab, the design, simulation, and experimental testing of first-order RC and RL circuits were successfully completed using PSpice and bench measurements. The preliminary calculations provided accurate predictions for time constants and helped in selecting appropriate pulse widths and frequencies to observe the expected transient and steady-state behaviors. For both RC and RL cases, the simulated and experimental waveforms closely matched, confirming the theoretical models and validating the accuracy of PSpice as a design tool. Minor deviations between simulation and experimental data were attributed to component tolerances and measurement limitations. The lab reinforced the relationship between circuit parameters and transient response, demonstrated the correct procedure for measuring voltage, current, and power in time-domain analysis, and strengthened practical skills in both simulation and hardware testing of first-order systems.

This lab addressed several Student Learning Outcomes (SLOs), including the ability to design, analyze, and experimentally test first-order circuits, apply laboratory instrumentation effectively, and compare theoretical, simulated, and experimental results. It also met key Course Learning Objectives (CLOs) such as learning to determine the complete

response of first-order systems to various input waveforms, applying CAD tools like PSpice for design and verification, and building confidence in troubleshooting and measurement techniques for fundamental electrical circuits.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

DESIGN EXPERIMENT – CIRCUIT III - IMPLEMENTING RC CIRCUITS, SIMULATION & EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab implements and validates a first-order RC circuit that reproduces the target input–output behavior. Using the 5 V square-wave excitation, we designed the time constant $\tau = RC$ to match the measured rise and fall trajectories of $V_C(t)$ and the complementary $V_R(t)$ across one full period. Component values (R, C) were selected analytically from the required exponential fit and then verified through PSpice simulation.

KEYWORDS:

First-order circuit, RC circuit, Time constant, Transient response, power dissipation

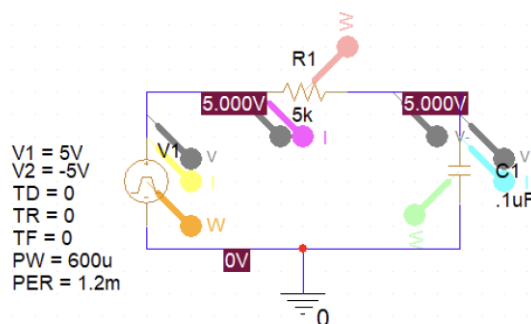
10.1 INTRODUCTION

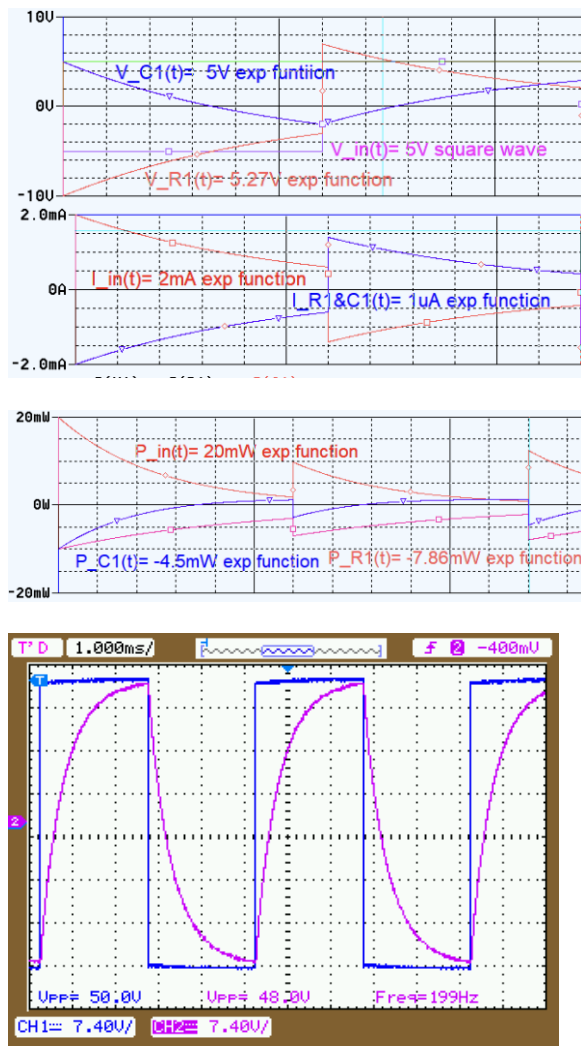
This Lab focuses on the design, simulation, and experimental verification of a first-order RC circuit, emphasizing both theoretical analysis and practical implementation. Using the configuration specified in Figure 10.3 of the lab manual, resistor and capacitor values were selected that reproduce the target waveform shown in Figure 10.4. This exercise integrates concepts from transient circuit analysis, time constant calculation, and waveform shaping, while reinforcing skills in PSpice simulation, oscilloscope measurements, and power analysis. The lab's objective is to demonstrate how appropriate component selection influences circuit response to a square-wave input, thereby bridging the gap between analytical predictions and experimental results.

10.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The required time constant τ is determined from the target waveform, and unique resistor R and capacitor C values are chosen such that $\tau = R \times C$, with only the resistor value shared between lab partners. The design is first implemented in PSpice with a 5 V square-wave input, generating three subplots: voltages across the input, capacitor, and resistor; loop current; and input power, resistor power dissipation, and capacitor stored power. An additional plot overlays the input and capacitor voltages. Experimentally, the circuit is built on a breadboard and tested using a function generator and two-channel oscilloscope to capture the same measurements, overlaying input with capacitor and resistor voltages separately.

10.3 EXPERIMENTAL & SIMULATION DATA & RESULTS





x10 magnification

10.4 DISCUSSION & CONCLUSION

In this lab, a first-order RC circuit was designed, simulated, and experimentally tested to meet the waveform requirements of the specifications. By selecting appropriate resistor and capacitor values, the circuit's time constant was tailored to closely match the reference waveform. PSpice simulations provided clear insight into the expected transient behavior, with voltage, current, and power plots

confirming theoretical predictions. The experimental results validated the simulation, showing close agreement in rise and fall times, peak values, and overall waveform shape. Minor discrepancies were attributed to component tolerances and oscilloscope measurement limitations. This exercise reinforced the relationship between time constant selection and transient response, strengthened skills in both simulation and hands-on circuit testing, and demonstrated how theoretical design choices translate into practical performance. The activity directly supported the Student Learning Outcomes (SLO) by developing the ability to design, analyze, and test first-order circuits, and addressed the Course Learning Objectives (CLO) by integrating analytical methods, simulation tools, and experimental techniques to meet specified design criteria.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

SECOND ORDER CIRCUITS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab explored the behavior of second-order RLC circuits in series and parallel configurations through theory, PSpice simulation, and experimental testing. The transient responses, including oscillations and damping, were analyzed under square-wave excitation and compared with preliminary calculations. Results showed close agreement between theory, simulation, and experiment, with small discrepancies due to component tolerances.

KEYWORDS:

Second-order circuits, RLC circuit, Series RLC, Parallel RLC, Resonance, Damping, Transient response

11.1 INTRODUCTION

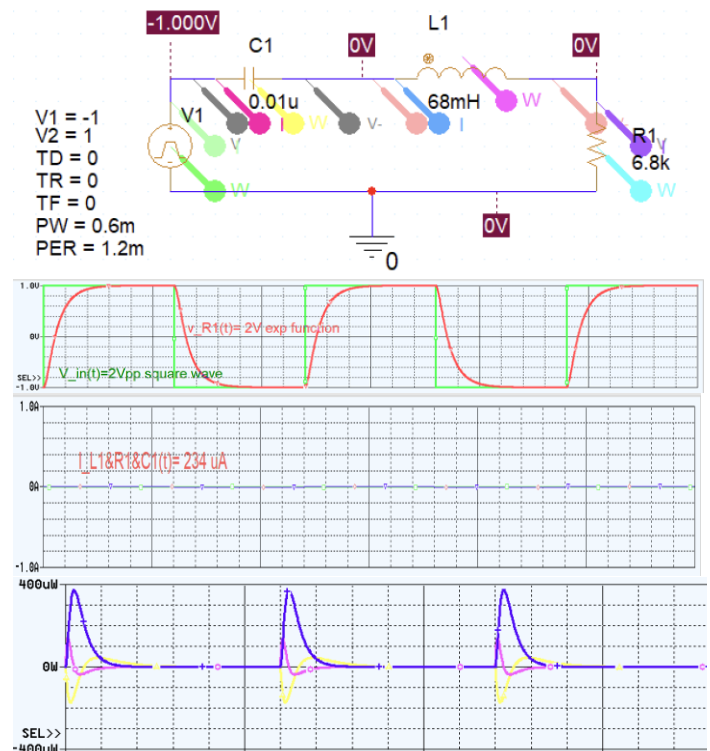
Second-order circuits are fundamental in electrical engineering because they exhibit unique transient and steady-state behaviors such as oscillations, resonance, and damping. In this lab, both series and parallel RLC circuits were designed and analyzed using theoretical calculations, PSpice simulations, and experimental testing. By applying a square-wave input and varying component values, the lab demonstrates how resistors, inductors, and capacitors interact to store, dissipate, and exchange energy. The objective is to compare theoretical predictions with simulated and measured results, reinforcing the principles of natural frequency, damping ratio, and circuit response in second-order systems.

11.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

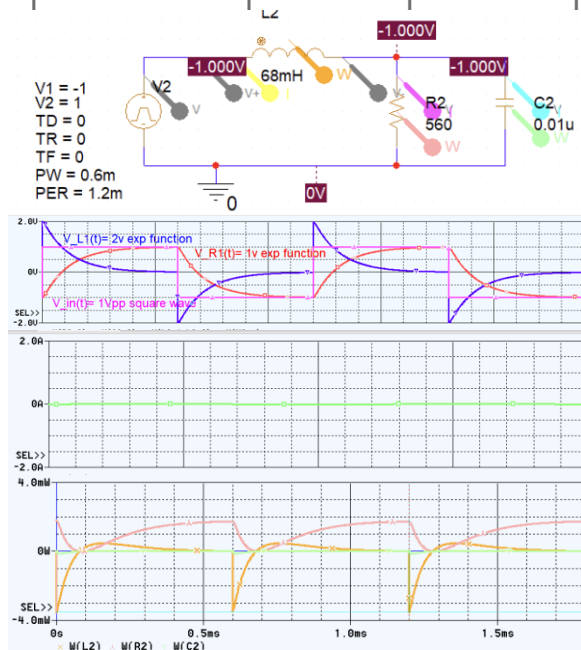
For the simulations, the circuits were built in PSpice with specified R, L, and C values, and a square-wave input was applied with carefully selected pulse width and period to capture transient behavior. Voltage across each component, circuit current, and power

were measured using separate plots for comparison. For the experimental portion, the circuits were constructed on a breadboard and tested with a function generator and oscilloscope. Input and output voltages were recorded by overlapping waveforms across the resistor (for the series circuit) and across the capacitor (for the parallel circuit), both over multiple cycles and a single cycle, to validate the theoretical and simulated results.

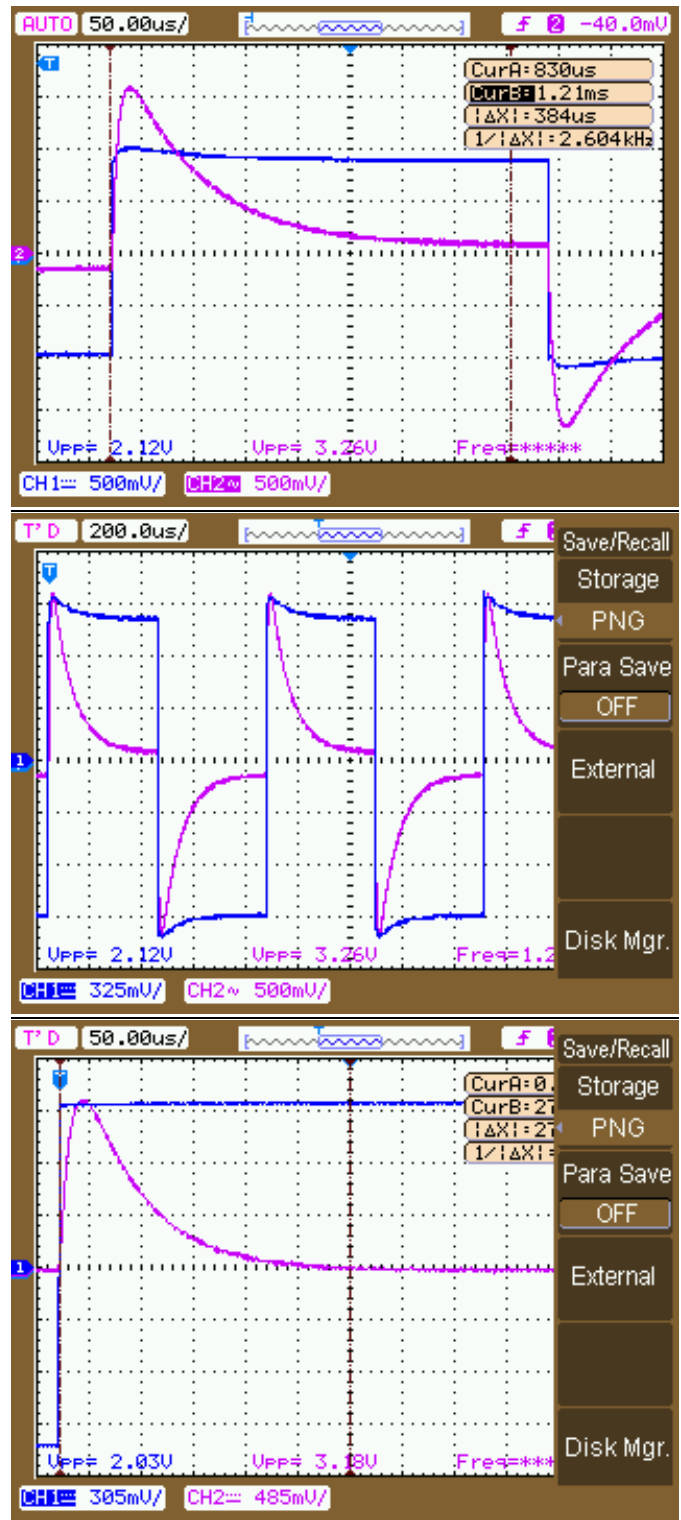
11.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

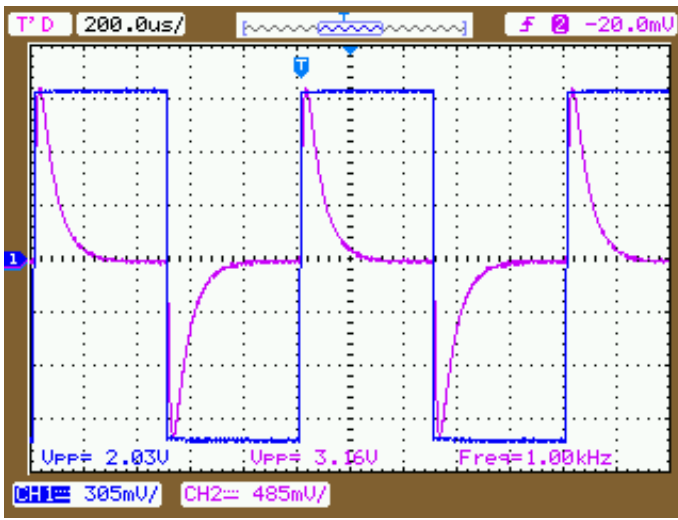


X Values	288.417u	0.000
W(V1)	-2.6126u	2.8863p
W(C1)	2.5745u	-2.9559p
W(L1)	-8.3156n	58.823f
W(R1)	46.416n	58.82E-21
V(C1:2,L1:2)	-3.1829m	0.000
I(V1)	-2.6126u	-2.9452p
I(C1)	2.6126u	2.9559p
I(L1)	2.6126u	2.9412p
-I(R1)	2.6126u	2.9412p
V(R1:2)	17.766m	20.000n
V(V1:~)	1.0000	-1.0000
V(V1:~,C1:2)	985.417m	-1.0000



X Values	600.043u	0.000
I(L2)	1.7657m	-1.7857m
-I(R2)	1.7647m	-1.7857m
-I(C2)	1.0184u	2.8422p
W(L2)	20.777u	65.574u
W(R2)	1.7439m	1.7857m
W(C2)	1.0064u	65.574u
V(R2:2)	988.294m	-1.0000
V(C2:2)	988.233m	-1.0000
V(L2:1,L2:2)	11.767m	20.000m
V(V2:~)	1.0000	-1.0164





11.4 DISCUSSION & CONCLUSION

This lab successfully demonstrated the design, simulation, and experimental analysis of second-order RLC circuits in both series and parallel configurations. Through preliminary calculations, PSpice simulations, and oscilloscope measurements, we confirmed the theoretical predictions of resonance, transient response, and damping behavior. Although minor discrepancies were observed due to non-ideal components and measurement limitations, the overall agreement across theory, simulation, and experiment reinforced the accuracy of the design. This exercise addressed key Student Learning Outcomes (SLOs), including the ability to design and analyze second-order circuits, apply computer-aided tools such as PSpice, and validate theoretical models through hands-on

experimentation. Furthermore, the lab enhanced problem-solving and technical communication skills by requiring careful selection of input signal parameters, proper organization of simulation results, and comparison with real-world measurements. Overall, this experiment strengthened the understanding of energy exchange and dissipation in RLC systems while reinforcing the importance of integrating theory, simulation, and laboratory practice in circuit design.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

IMPEDANCE AND ADMITTANCE CIRCUITS DESIGN, SIMULATION & EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab investigated impedance and admittance circuits using a series RLC configuration to study the effects of frequency and capacitance on circuit behavior. Preliminary impedance and phasor calculations were performed and compared with PSpice simulations and experimental oscilloscope measurements. The results showed that varying the frequency and capacitor value significantly altered the impedance, phase angle, and voltage division across the circuit. Simulation and experimental data closely matched theoretical predictions, with only minor discrepancies due to component tolerances and measurement limitations. Overall, the lab reinforced the importance of phasor analysis, frequency response, and the role of reactive elements in AC circuit design while demonstrating the value of integrating theory, simulation, and experimental validation.

KEYWORDS:

Impedance, Admittance, RLC circuit, Series circuit, Phasor analysis, Frequency response, Phase angle, Voltage division

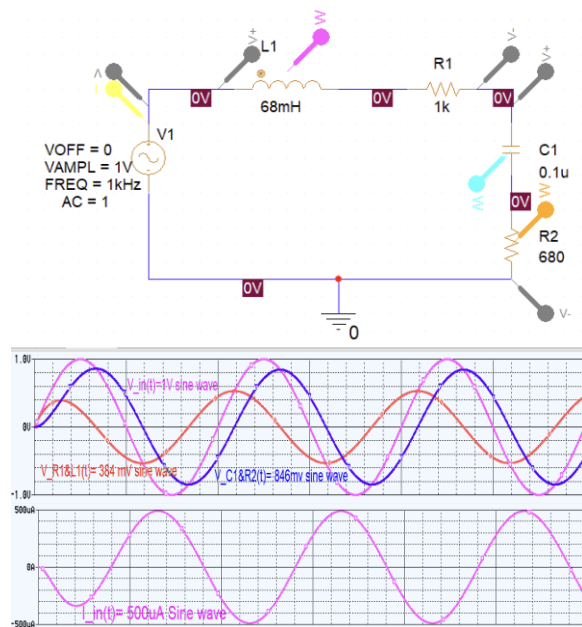
12.1 INTRODUCTION

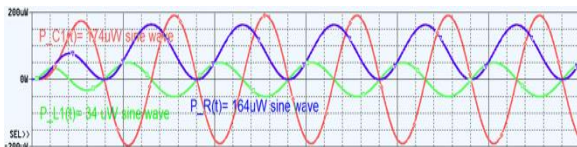
Impedance and admittance are fundamental concepts in AC circuit analysis, as they describe how resistors, capacitors, and inductors interact with sinusoidal signals. In this lab, a series RLC circuit was analyzed through theoretical impedance and phasor calculations, PSpice simulations, and experimental measurements. By varying frequency and capacitance, the lab explored how these parameters affect voltage division, current, and phase relationships within the circuit. The objective was to verify theoretical predictions and reinforce the use of phasor analysis in understanding AC circuit behavior.

12.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The lab involved analyzing a series RLC circuit through theoretical calculations, PSpice simulations, and experimental testing. Preliminary impedance and phasor calculations were performed for the assigned cases to predict voltage and phase relationships. In PSpice, the circuit was simulated with a sinusoidal input, and measurements included input and output voltages, circuit current, and power across each component. Experimental testing was conducted on a breadboard using a function generator and oscilloscope, where input and output voltages were overlapped across the resistor-inductor branch and resistor-capacitor branch to measure time delays and calculate phase angles. Results from simulation and experiment were then compared to theoretical predictions.

12.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

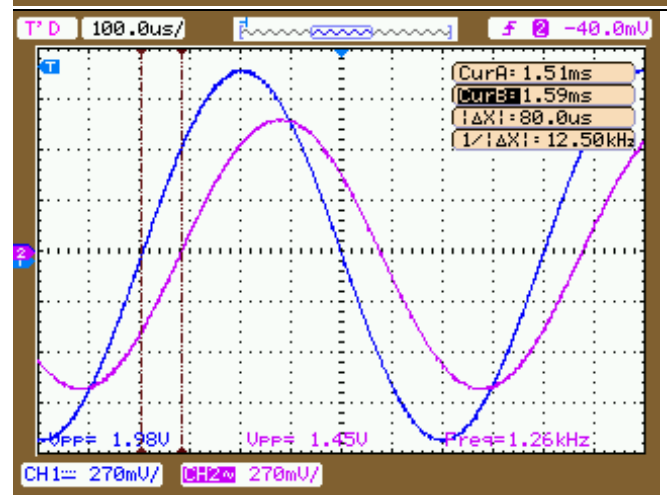
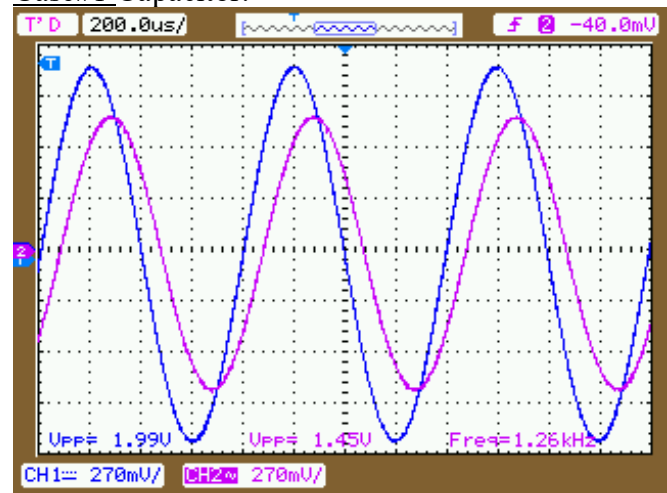
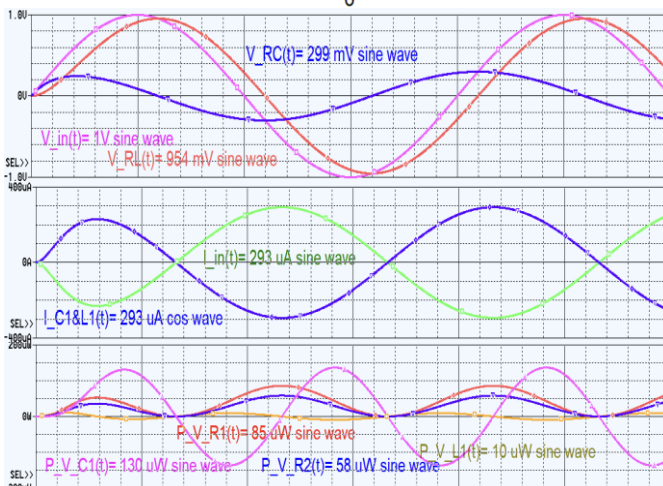
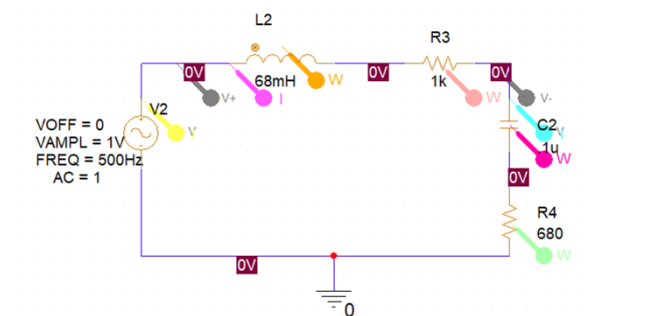


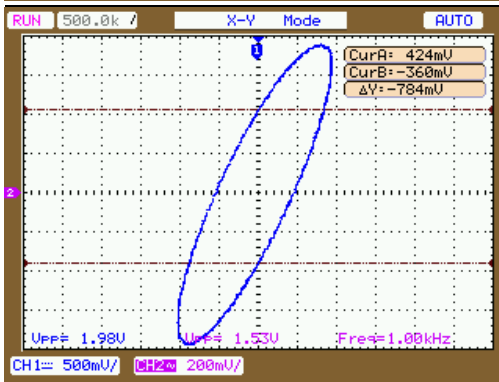
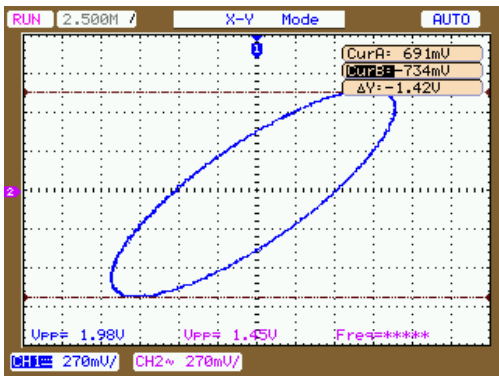


X Values	0.000	0.000
V(V1:~)	15.708u	15.708u
V(V1:~,R1:2)	15.708u	15.708u
V(R1:2,R2:2)	392.689p	392.689p
I(V1)	-577.463f	-577.463f
W(L1)	0.000	0.000
W(C1)	8.337E-27	8.337E-27
W(R2)	226.8E-24	226.8E-24
V(V2:~,R3:2)	7.8537m	7.8537m
V(R3:2,V2:~)	-7.8537m	-7.8537m
I(V2)	-288.729p	-288.729p
W(L2)	2.2675p	2.2675p
W(R3)	83.36E-18	83.36E-18
W(R4)	56.69E-18	56.69E-18
W(C2)	208.4E-24	208.4E-24

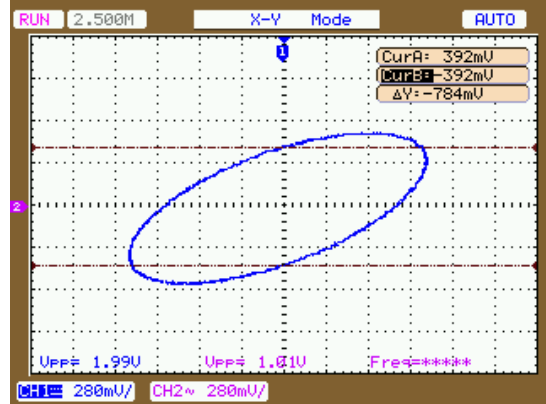
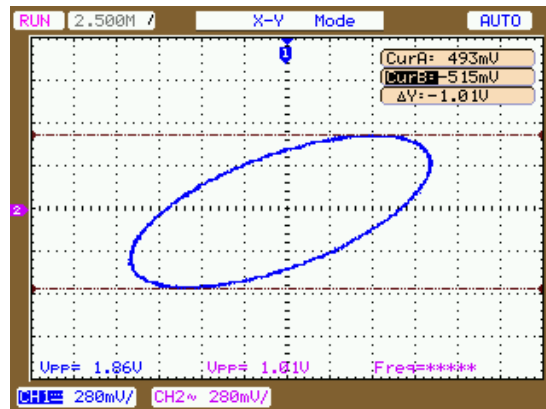
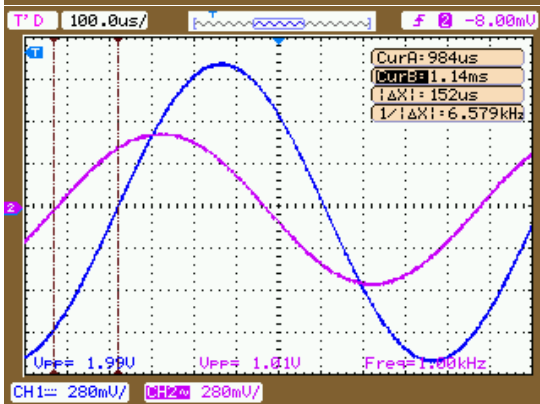
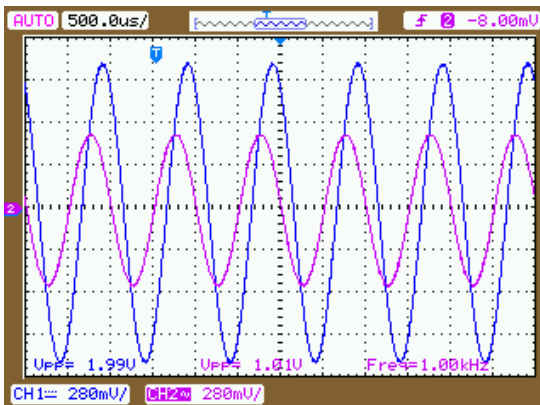
X Values	0.000	0.000
V(V2:~)	31.416u	31.416u
V(C2:2)	3.1413n	3.1413n
V(V2:~,R3:2)	31.413u	31.413u
I(V2)	-4.6189p	-4.6189p
I(L2)	4.6189p	4.6189p
-I(C2)	4.6189p	4.6189p
W(L2)	0.000	0.000
W(R3)	21.33E-21	21.33E-21
W(R4)	14.51E-21	14.51E-21
W(C2)	2.133E-24	2.133E-24

Case#1 Capacitor:

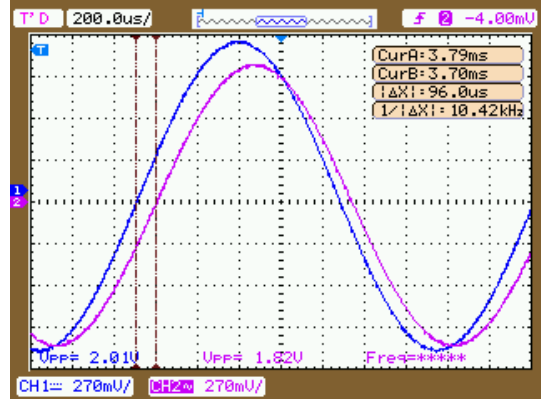
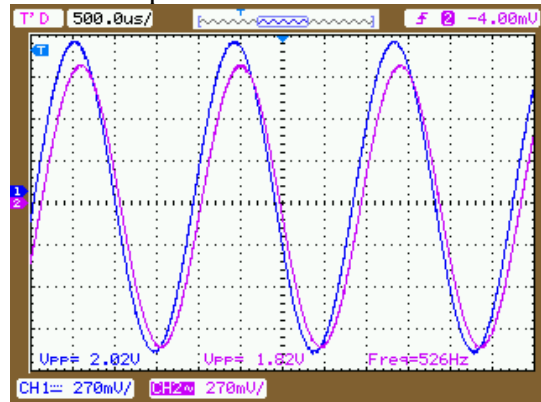


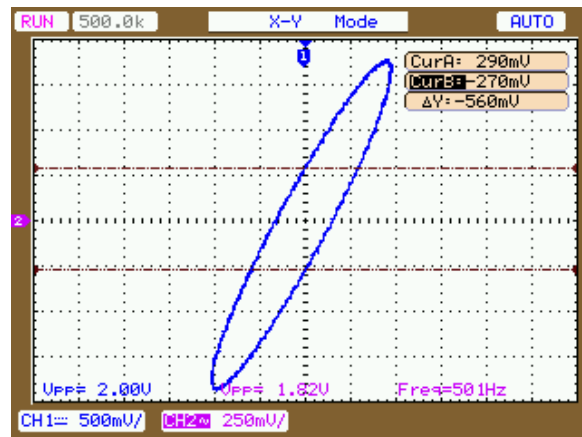
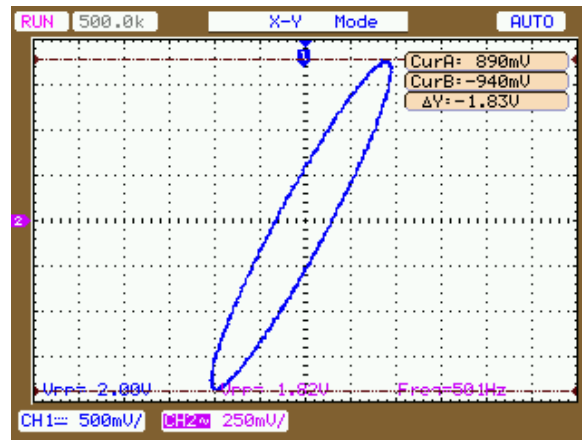
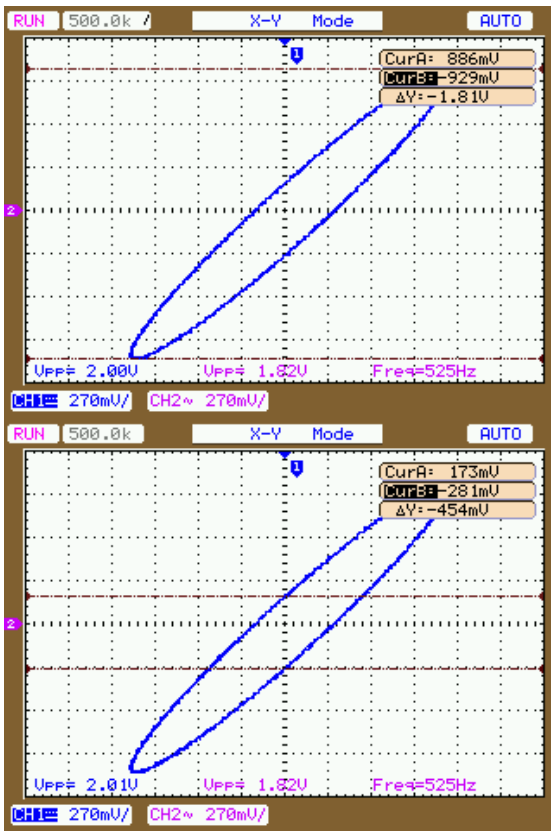


Inductor:

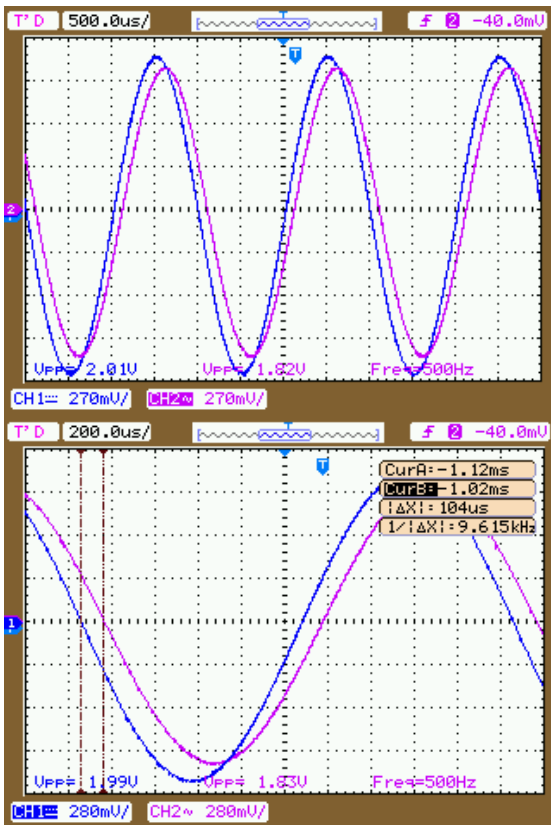


Case#3 Capacitor





Inductor:



12.4 DISCUSSION & CONCLUSION

In this lab, impedance and admittance circuits were analyzed through preliminary calculations, PSpice simulations, and experimental testing for the assigned cases. The results confirmed that circuit behavior strongly depends on frequency and capacitance, with changes in these parameters directly affecting the impedance, phase shift, and voltage distribution across the series RLC branches. The simulations provided clear verification of theoretical predictions, while the experimental measurements showed close agreement despite small discrepancies caused by component tolerances and oscilloscope resolution. Both $v_1(t)$ and $v_2(t)$ waveforms revealed measurable time delays relative to the input, allowing calculation of phase angles that matched the phasor analysis. Overall, this experiment reinforced the importance of impedance and admittance concepts in AC circuit design, demonstrated how reactive elements influence current and voltage relationships, and achieved key Student Learning Outcomes (SLOs) by connecting theory, simulation, and hands on measurements.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

FREQUENCY RESPONSE AC CIRCUITS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab investigated the frequency response of RC and RL circuits through theoretical analysis, PSPICE simulation, and experimental testing. Transfer functions and cutoff frequencies were derived to classify the circuits as low-pass or high-pass filters. Simulations and measurements confirmed the expected amplitude attenuation and phase shift around the cutoff frequency, with minor discrepancies due to component tolerances. The results demonstrated how frequency response analysis verifies filter behavior and highlighted PSPICE as a useful tool for predicting experimental outcomes.

KEYWORDS:

13.1 INTRODUCTION

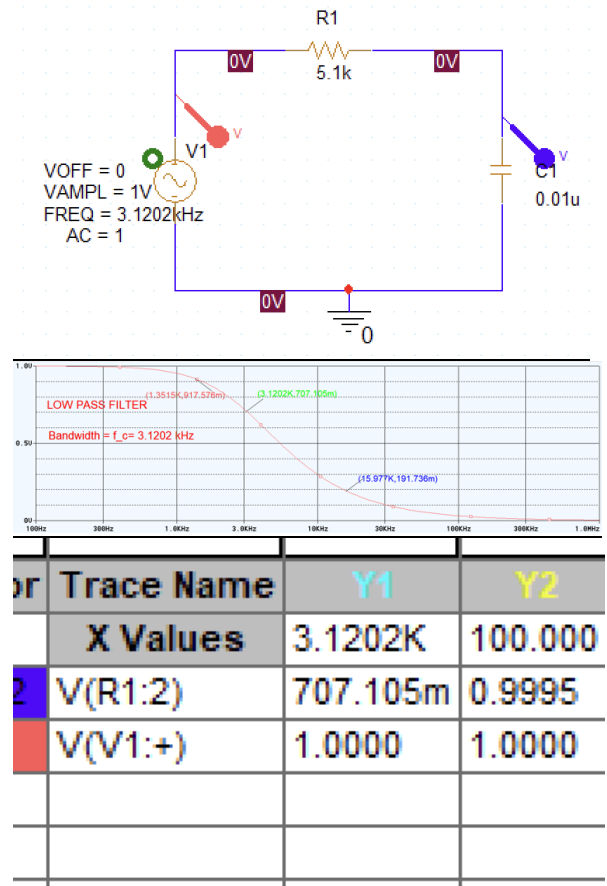
In this lab, the frequency response of RC and RL circuits was studied using theoretical analysis, PSPICE simulations, and experimental testing. Transfer functions were derived to determine cutoff frequencies and identify whether the circuits acted as low-pass or high-pass filters. PSPICE was then used to perform AC sweep and time-domain simulations, which were compared with experimental measurements obtained from the oscilloscope and function generator. By analyzing input and output signals at frequencies below, at, and above the cutoff frequency, the effects of attenuation and phase shift were observed. This lab provided practical verification of frequency response theory and demonstrated the filtering applications of RC and RL circuits.

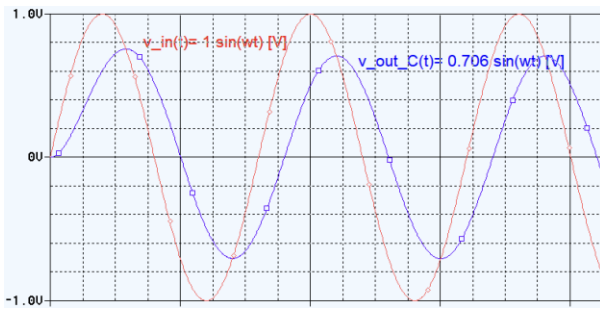
13.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

In the lab setup we used a function generator, oscilloscope, and discrete RC and RL components. Circuits were designed according to the assigned

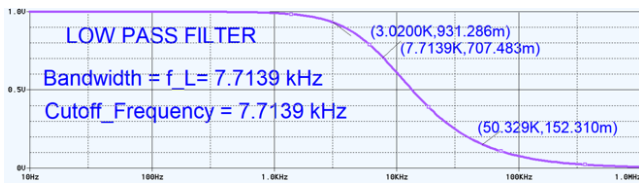
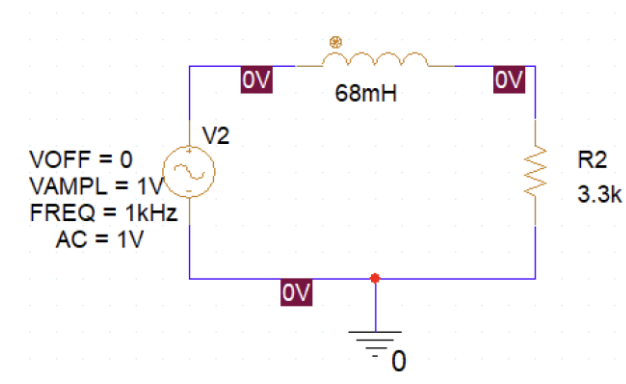
cases and built on a breadboard. PSPICE was first used to simulate each circuit with a 1 V sinusoidal input. An AC sweep (10 Hz–1 MHz, 40 points/decade) was performed to obtain the frequency response, and time-domain simulations were run at frequencies below, at, and above the cutoff frequency. Experimentally, Channel 0 of the oscilloscope measured the input voltage while Channel 1 measured the output. Input and output signals were overlapped at the three frequency cases, and peak output voltages were recorded across a range of frequencies. Data was tabulated and plotted in Excel to compare theoretical, simulated, and experimental results.

13.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

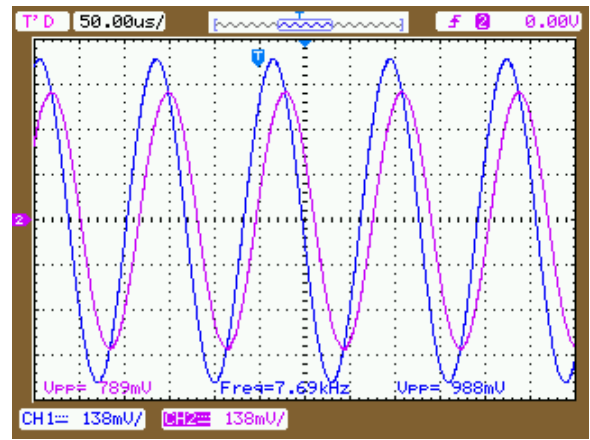
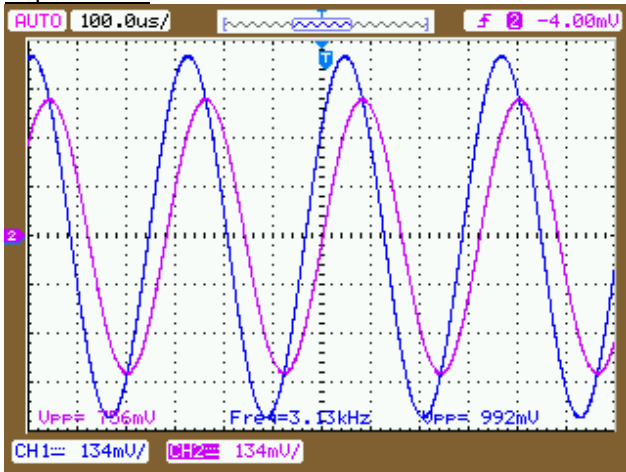




Trace Name	Y1	Y2
X Values	116.222u	0.000
V(R1:2)	756.218m	0.000
V(V1:~)	759.857m	196.048u

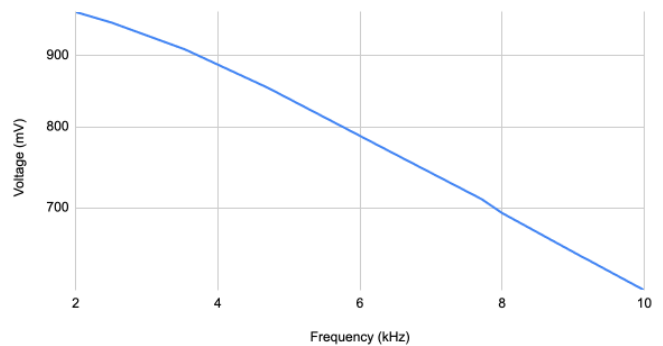


Experimental



Frequency (kHz)	Voltage (mV)
2	967
2.51	950
3.54	909
4.71	853
6.51	764
7.714	710
8	694
9.14	645
9.17	644
9.44	633
10	611

Voltage (mV) vs. Frequency (kHz)



13.4 DISCUSSION & CONCLUSION

In this lab, we analyzed, simulated, and experimentally tested the frequency response of RC and RL circuits to better understand their behavior as filters. Theoretical calculations of the transfer functions and cutoff frequencies provided the foundation for PSPICE simulations, which

confirmed the expected amplitude attenuation and phase shifts across frequency. Experimental results further validated these findings, showing good agreement with theory despite small deviations due to component tolerances and measurement limitations. Overall, the lab demonstrated how RC and RL circuits can be designed as low-pass and high-pass filters, emphasized the role of cutoff frequency in determining bandwidth, and highlighted the usefulness of PSPICE as a predictive tool prior to experimental verification.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

PASSIVE FILTERS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian
Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge
zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab focused on the design, simulation, and experimental testing of passive RC and RLC filters to investigate their frequency response. Transfer functions were derived to calculate the cutoff frequencies, center frequency, and bandwidth of the circuits. PSPICE simulations using AC sweep and time-domain analysis showed the expected band-pass behavior, with maximum output at the center frequency and attenuation outside the passband. Experimental measurements using a function generator and oscilloscope confirmed these results, with minor deviations attributed to component tolerances and practical limitations. The lab demonstrated how passive filters shape signal response across frequency and highlighted the usefulness of simulation in predicting real-world circuit performance.

KEYWORDS:

Passive filters, Band-pass filter, Cutoff frequency, Center frequency, Bandwidth, Transfer function, Frequency response, RC circuit, RLC circuit, PSPICE simulation, Oscilloscope measurement, Signal attenuation

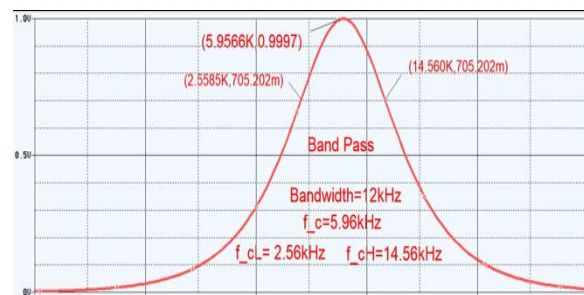
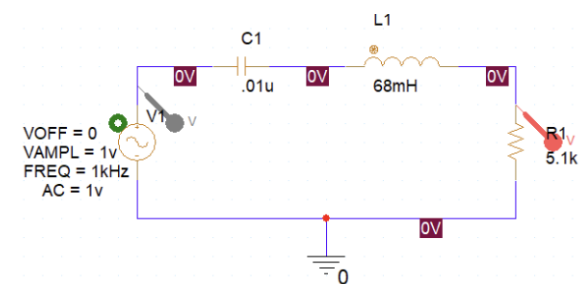
14.1 INTRODUCTION

In this lab, passive RC and RLC filters were analyzed through theoretical calculations, PSPICE simulations, and experimental testing to study their frequency response. Transfer functions were derived to determine the lower cutoff, upper cutoff, center frequency, and bandwidth of each circuit. PSPICE was then used to obtain frequency-domain and time-domain responses, which were compared with oscilloscope measurements from the experimental setup. By observing input and output signals across a range of frequencies, the filtering characteristics of the circuits were confirmed, demonstrating how passive filters can selectively pass or attenuate signals based on frequency.

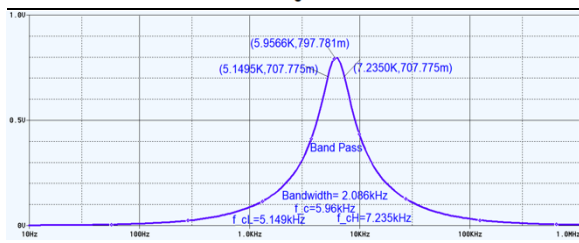
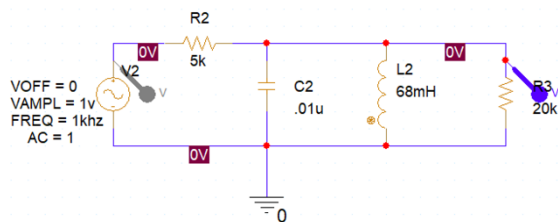
14.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

The circuits were built on a breadboard using the assigned resistor, inductor, and capacitor values for each case. PSPICE was first used to simulate the circuits with a 1 V sinusoidal input. An AC sweep from 10 Hz to 1 MHz (40 points/decade) was performed to obtain the frequency response, and time-domain simulations were run at frequencies below, at, and above the center frequency. Experimentally, the function generator provided the input signal, while Channel 1 of the oscilloscope measured the input and Channel 2 measured the output. Input and output signals were overlapped at the lower cutoff, center, and upper cutoff frequencies as well as at additional points across the frequency range. Peak output voltages were recorded, tabulated, and plotted in Excel for comparison with theoretical and simulated results.

14.3 EXPERIMENTAL & SIMULATION DATA & RESULTS

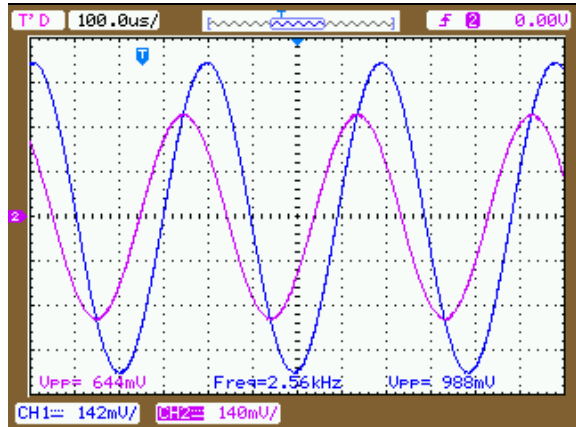
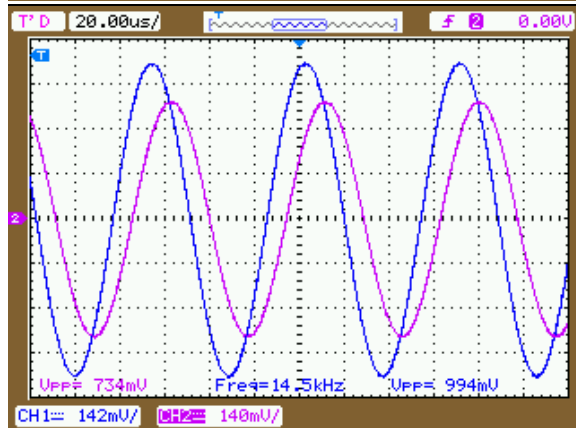
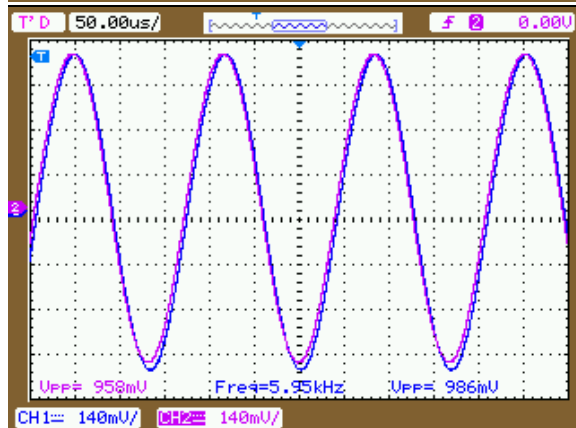
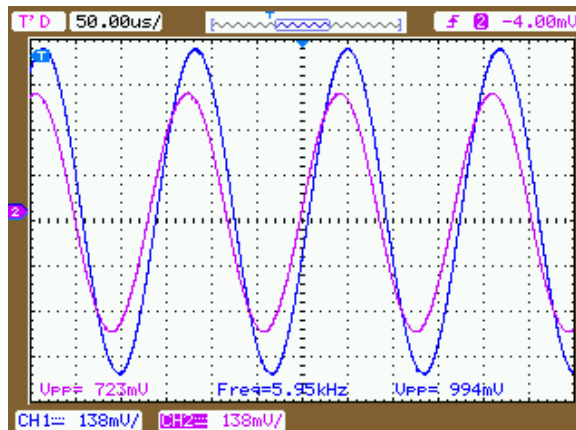
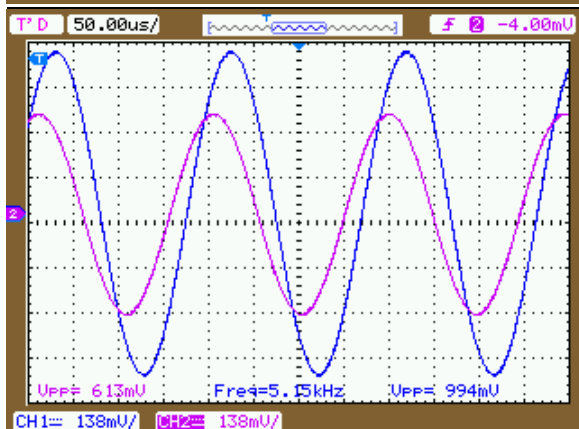
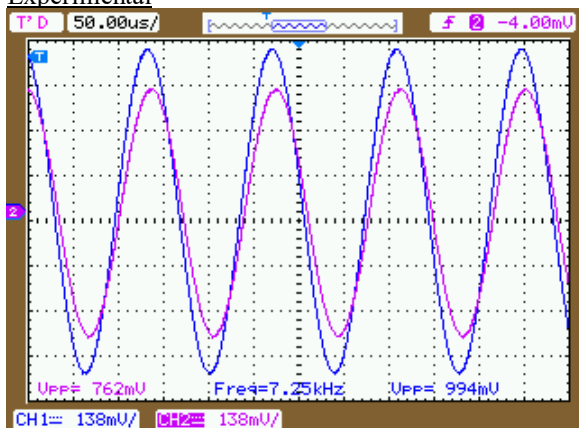


X Values	6.0509K	10.000
V(C1:1)	1.0000	1.0000
V(R1:2)	0.9996	3.2044m

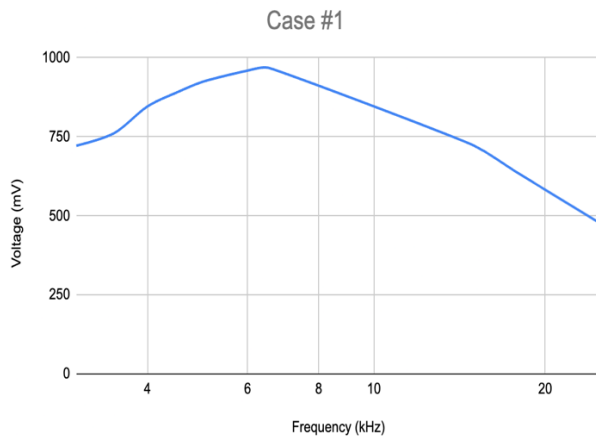


X Values	6.2048K	10.000
V(V2:~)	1.0000	1.0000
V(R3:2)	796.427m	854.515u

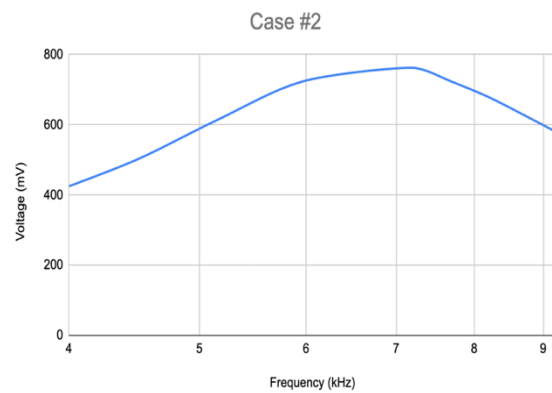
Experimental



Case #1	
Frequency (kHz)	Voltage (mV)
3	722
3.5	762
4	846
4.5	890
5	924
5.95	958
6.5	969
15	722
18	633
25	476



Case #2	
Frequency (kHz)	Voltage(mV)
4	425
4.5	502
5.1495	613
5.9556	723
7.235	762
7.7	723
8.2	679
8.7	629
9.2	580



14.4 DISCUSSION & CONCLUSION

In this lab, the design, simulation, and experimental testing of passive RC and RLC filters were successfully completed to analyze their frequency response characteristics. Theoretical calculations of transfer functions, cutoff frequencies, and center frequencies provided the foundation for predicting circuit behavior. PSPICE simulations confirmed the expected band-pass behavior, showing attenuation of signals outside the passband and maximum response at the center frequency. Experimental measurements using the oscilloscope and function generator closely matched the simulated results, with small deviations explained by component tolerances and practical limitations. Overall, this lab reinforced the concept of passive filter operation, demonstrated the relationship between cutoff frequencies, bandwidth, and center frequency, and highlighted the usefulness of PSPICE as a design and verification tool before physical implementation.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory

Syllabus,” Summer 2025, Department of
Electrical and Computer Engineering,
California State University, Northridge.

DIODES AND TRANSISTORS CIRCUITS DESIGN, SIMULATION AND EXPERIMENTAL TEST AS WELL AS ANALYSIS

Zachary Ramos and Steven Ovanessian

Mechanical Engineering and Computer Engineering Dept.,
California State University, Northridge

zachary.ramos.438@my.csun.edu, steven.ovanessian.199@my.csun.edu

ABSTRACT:

This lab examined the operation of diodes and transistors through theoretical analysis, PSPICE simulation, and experimental testing. LED and silicon diodes were characterized to observe rectification and nonlinear V-I behavior, while transistor circuits were studied for their amplification properties. Simulations provided expected input-output responses, which were then confirmed with oscilloscope measurements using sinusoidal and ramp inputs at varying frequencies. Experimental results showed close agreement with theory, with minor deviations due to component tolerances.

KEYWORDS:

Diode, LED (Light Emitting Diode), Transistor, Rectification, Amplification, Frequency response, Gain, Oscilloscope

15-16.1 INTRODUCTION

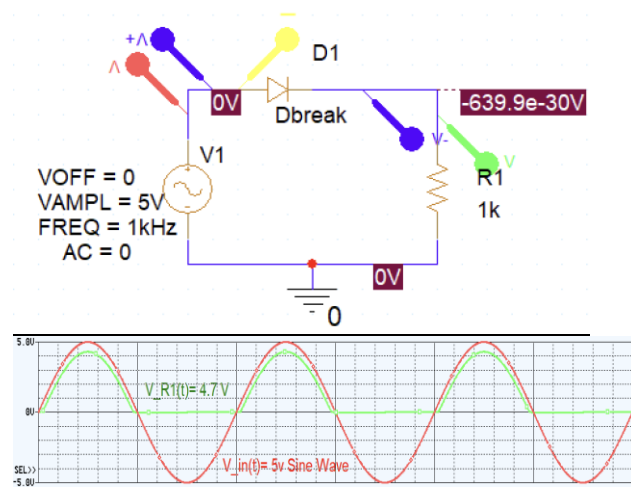
In this lab, the electrical behavior of diodes and transistors was investigated through theoretical analysis, PSPICE simulation, and experimental testing. Diodes are nonlinear semiconductor devices that permit current flow primarily in one direction, and their V-I characteristics make them useful for rectification and signal shaping. Light-emitting diodes (LEDs) were studied to compare forward voltage drops and response to alternating signals. Transistors were then examined as active devices capable of signal amplification, highlighting their role in moving beyond passive circuit elements. PSPICE simulations were used to model input-output waveforms, V-I characteristics, and frequency response under different input conditions. These results were compared with experimental measurements obtained using a function generator and oscilloscope. By analyzing rectification, clamping, and amplification behavior, this lab reinforced the fundamental operation of semiconductor devices and demonstrated their importance in both analog circuit applications and as

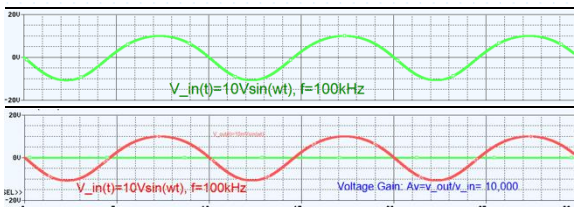
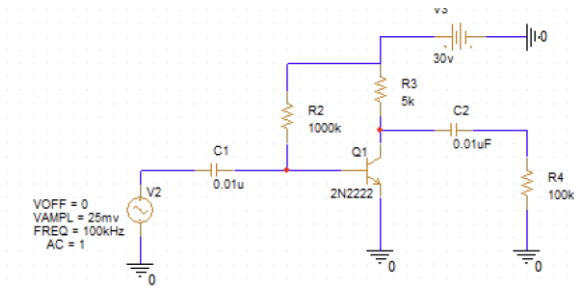
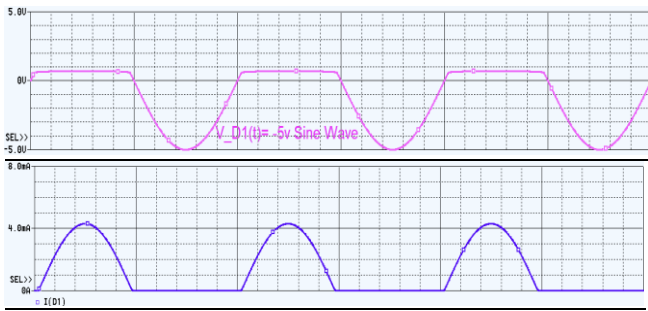
building blocks for more complex electronic systems.

15-16.2 EXPERIMENTAL & SIMULATION SETUPS & PROCEDURES

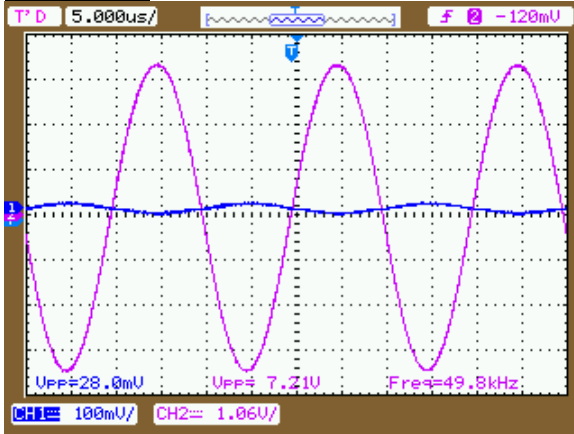
The circuits were built using LEDs, standard diodes, and transistor components according to the assigned cases. PSPICE simulations were first performed with sine and ramp inputs to observe the V-I characteristics, rectification, and amplification behavior. AC sweep and time-domain simulations were used to evaluate circuit response at multiple frequencies and input voltages. Experimentally, a function generator provided the input signals, while the oscilloscope measured both input and output waveforms to compare with simulation results. Different resistor values and capacitor options were tested to analyze their effect on gain and frequency response. Peak-to-peak voltages were recorded, and voltage gain was calculated to validate theoretical and simulated predictions.

15-16.3 EXPERIMENTAL & SIMULATION DATA & RESULTS





Experimental



	Voltage
Vin	28 mV
Vout	7.21 V
Gain (Vin/Vout)	257.5

15-16.4 DISCUSSION & CONCLUSION

In this lab, diode and transistor circuits were designed, simulated, and experimentally tested to investigate their voltage current behavior and amplification characteristics. Theoretical calculations and PSPICE simulations provided expected responses for both LED and standard

diodes, as well as transistor-based circuits. Experimental testing confirmed the non-linear behavior of the diodes, the rectification effect, and the amplification capabilities of transistor circuits. The measured gain values showed close agreement with theoretical predictions, though small deviations were observed due to component tolerances and measurement limitations. Overall, this lab reinforced the fundamental principles of semiconductor device operation, demonstrated the transition from passive to active circuit behavior, and highlighted the effectiveness of PSPICE as a tool for predicting and validating experimental results.

REFERENCES

- [1] S. Roosta, ECE240L Electrical Engineering Fundamentals Laboratory Manual, Version 2.0, California State University, Northridge, 2024.
- [2] B. F. Mallard, ECE 240L – Electrical Engineering Fundamentals Laboratory Manual, Revised 01/03/2014, California State University, Northridge.
- [3] S. Daniel-Berhe, “ECE 240L Electrical Engineering Fundamentals Laboratory Syllabus,” Summer 2025, Department of Electrical and Computer Engineering, California State University, Northridge.

