Summer Fellowship Report

On

Integrated Circuit Design using Subcircuit feature of eSim

Submitted by

Anjali Thakur

Under the guidance of

Prof.Kannan M. Moudgalya

Chemical Engineering Department

IIT Bombay

August 31, 2024

Acknowledgment

We take this occasion to offer our heartfelt gratitude to the FOSSEE, IIT Bombay Team for offering us this wonderful opportunity to work on the design and integration of multiple sub-circuits in eSim. Working on eSim has prov ided us with invaluable insights into various open-source EDA tools for circuit simulation and their applications in the practical world.

We extend our sincere regards to Prof. Kannan M. Moudgalya for his valuable guidance and motivation to throughout this fellowship program.

We would like to express our heartfelt appreciation to the entire FOSSEE team including our mentors Mr. Sumanto Kar, Mrs. Vineeta Ghavri, and Mrs. Usha Vishwanathan for constantly guiding and mentoring us throughout the duration of our internship.

It is with their support that we have been able to fulfill our project demands successfully. Whenever faced with an issue, our mentors were always accessible to help us assess and debug them. Our learnings from them have been invaluable and shall be of paramount importance to us in the future.

Overall, it was a delightful experience interning at FOSSEE and contributing to its growth and I take away some great insights and knowledge from it. As enthusiastic beginners in the semiconductor industry, this internship is a milestone for us in our pursuit of a successful career.

Contents

Chapter 1

Introduction

FOSSEE which stands for Free/Libre and Open Source Software for Education is an organization, based at IIT Bombay, as a remarkable initiative aimed at promoting the use of open-source software in education and research. It was established with the mission to reduce the dependency on proprietary software and to encourage the adoption of open-source alternatives. FOSSEE offers a wide range of tools and resources that cater to various academic and professional needs.

It provides comprehensive documentation, tutorials, workshops, and hands-on training sessions, for empowering students, educators, and professionals to leverage open-source software for their projects and coursework. The organization's commitment to fostering a collaborative and inclusive environment has significantly contributed to the democratization of technology and has opened up new avenues for innovation and learning.

1.1 eSim

eSim, created by the FOSSEE project at IIT Bombay, is a versatile open-source software tool for circuit design and simulation. It combines various open-source software packages into one cohesive platform, making it easier to design, simulate, and analyze electronic circuits. This tool is particularly useful for students, educators, and professionals who need an affordable and accessible alternative to proprietary software.

eSim offers features for schematic creation, circuit simulation, PCB design, and includes an extensive library of components. The Subcircuit feature is a significant enhancement, enabling users to design complex circuits by integrating simpler subcircuits. Through eSim, FOSSEE promotes the use of open-source solutions in engineering education and professional fields, encouraging innovation and collaboration.

1.2 NgSpice

NgSpice is the open-source spice simulator for electric and electronic circuits. Such a circuit may comprise JFETs, bipolar and MOS transistors, passive elements like R, L, or C, diodes, transmission lines and other devices, all interconnected in a netlist.

Digital circuits are simulated as well, event-driven and fast, from single gates to complex circuits and the combination of both analog and digital as well as a mixed-signal circuits. NgSpice offers a wealth of device models for active, passive, analog, and digital elements. Model parameters are provided by our collections, by the semiconductor device manufacturers, or from semiconductor foundries. The user adds her circuits as a netlist, and the output is one or more graphs of currents, voltages and other electrical quantities or is saved in a data file.

1.3 Makerchip

Makerchip is a platform that offers convenient and accessible access to various tools for digital circuit design. It provides both browser-based and desktop-based environments for coding, compiling, simulating, and debugging Verilog designs. Makerchip supports a combination of open-source tools and proprietary ones, ensuring a comprehensive range of capabilities.

One can simulate Verilog/SystemVerilog/Transaction-Level Verilog code in Makerchip. eSim is interfaced with Makerchip using a Python based application called Makerchip-App which launches the Makerchip IDE. Makerchip aims to make circuit design easy and enjoyable for users of all skill levels. The platform provides a user-friendly interface, intuitive workflows, and a range of helpful features that simplify the design process and enhance the overall user experience.

The main drawback of these open source tools is that they are not comprehensive. Some of them are capable of PCB design (e.g. KiCad) while some of them are capable of performing simulations (e.g. gEDA). To the best of our knowledge, there is no open source software that can perform circuit design, simulation and layout design together. eSim is capable of doing all of the above.

Chapter 2

Feautures Of eSim

The objective behind the development of eSim is to provide an open source EDA solution for electronics and electrical engineers. The software should be capable of performing schematic creation, PCB design and circuit simulation (analog, digital and mixed-signal). It should provide facilities to create new models and components. Thus, eSim offers the following features -

1. Schematic Creation: eSim provides an easy-to-use graphical interface for drawing circuit schematics, making it accessible for users of all levels. Users can drag and drop components from the library onto the schematic, simplifying the design process. Comprehensive editing tools allow for easy modification of schematics, including moving, rotating, and labeling components.

2. Circuit Simulation: eSim supports SPICE (Simulation Program with Integrated Circuit Emphasis), a standard for simulating analog and digital circuits. Users can perform various types of analysis such as transient, AC, and DC, providing insights into circuit behavior over time and frequency.An integrated waveform viewer helps visualize simulation results, aiding in the analysis and debugging of circuit designs.

3. PCB Design: The PCB layout editor allows users to place components and route traces with precision. eSim includes DRC capabilities to ensure that the PCB design adheres to manufacturing constraints and electrical rules. Users can generate Gerber files, which are standard for PCB fabrication, directly from their designs.

4. Subcircuit Feature: This feature enables users to create complex circuits by integrating smaller, simpler subcircuits, promoting modular and hierarchical design approaches. Subcircuits can be reused in different projects, saving time and effort in redesigning common circuit elements.

5. Open Source Integration: eSim integrates several open-source tools like KiCad, Ngspice, and GHDL, providing a comprehensive suite for electronic design automation. Being open-source, eSim is free to use, making advanced circuit design tools accessible without the need for expensive licenses.

Chapter 3

Problem Statement

To design and develop various Analog and Digital Integrated Circuit Models in the form of sub-circuits using device model files already present in the eSim library. These IC models should be useful in the future for circuit designing purposes by developers and users, once they get successfully integrated into the eSim subcircuit Library.

3.1 Approach

Our approach to implementing the problem statement began with examining datasheets from prominent Integrated Circuit (IC) manufacturers such as Texas Instruments, Analog Devices, and NXP Semiconductors. we selected ICs that offer a diverse range of functionalities, including precision amplifiers, comparators, encoders, and audio amplifiers. After building the subcircuits, we tested them to verify basic circuit configurations using NgSpice simulations. The step-by-step roadmap of this process is outlined below :

1. Analyzing Datasheets : The primary step is to browse through various analog and digital IC datasheets, and hence find suitable circuits to implement in eSim, that are not previously included into the eSim library. Check for the detailed schematic of the IC's and once the component values and the truth table is ascertained, then finalise the IC to be created.

2. Subcircuit Creation : After deciding the IC, we start modeling it as a sub-circuit in eSim, using the model files present in the eSim device model library only. The design is strictly according to the information given in the official data-sheets of the ICs.This step also includes building the Symbol/Pin diagram of the IC according to the packaging and pin description given in the data-sheets only.

3. Test Circuit Design : Once the component of the IC is ready, now we can build the test circuits, according to the data-sheets. In this step we build the test cases and test circuits using the component IC.

4. Schematic Testing : Once the test circuits are ready, now it's time to simulate the test circuits so that the output can be obtained in the form of wave-forms and plots. Here we take help of KiCad to NgSpice conversion and Simulation feature in eSim

If the output of the test circuit is not as per expectation, this implies that the test case has failed, and there is some error in the schematic. In such cases we go back to the design phase of the IC or the test circuits, to look for possible errors and then repeat the testing process again after making required changes.

Once the expected output of the test cases are correct and satisfy the expected results, then in such a case the IC is declared successfully working. The test case has been verified and the designing process is complete.

Chapter 4

Analog IC's

4.1 LT1011-Voltage Comparator

The LT1011 is a precision voltage comparator designed for high accuracy and reliability in signal processing tasks. It operates over a wide supply voltage range and is suitable for both single and dual-supply applications. With low offset voltage and high input impedance, it ensures precise performance, making it ideal for applications like voltage monitoring and signal detection. Its fast response time and open-collector output allow for flexible integration into various circuits, making the LT1011 a popular choice in industrial and consumer electronics.

The LT1011 IC is used in a variety of applications, including:

1. SAR A/D Converters: It helps in converting analog signals to digital with precision in Successive Approximation Register (SAR) systems.

2. Voltage-to-Frequency Converters: The IC is used to convert voltage levels into corresponding frequency signals.

3. Precision RC Oscillators: It aids in creating precise timing signals using resistor-capacitor (RC) circuits.

4. Peak Detectors: The LT1011 is used to capture and hold the peak value of a signal.

5. Motor Speed Control: It helps regulate motor speed by comparing voltage levels.

6. Pulse Generators: The IC is used to generate pulses for timing and control applications.

4.1.1 IC Layout

This figure represents the 8-Pin Package Diagram of the LT1011 Voltage Comparator

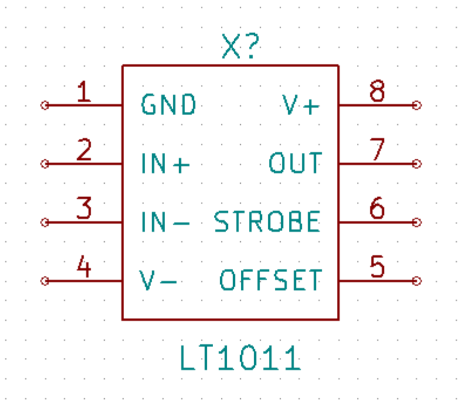

Figure 4.1: LT1011 Voltage Comparator

4.1.2 Subcircuit Schematic Diagram

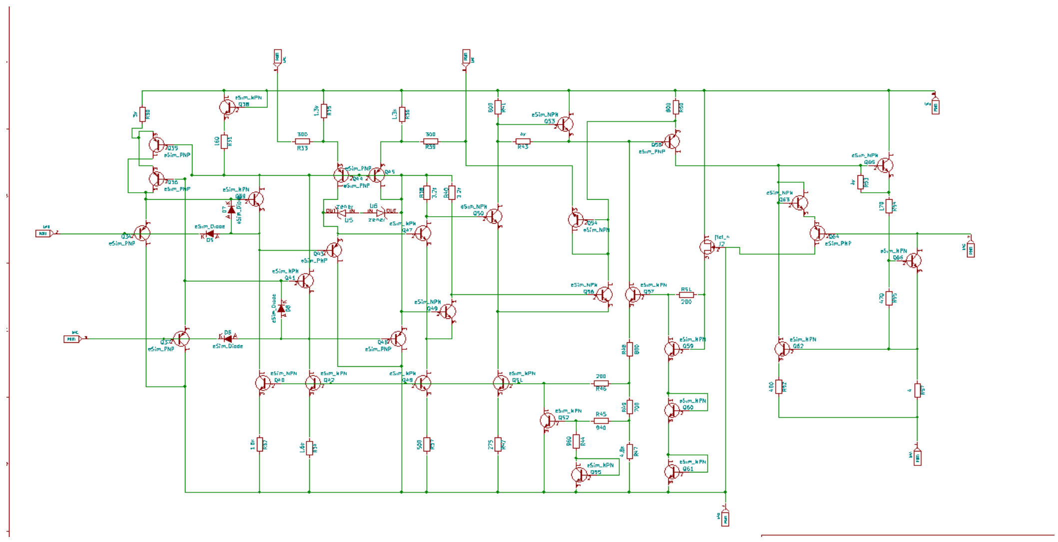

Figure 4.2: Subcircuit Schematic of LT1011

4.1.3 Test Circuit

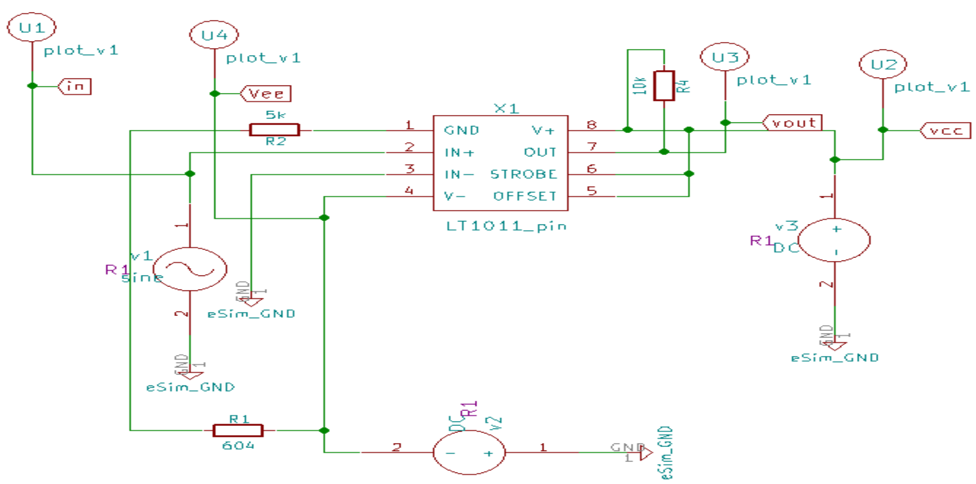

Figure 4.3: Test Circuit of LT1011 IC

4.1.4 Input Plots

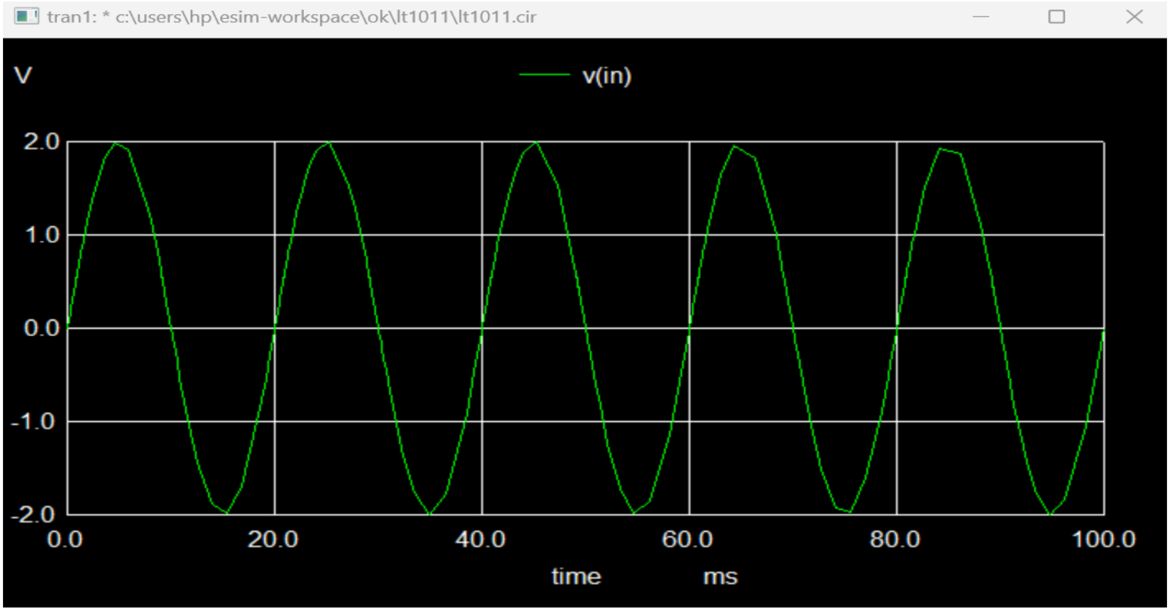

Figure 4.4: Input Voltage Waveform of LT1011

4.1.5 Output Plots

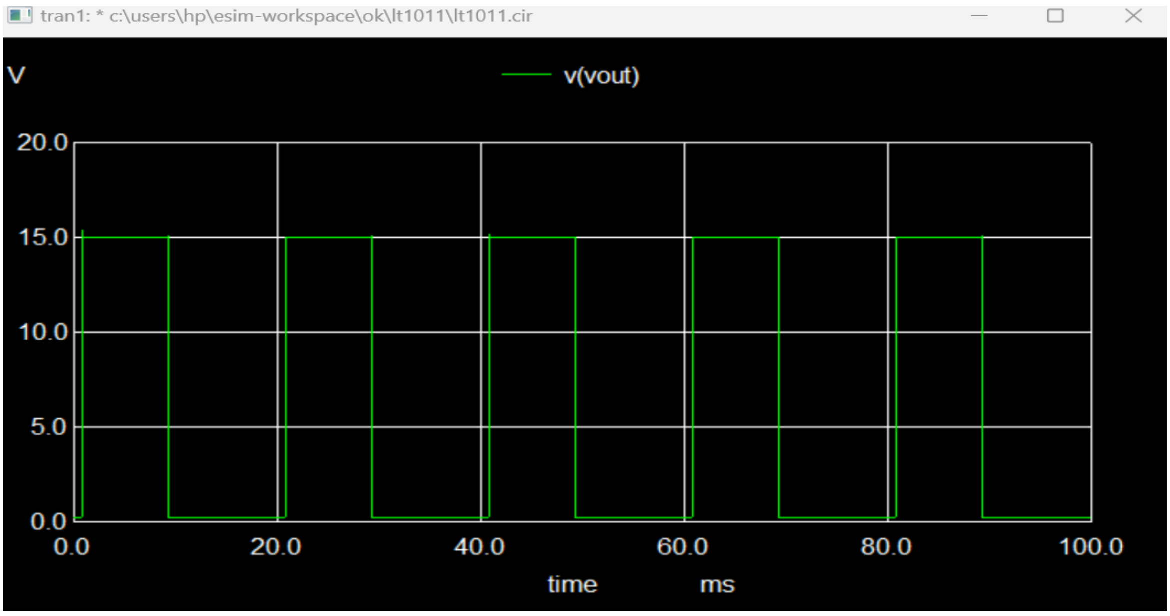

Figure 4.5: Output Voltage Waveform of LT1011

4.2 MC7900-Negative Voltage Regulator

The MC7900 is a series of negative voltage regulators used to provide a stable and consistent negative output voltage. It is commonly used in electronic circuits that require a fixed negative voltage supply. These regulators come in various output voltage options, such as -5V, -12V, and -15V, to suit different needs. The MC7900 is designed to maintain a steady voltage despite changes in load current or input voltage, making it ideal for powering analog and digital circuits that need a reliable negative voltage. It includes built-in protection features like thermal shutdown and current limiting to prevent damage from overheating or excessive current.

Applications:

1. Power Supply Design: The MC7900 is used to provide a stable negative voltage in power supply circuits for various electronic devices.

2. Operational Amplifier Circuits: It supplies the negative voltage needed for dual-supply op-amp configurations, enabling signal amplification and processing.

3. Analog Signal Processing: The regulator ensures a consistent negative voltage for analog circuits, improving the performance of audio and sensor systems.

4. Communication Equipment: It provides necessary negative bias voltages for components in communication devices, ensuring proper operation.

5. Instrumentation: The MC7900 is used in measurement and testing equipment to maintain stable negative voltages, enhancing accuracy and reliability.

4.2.1 IC Layout

This figure represents the 3-Pin Package Diagram of the MC7900 Negative Voltage Comparator

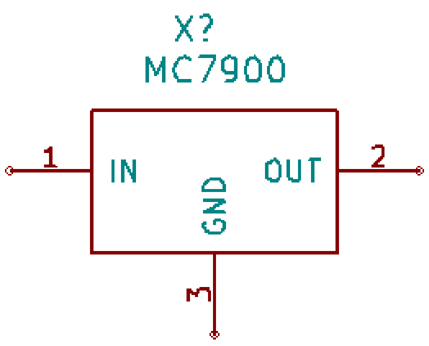

Figure 4.6: MC7900 Negative Voltage Regulator

4.2.2 Subcircuit Schematic Diagram

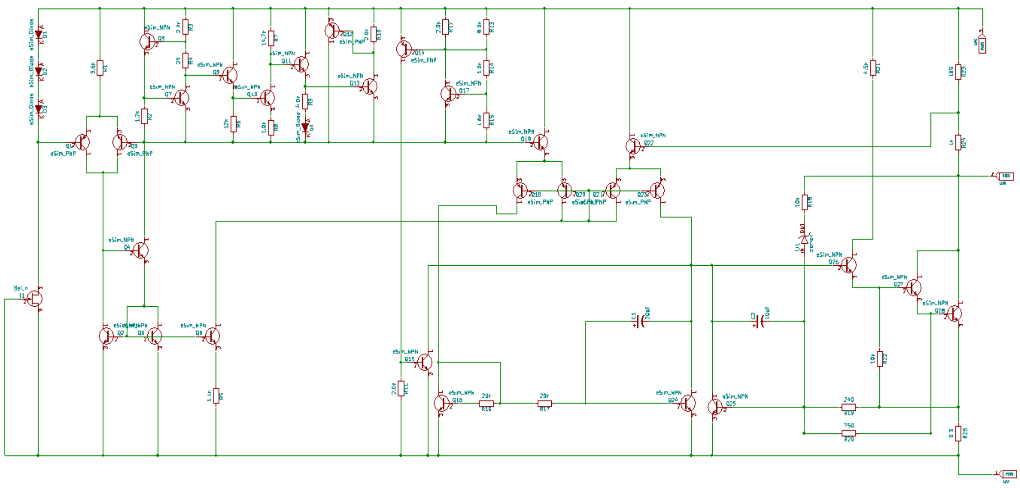

Figure 4.7: Subcircuit Schematic of MC7900

4.2.3 Test Circuit

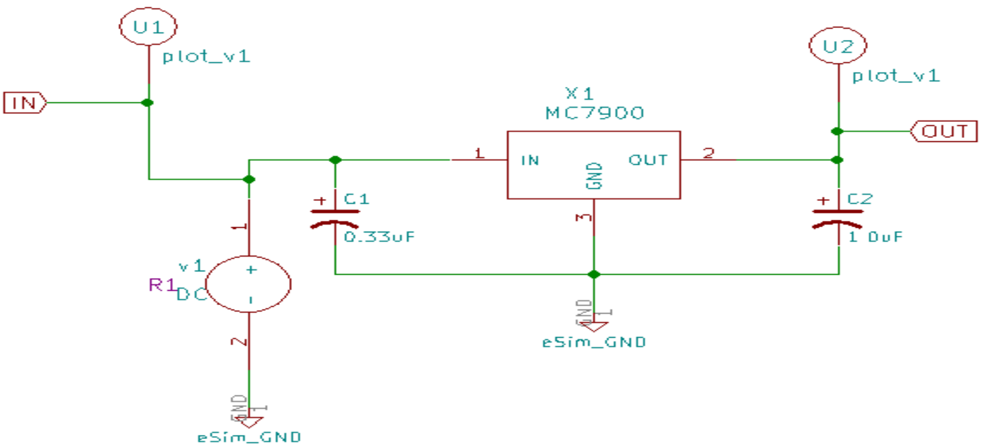

Figure 4.8: Test Circuit of MC7900 IC

4.2.4 Input Plots

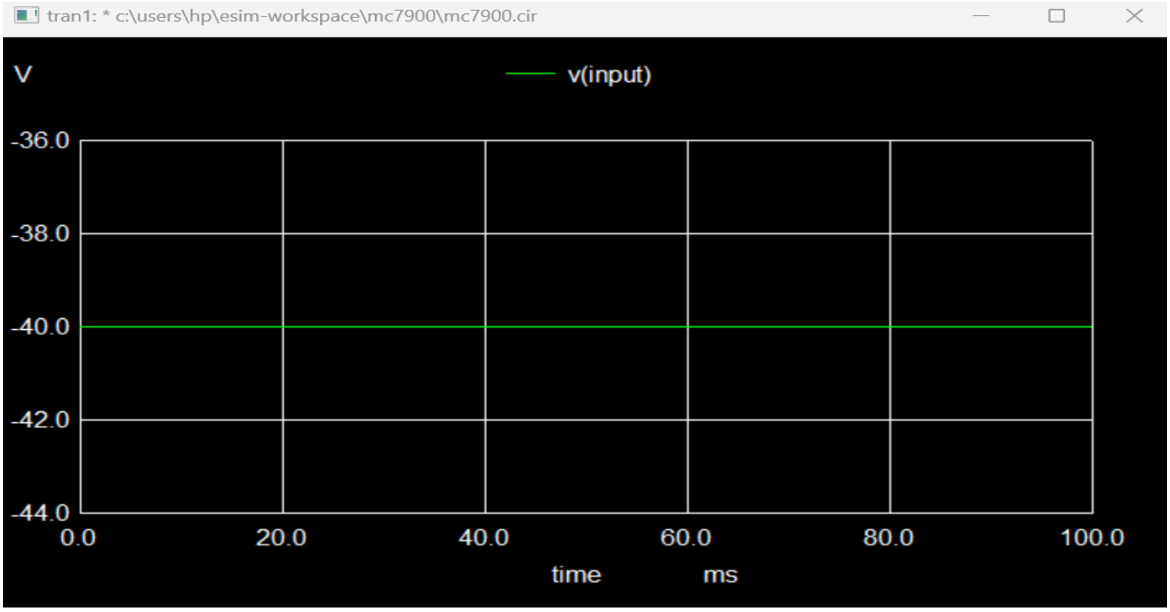

Figure 4.9: Input Voltage Waveform of MC7900

4.2.5 Output Plots

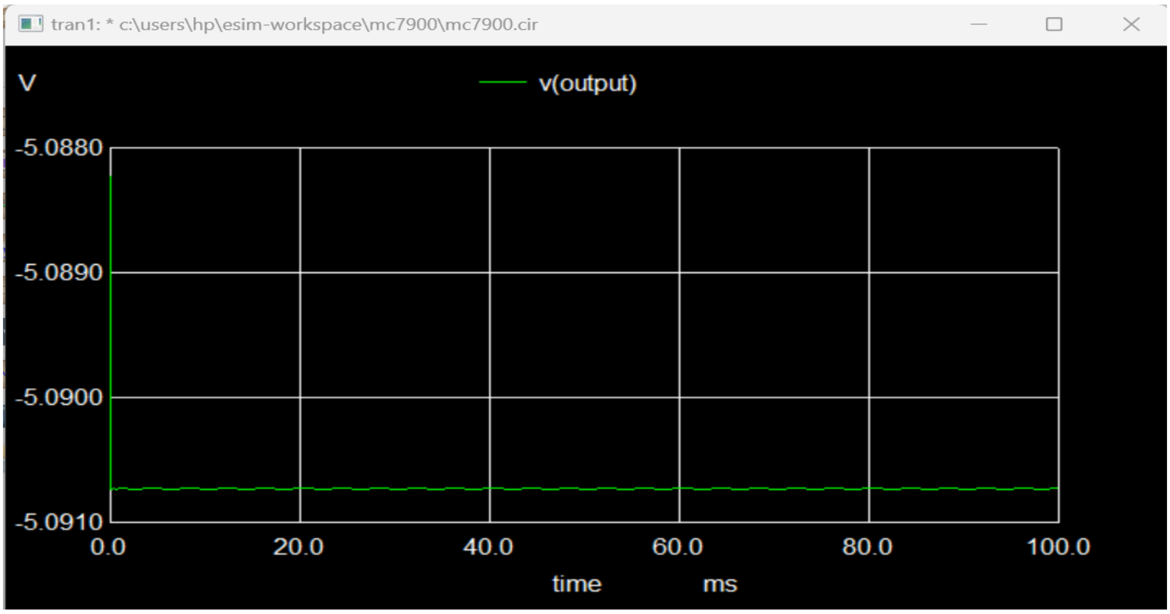

Figure 4.10: Output Voltage Waveform of MC7900

4.3 LM78L-100-mA Fixed Output Linear Regulator

The LM78L series are fixed-output voltage regulators that provide a stable output voltage for low-power applications. They typically offer output voltages of 5V, 6V, 8V, 9V, 10V, 12V, and 15V, with a maximum output current of 100 mA. These regulators require an input voltage that is 2-3 volts higher than the output voltage.

Applications:

1.Battery Chargers: They provide a stable voltage output, making them ideal for charging batteries without overcharging or damaging them.

2.Portable Instrumentation: The compact size and reliable output make them perfect for portable devices that require consistent power supply.

3.LED Lighting: They offer a steady current to LEDs, ensuring uniform brightness and extended lifespan.

4.Appliances: These regulators are used in household appliances to maintain a constant voltage level, improving performance and reliability.

4.3.1 IC Layout

This figure represents the 3-Pin Package Diagram of the Fixed Output Linear Regulator

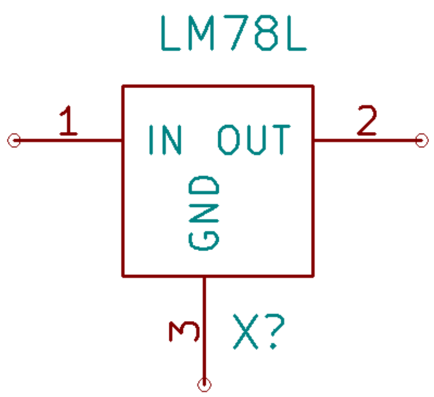

Figure 4.11: LM78L Fixed Output Linear Regulator

4.3.2 Subcircuit Schematic Diagram

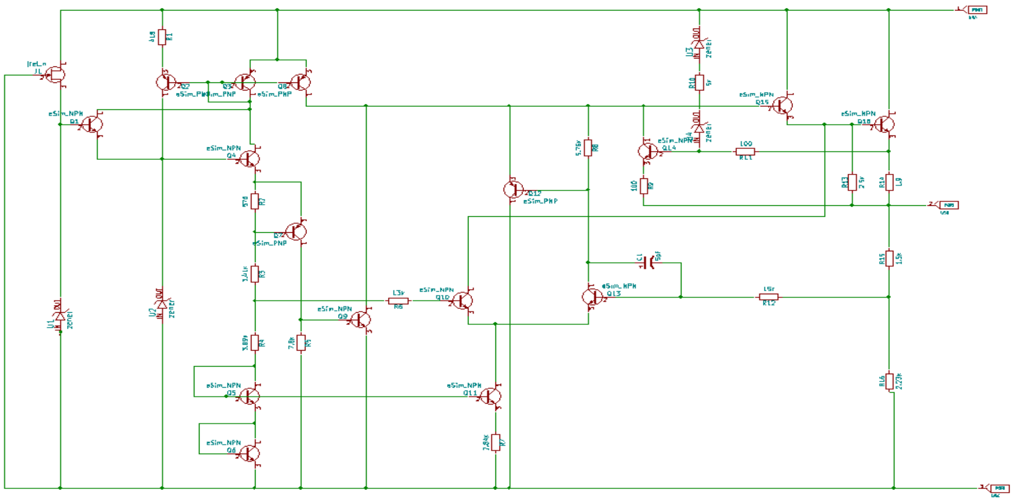

Figure 4.12: Subcircuit Schematic of LM78L

4.3.3 Test Circuit

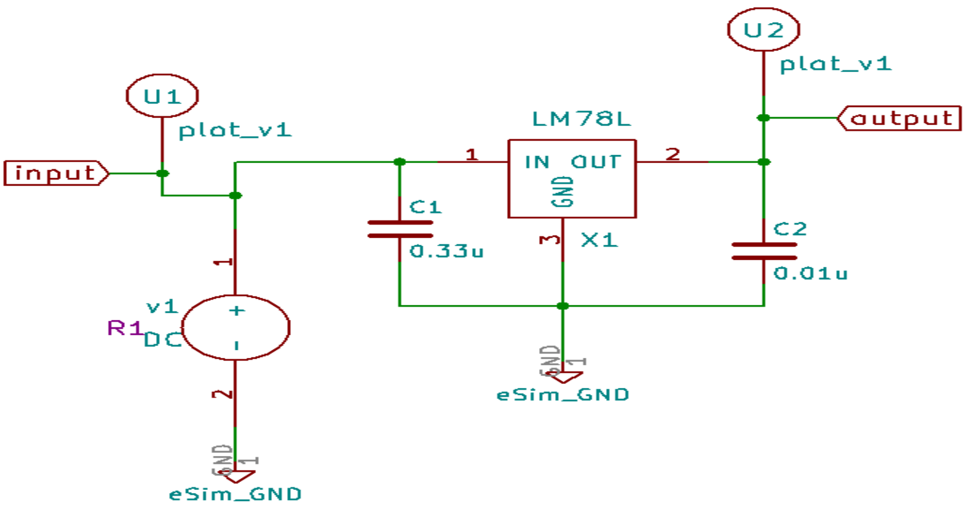

Figure 4.13: Test Circuit of LM78L IC

4.3.4 Input Plots

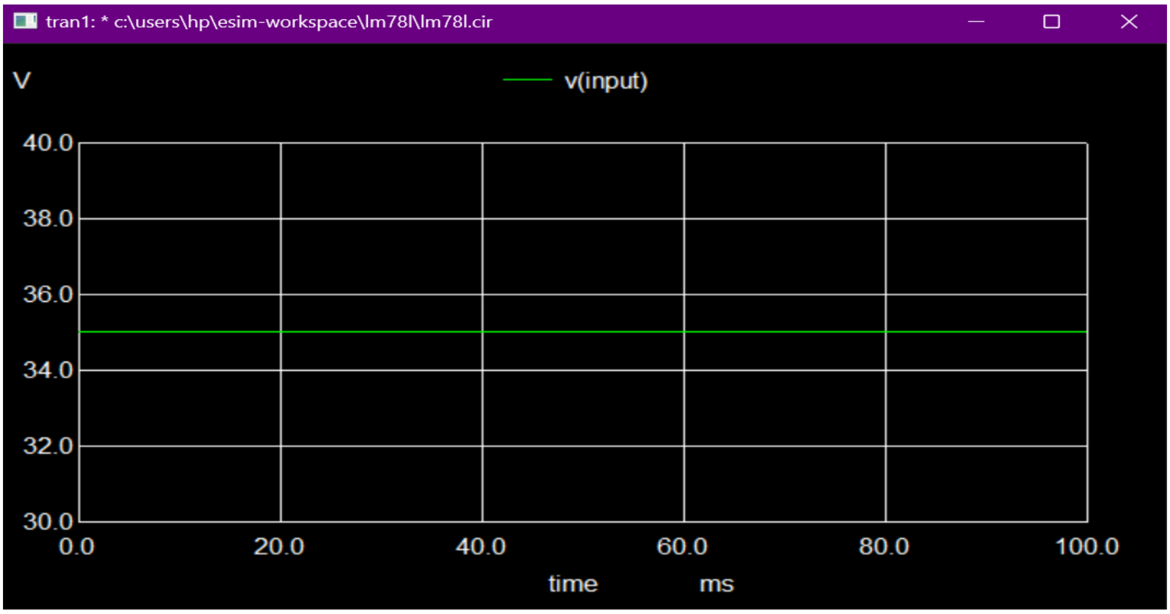

Figure 4.14: Input Voltage Waveform of LM78L

4.3.5 Output Plots

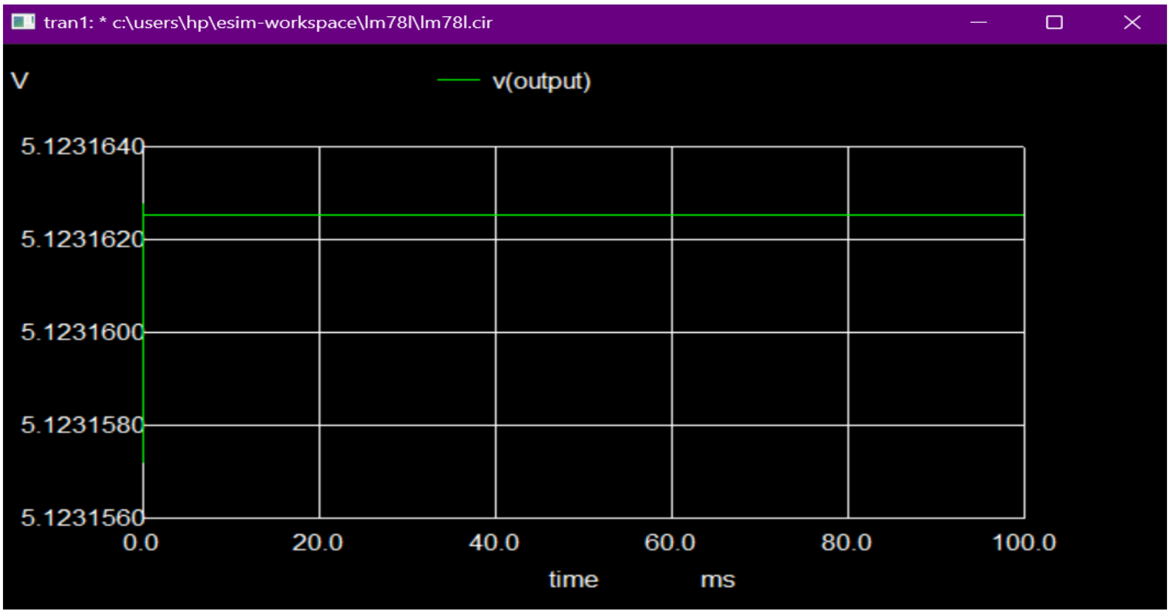

Figure 4.15: Output Voltage Waveform of MC7900

4.4 LM140K- 3 terminal positive regulator

The LM140K is a three-terminal positive voltage regulator designed to provide a fixed output voltage with high reliability and ease of use. It is capable of delivering up to 1.5A of output current, making it suitable for a wide range of applications requiring a stable power supply.

Key Features:

1. Fixed Output Voltage: Available in standard output voltages like 5V, 12V, and 15V.

2. High Output Current: Can supply up to 1.5A, supporting moderate power needs.

3. Built-in Protections: Includes features like thermal overload protection, short-circuit protection, and safe area protection to enhance reliability.

4. Minimal External Components: Requires only a few external capacitors for stability, simplifying circuit design.

5. Package Options: Typically available in TO-3 and TO-220 packages, offering flexibility in mounting and heat dissipation.

4.4.1 IC Layout

This figure represents the 3-Pin Package Diagram of the 3 terminal positive regulator

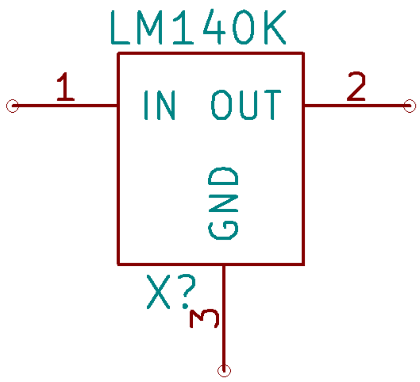

Figure 4.16: LM140K- 3 terminal positive regulator

4.4.2 Subcircuit Schematic Diagram

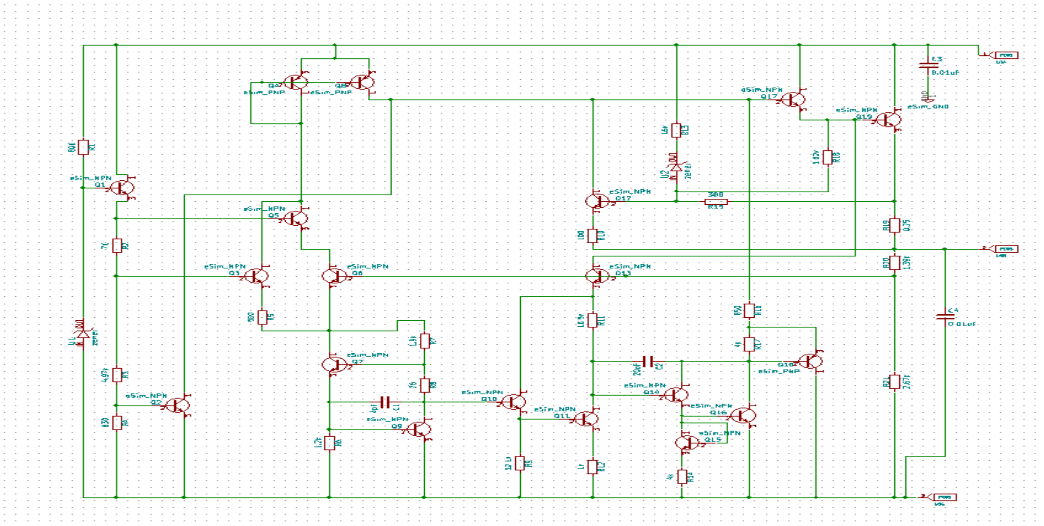

Figure 4.17: Subcircuit Schematic of LM140K

4.4.3 Test Circuit

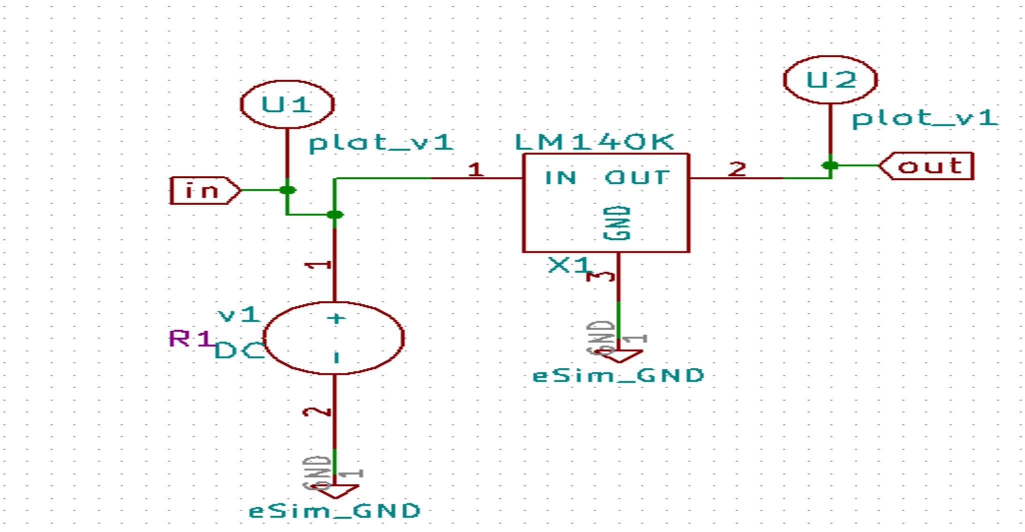

Figure 4.18: Test Circuit of LM140K IC

4.4.4 Input Plots

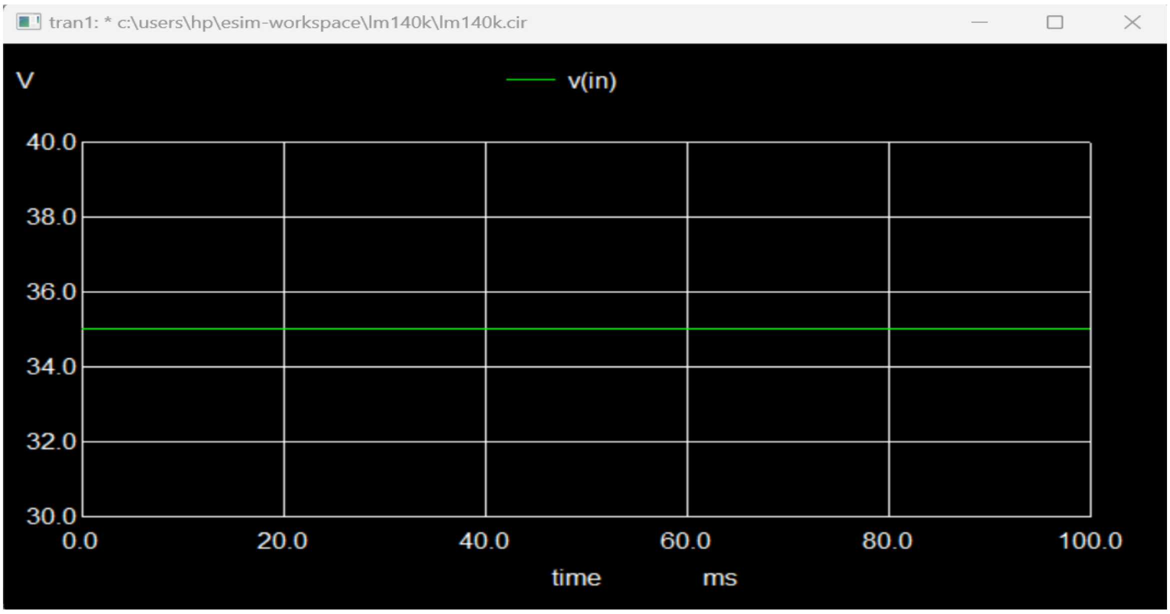

Figure 4.19: Input Voltage Waveform of LM140K

4.4.5 Output Plots

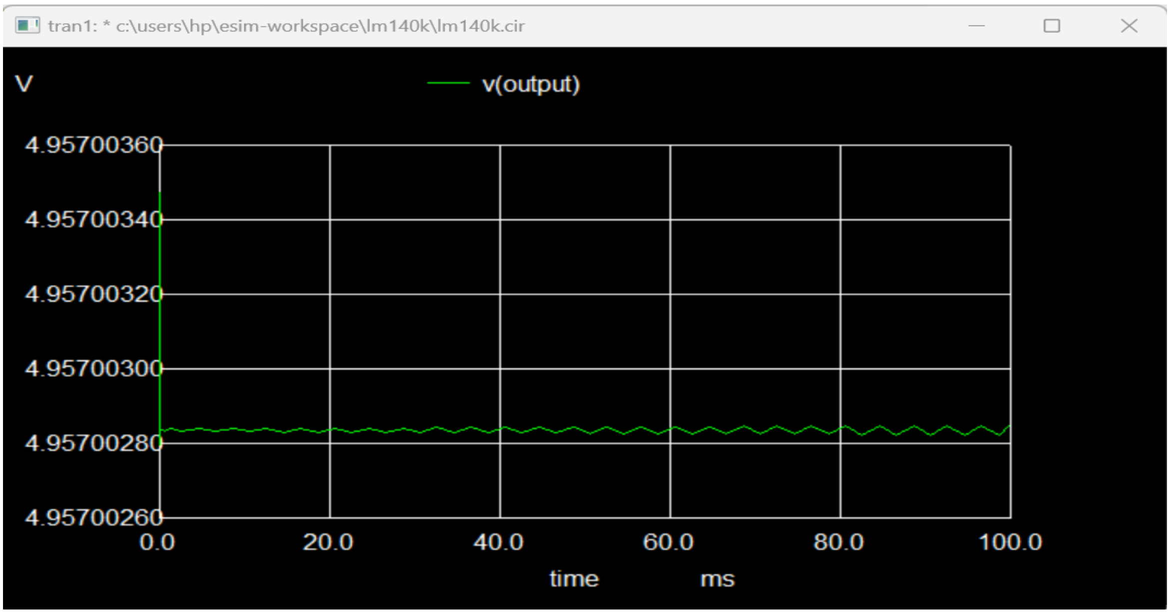

Figure 4.20: Output Voltage Waveform of LM140K

4.5 UA777-Precision Operational Amplifier

The UA777 is a precision operational amplifier designed for high-performance analog applications requiring accuracy and stability. It features low offset voltage, low drift, and excellent linearity, making it suitable for precision signal processing tasks.

Key Features:

1. Low Offset Voltage: Provides minimal input offset, enhancing accuracy in applications where precision is critical.

2. Low Drift: Maintains stability over temperature changes, ensuring consistent performance.

3. High Input Impedance: Offers high input impedance, minimizing loading on the input source.

4. Wide Bandwidth: Capable of handling a wide range of frequencies, making it versatile for various applications.

5. Low Noise: Delivers low noise performance, which is essential for sensitive analog circuits.

4.5.1 IC Layout

This figure represents the 8-Pin Package Diagram of the Precision Operational Amplifier

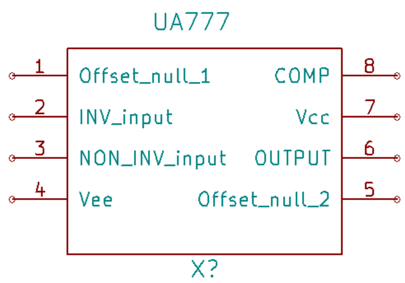

Figure 4.21: UA777-Precision Operational Amplifier

4.5.2 Subcircuit Schematic Diagram

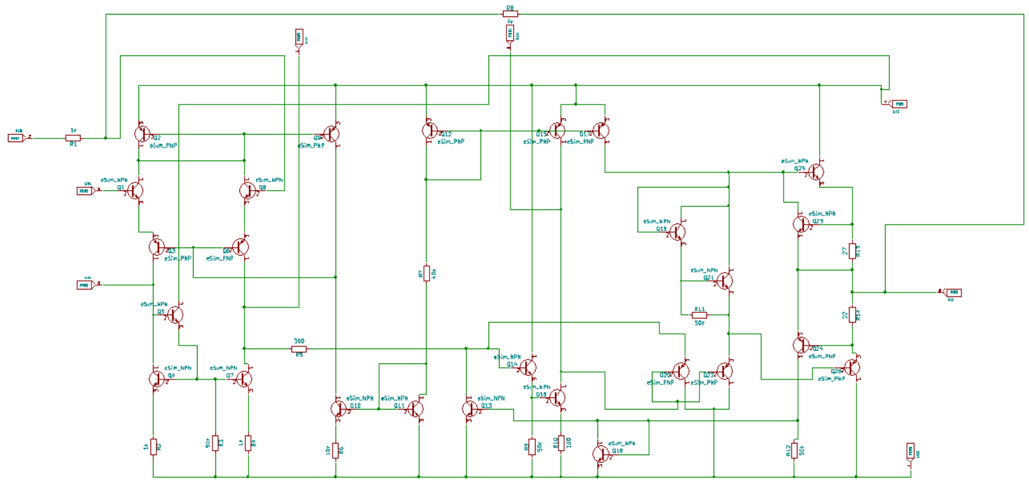

Figure 4.22: Subcircuit Schematic of UA777

4.5.3 Test Circuit

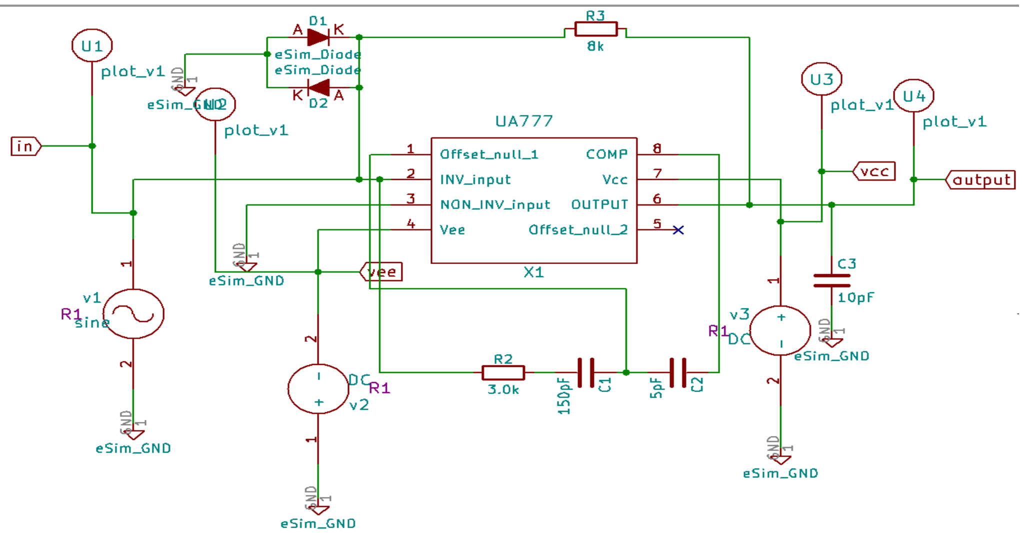

Figure 4.23: Test Circuit of UA777 IC

4.5.4 Input Plots

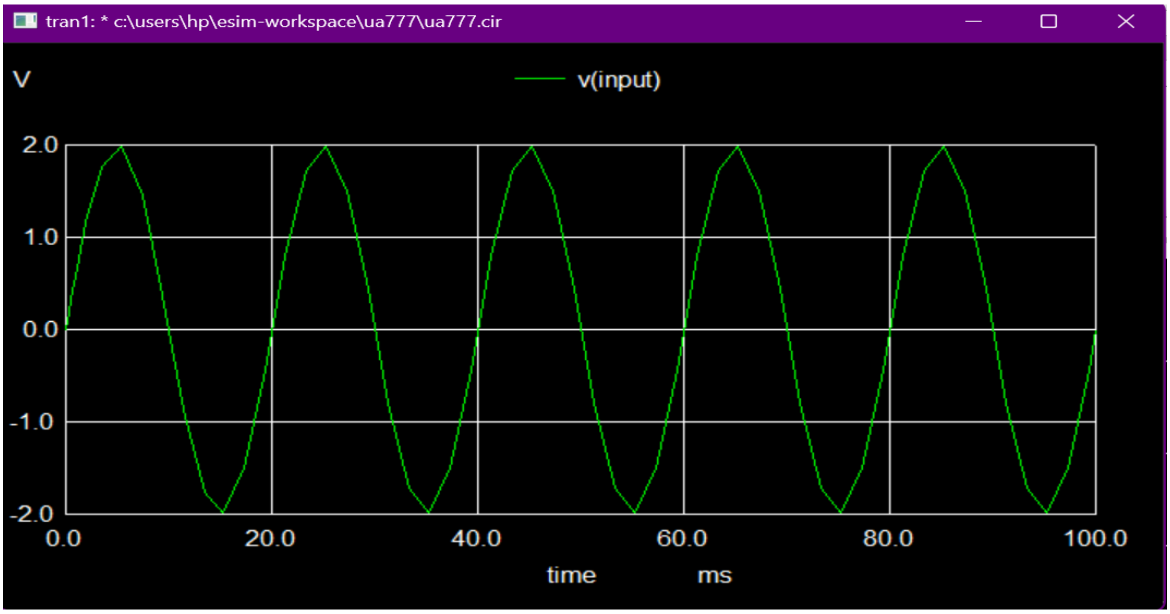

Figure 4.24: Input Voltage Waveform of UA777

4.5.5 Output Plots

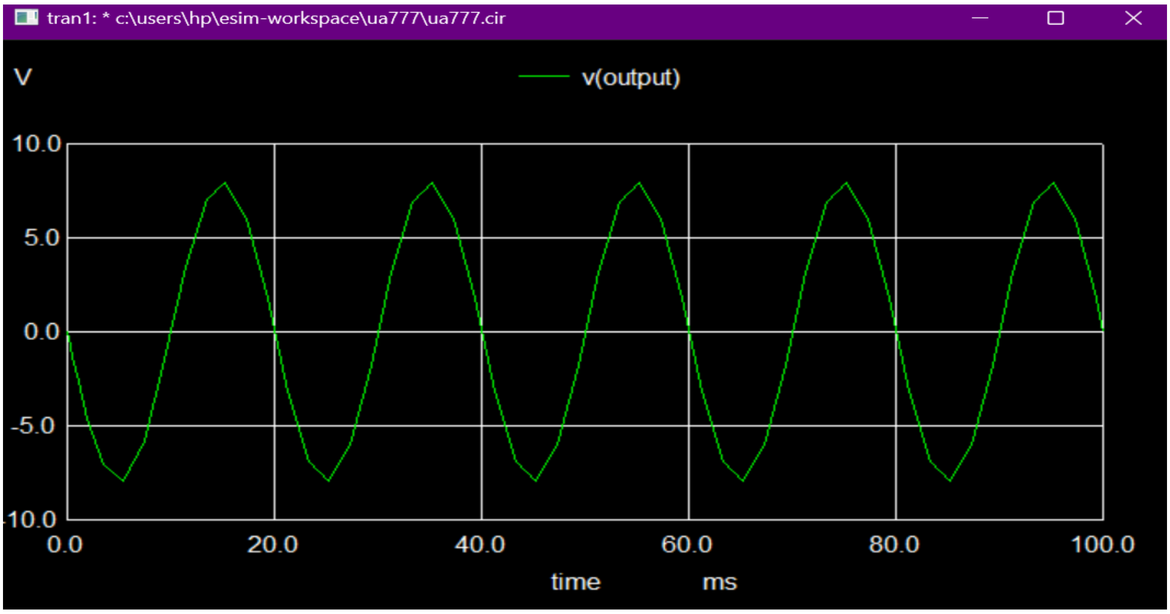

Figure 4.25: Output Voltage Waveform of UA777

4.6 MC1741- Internally Compensated, High Performance Operational Amplifier

The MC1741C is a high-performance operational amplifier designed for applications requiring precision and versatility. It is internally compensated, making it easy to use in a variety of circuits without needing additional frequency compensation components.

Key Features:

- **No Frequency Compensation Required:** Simplifies design by eliminating the need for external compensation.

- **Short Circuit Protection:** Protects the device from damage in the event of accidental short circuits.

- **Offset Voltage Null Capability:** Allows adjustment of offset voltage for improved accuracy.

- **Wide Voltage Ranges:** Supports broad common-mode and differential voltage ranges, enhancing its versatility.

- **Low Power Consumption:** Efficient power usage makes it suitable for battery-operated devices.

- **No Latch-Up:** Ensures reliable performance without risk of latch-up under normal operating conditions.

4.6.1 IC Layout

This figure represents the 8-Pin Package Diagram of the Internally Compensated, High Performance Operational Amplifier

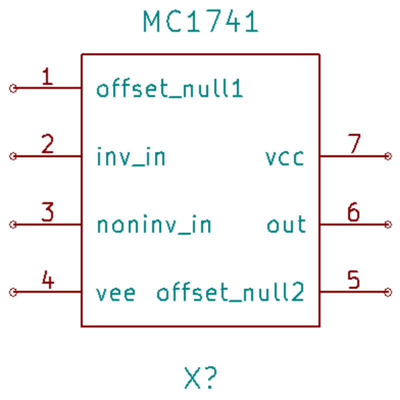

Figure 4.26: MC1741-Internally Compensated, High Performance Operational Amplifier

4.6.2 Subcircuit Schematic Diagram

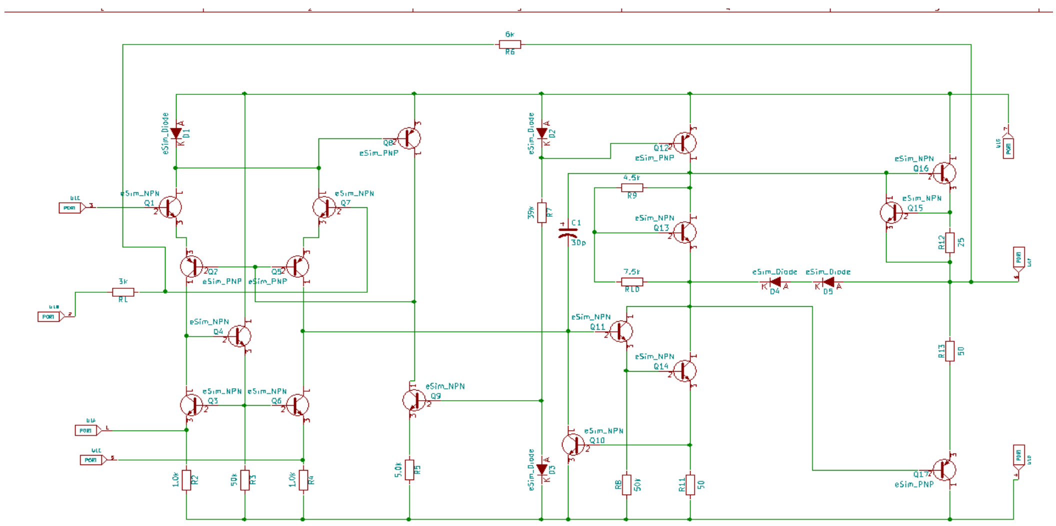

Figure 4.27: Subcircuit Schematic of MC1741

4.6.3 Test Circuit

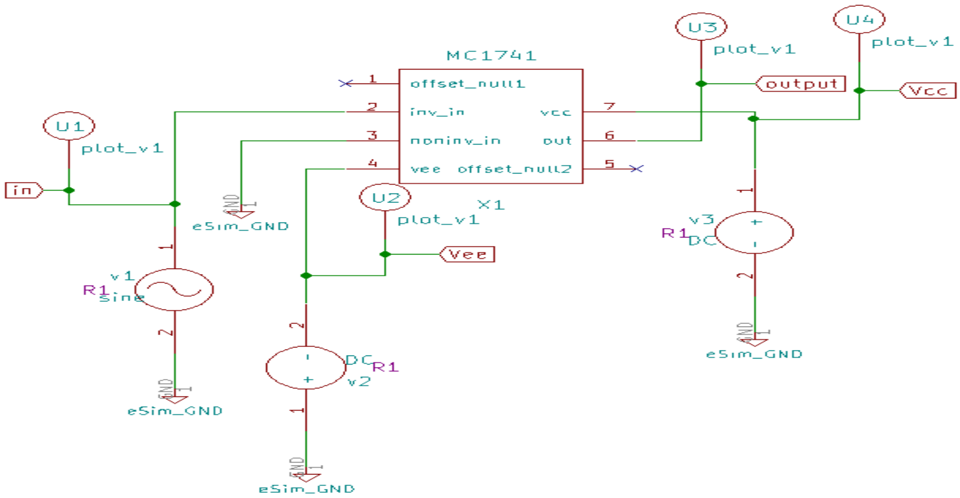

Figure 4.28: Test Circuit of MC1741 IC

4.6.4 Input Plots

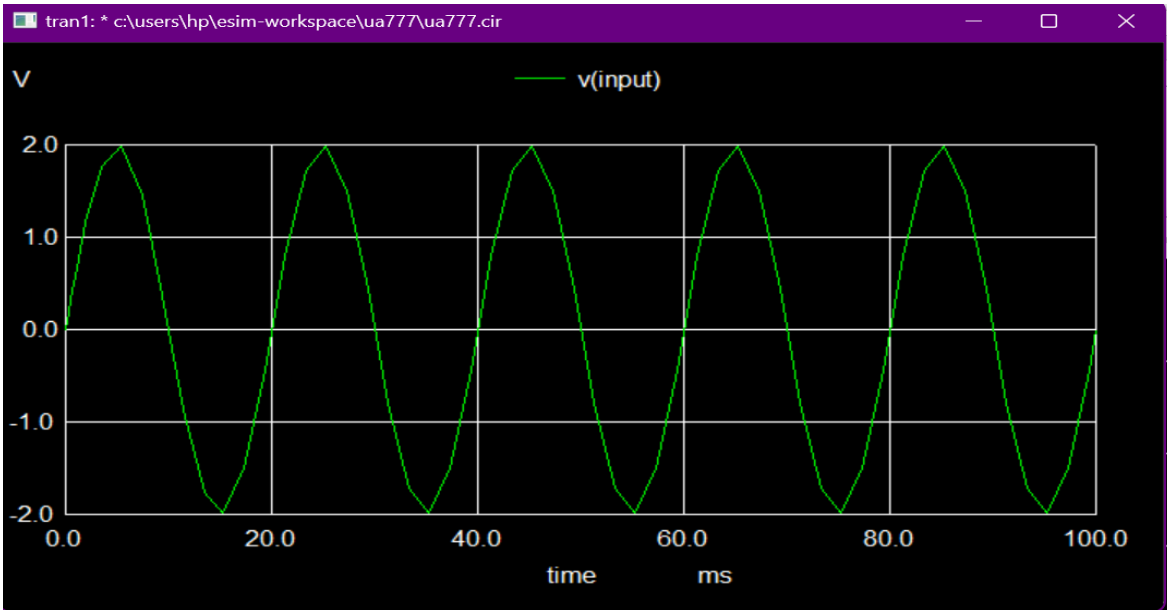

Figure 4.29: Input Voltage Waveform of MC1741

4.6.5 Output Plots

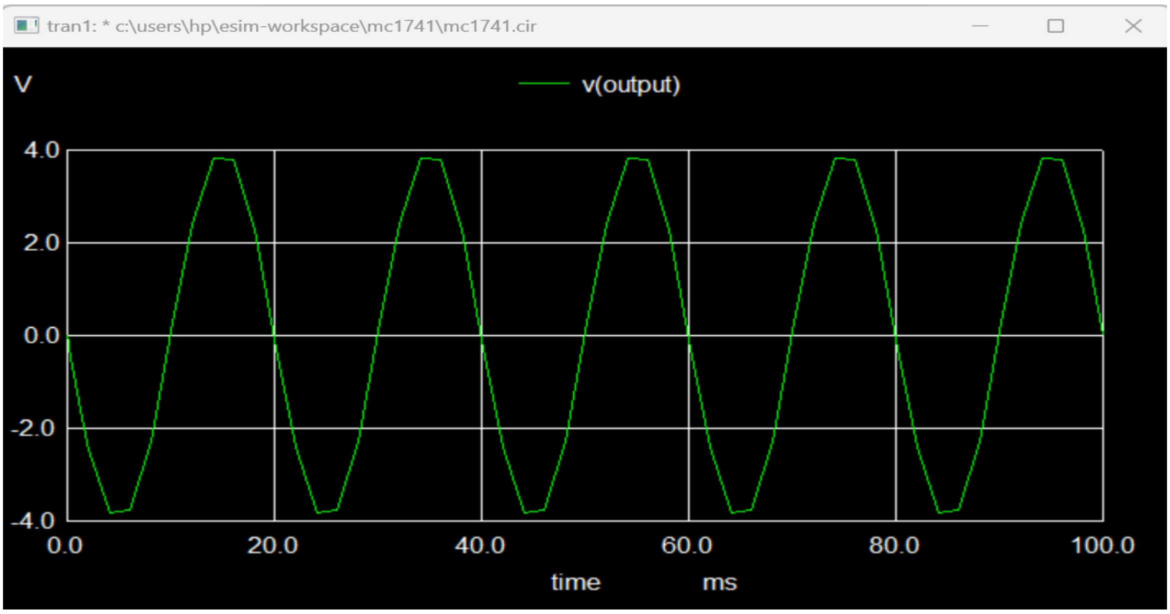

Figure 4.30: Output Voltage Waveform of MC1741

4.7 LM160 – High Speed Differential Comparator

The LM160 is a high-speed differential comparator designed for fast and accurate voltage comparison tasks. It is optimized for applications requiring rapid response and precise threshold detection.

Key Features:

- **High Speed:** Capable of propagation delays as low as 20 nanoseconds, making it suitable for high-frequency applications.

- **Wide Input Voltage Range:** Supports a broad range of input voltages, enhancing compatibility with various signal levels.

- **Differential Inputs:** Allows comparison of two input voltages, providing flexibility in circuit design.

- **TTL-Compatible Outputs:** Outputs are compatible with TTL logic levels, facilitating easy integration with digital systems.

- **Low Input Offset Voltage:** Ensures accurate comparison with minimal input voltage error

4.7.1 IC Layout

This figure represents the 8-Pin Package Diagram of the High Speed Differential Comparator

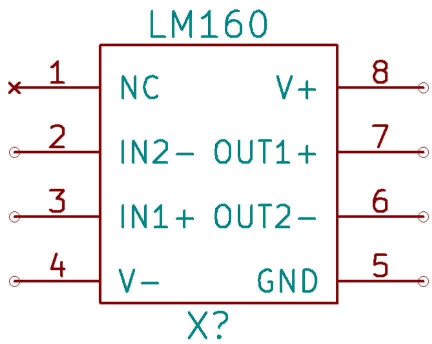

Figure 4.31: LM160 – High Speed Differential Comparator

4.7.2 Subcircuit Schematic Diagram

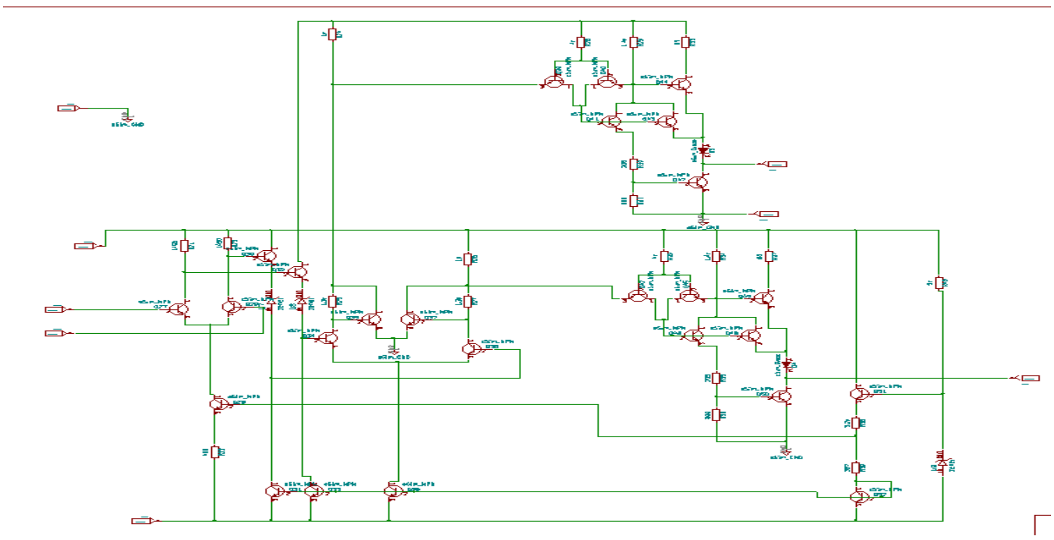

Figure 4.32: Subcircuit Schematic of LM160

4.7.3 Test Circuit

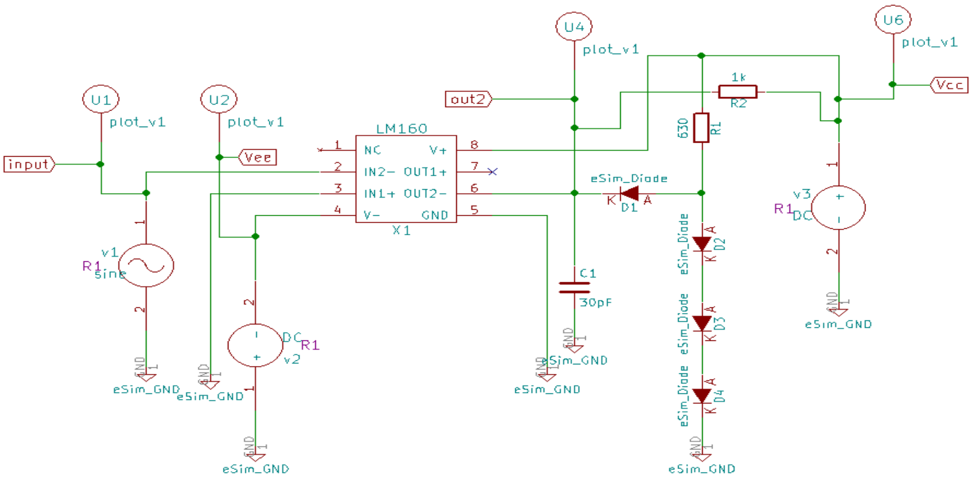

Figure 4.33: Test Circuit of LM160 IC

4.7.4 Input Plots

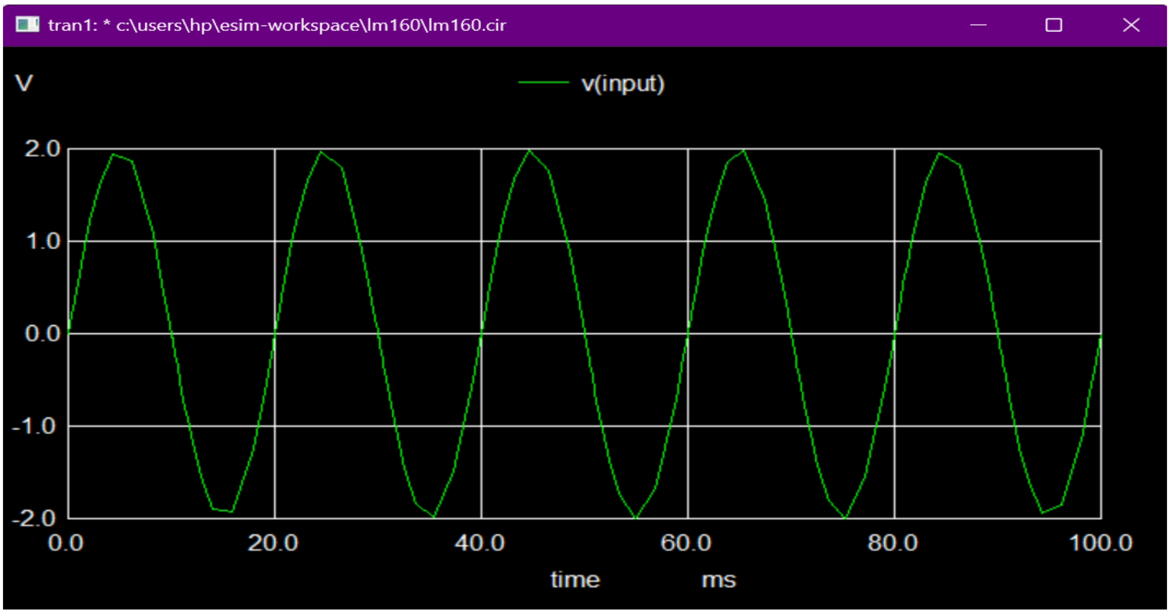

Figure 4.34: Input Voltage Waveform of LM160

4.7.5 Output Plots

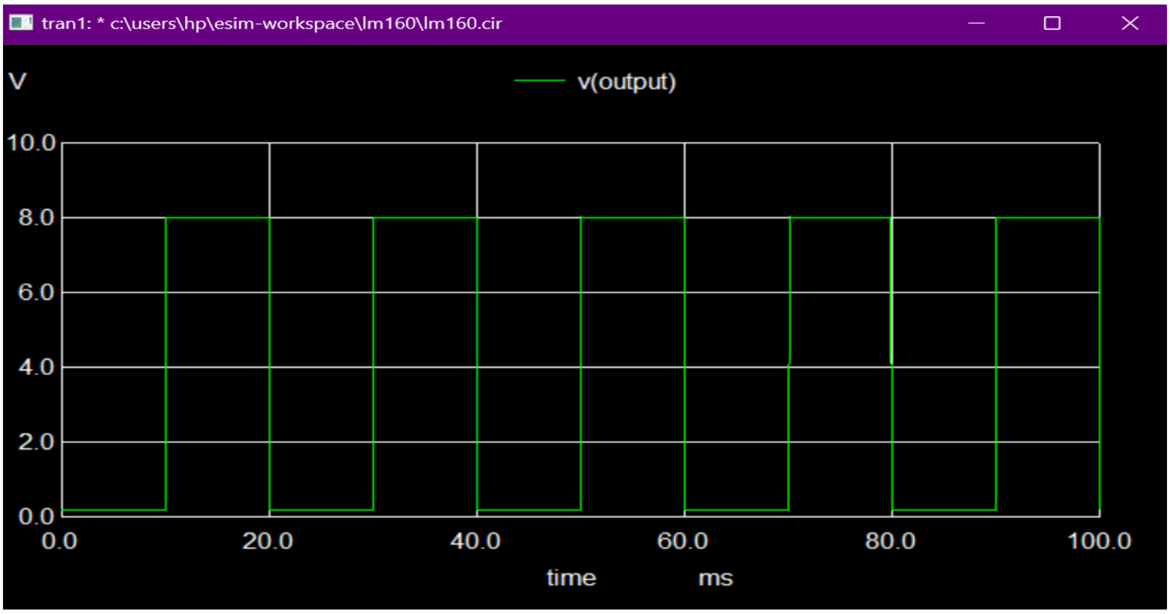

Figure 4.35: Output Voltage Waveform of LM160

4.8 LM119 Dual Comparator

The LM119 is a high-speed dual comparator designed for applications requiring rapid and accurate voltage comparisons. It features two independent comparators in a single package, making it versatile and efficient for complex circuit designs.

Key Features:

- **Dual Comparators:** Contains two independent comparators, offering flexibility in circuit configurations.

- **Single 5V Supply Operation:** Can operate from a single 5V power supply, simplifying power management.

- **Fast Response Time:** Provides a typical response time of 80 nanoseconds at ±15V, suitable for high-speed applications.

- **High Fan-out:** Supports a minimum fan-out of 2 on each side, allowing it to drive multiple loads.

- **Isolation Capability:** Inputs and outputs can be isolated from system ground, enhancing design flexibility.

- **High Common-Mode Slew Rate:** Capable of handling rapidly changing input signals without degradation.

4.8.1 IC Layout

This figure represents the 14-Pin Package Diagram of the Dual Comparator

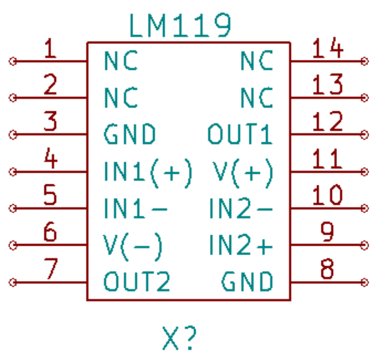

Figure 4.36: LM119 Dual Comparator

4.8.2 Subcircuit Schematic Diagram

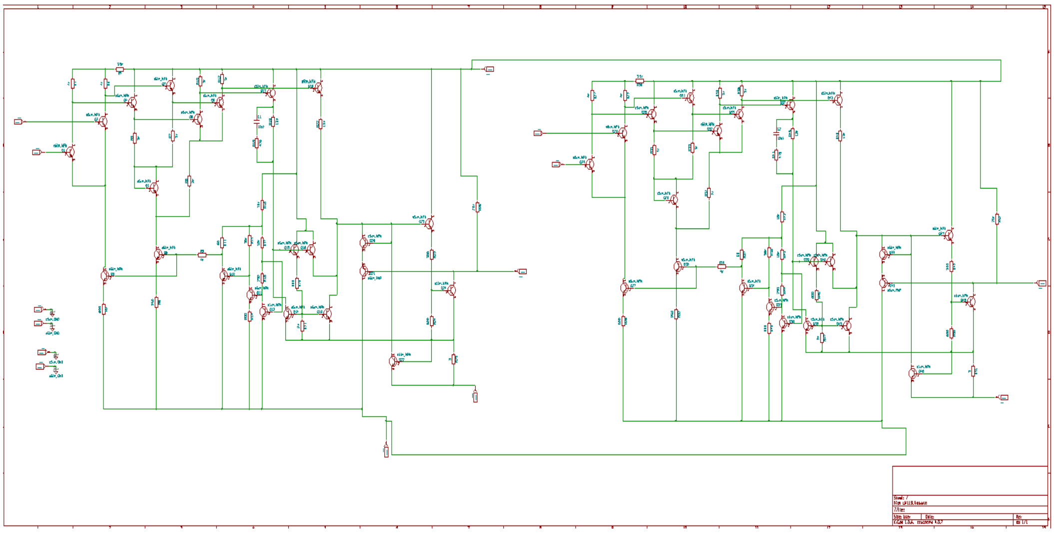

Figure 4.37: Subcircuit Schematic of LM119

4.8.3 Test Circuit

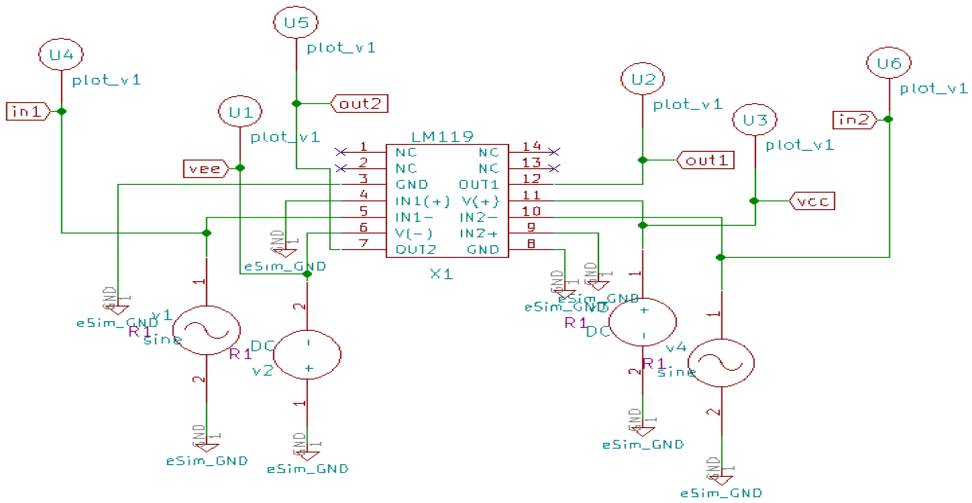

Figure 4.38: Test Circuit of LM119 IC

4.8.4 Input Plots

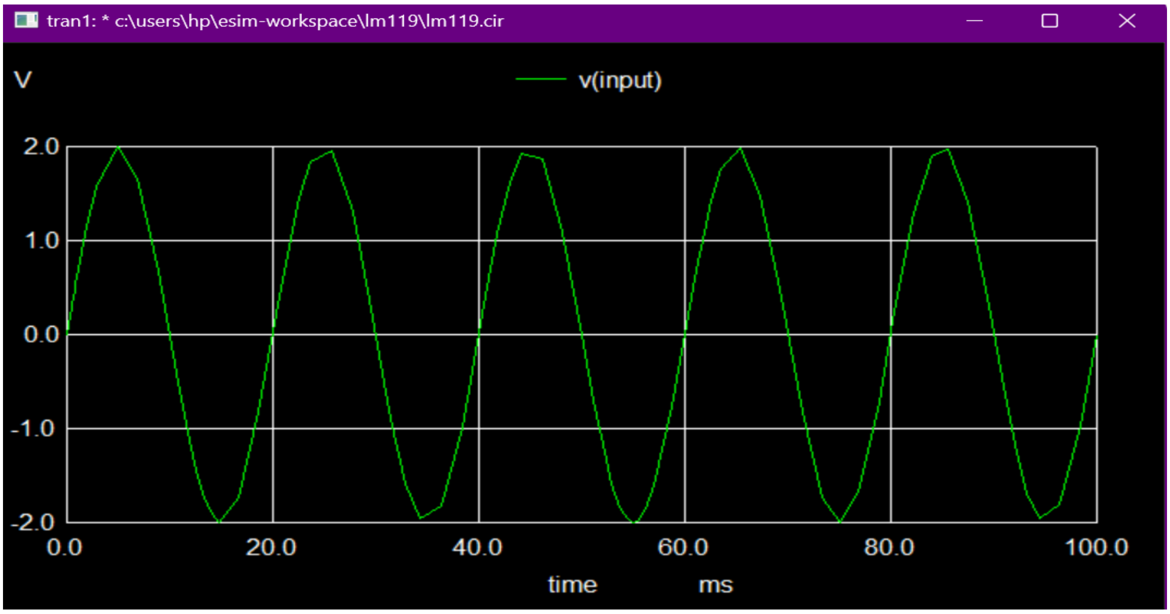

Figure 4.39: Input Voltage Waveform of LM119

4.8.5 Output Plots

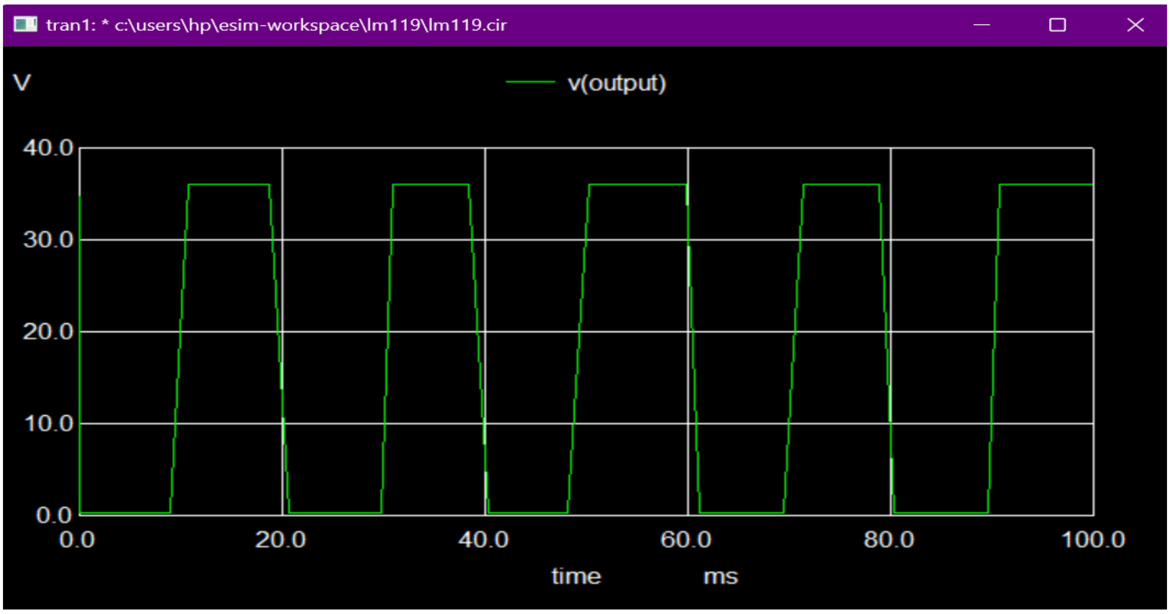

Figure 4.40: Output Voltage Waveform of LM119

4.9 LM4250 Programmable Operational Amplifier

The LM4250 is a versatile programmable operational amplifier known for its adaptability and low power usage. It supports a broad supply voltage range from ±1V to ±18V, making it flexible for various applications.

Key Features:

- **Wide Supply Range:** Operates from power supplies between ±1V and ±18V.

- **Low Offset Current:** Maintains an input offset current as low as 3 nA for high precision.

- **Minimal Power Consumption:** Offers a low standby power consumption of 500 µW, suitable for battery-operated devices.

- **No Need for Frequency Compensation:** Simplifies circuit design by not requiring external frequency compensation.

- **Customizable Characteristics:** Allows for programming of electrical characteristics to suit specific needs.

- **Offset Voltage Adjustment:** Includes the ability to nullify offset voltage for improved accuracy.

- **Battery Compatible:** Can be powered by two flashlight batteries, making it ideal for portable applications.

- **Short Circuit Protection:** Equipped with protection to prevent damage from short circuits.

4.9.1 IC Layout

This figure represents the 8-Pin Package Diagram of the Programmable Operational Amplifier

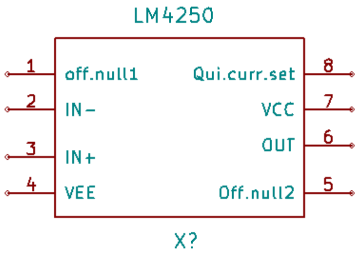

Figure 4.41: LM4250 Programmable Operational Amplifier

4.9.2 Subcircuit Schematic Diagram

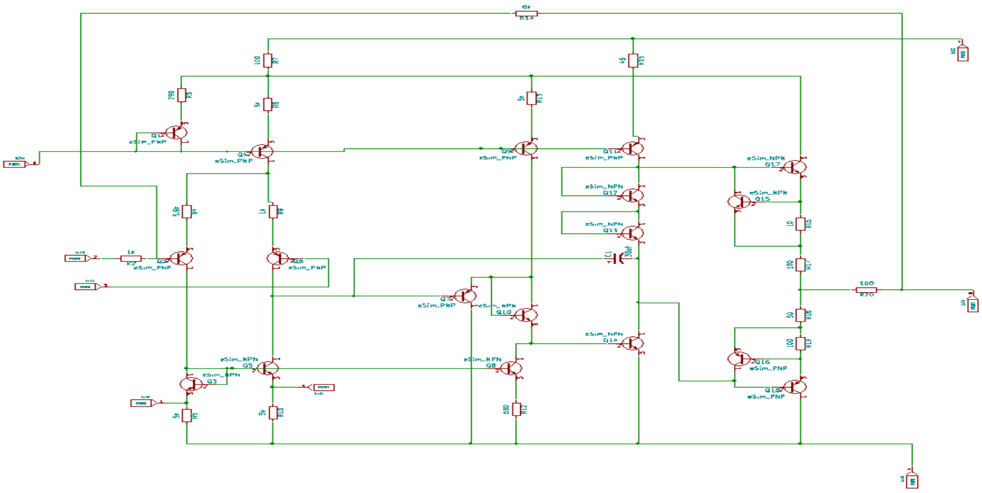

Figure 4.42: Subcircuit Schematic of LM4250

4.9.3 Test Circuit

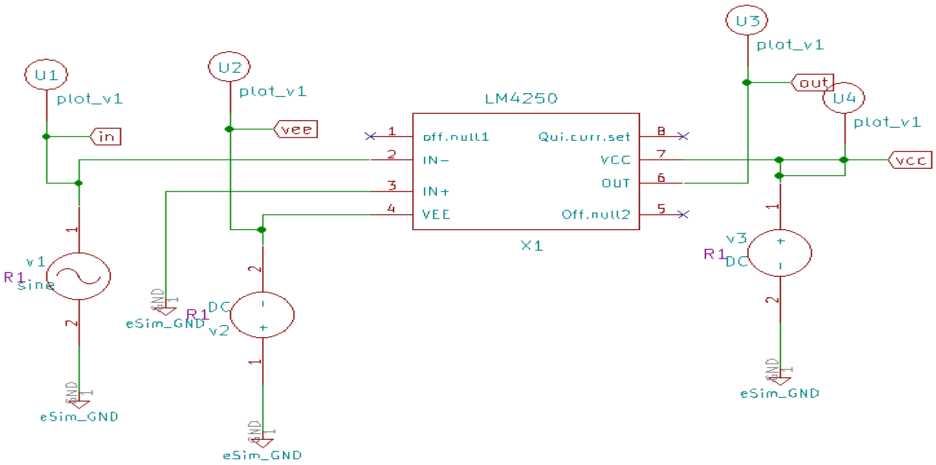

Figure 4.43: Test Circuit of LM4250 IC

4.9.4 Input Plots

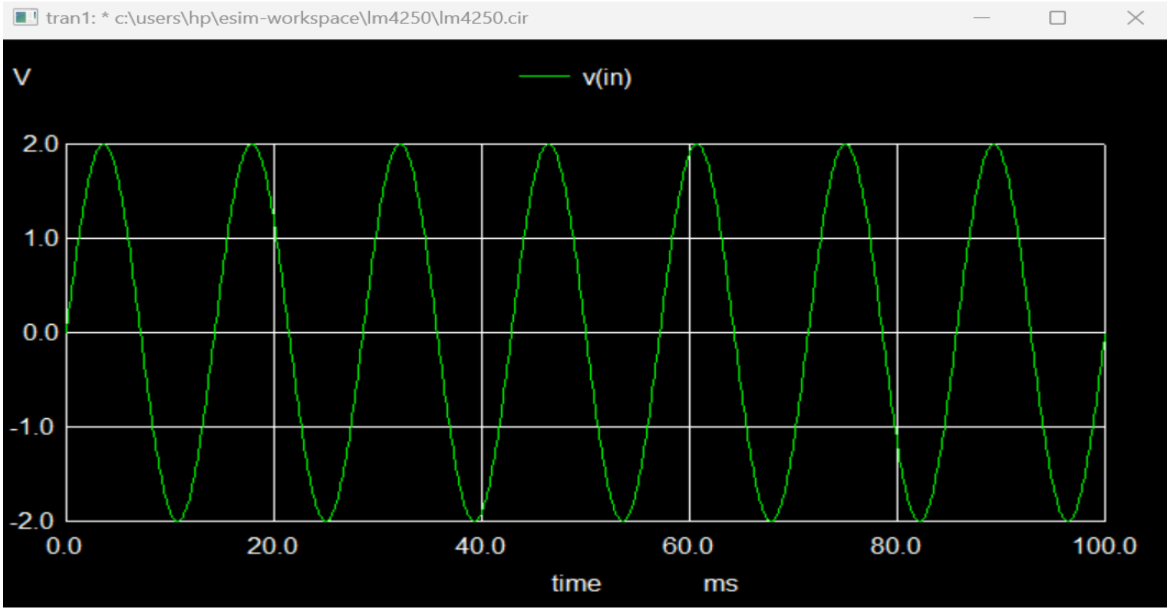

Figure 4.44: Input Voltage Waveform of LM4250

4.9.5 Output Plots

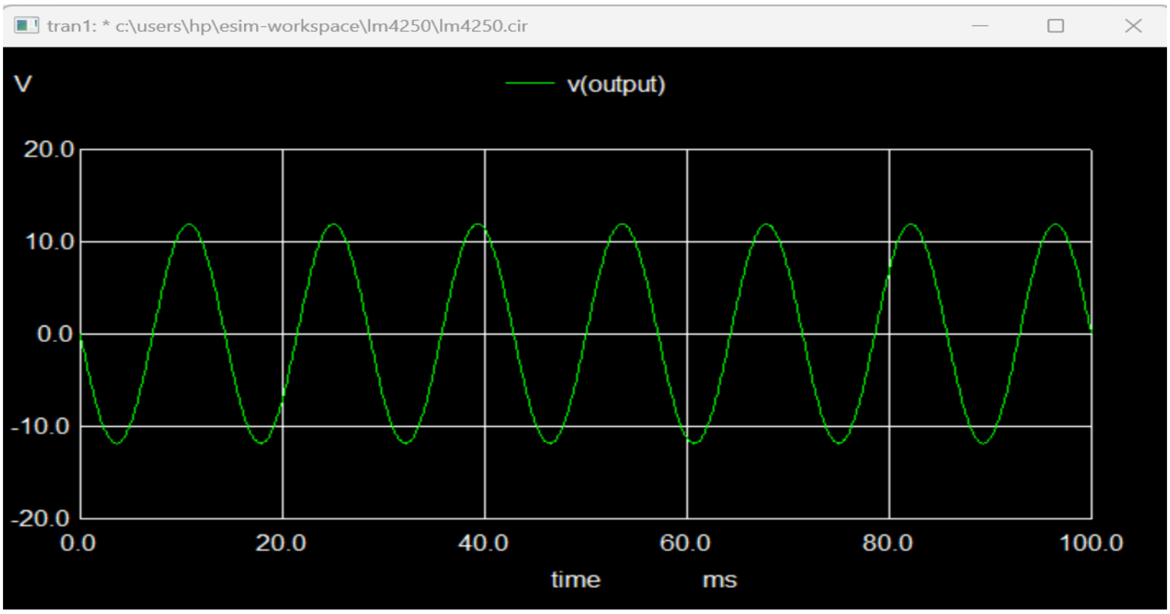

Figure 4.45: Output Voltage Waveform of LM4250

4.10 AS321-Low Power Single Operational Amplifier

The AS321 is a low-power single operational amplifier specifically engineered for applications where minimizing power usage is essential. The AS321 is employed in various analog signal processing tasks such as filtering, amplifying, and conditioning signals.

Key Features:

- **Efficient Power Usage:** It is designed to consume minimal power, making it ideal for use in battery-operated and portable devices.

- **Single Supply Compatibility:** Operates from a single power source, which simplifies the design of the circuit.

- **Broad Input Voltage Range:** Handles a wide range of input voltages.

- **Minimal Offset Voltage:** Ensures precise and stable operation with low offset errors.

- **High Gain Performance:** Delivers substantial gain for effective signal amplification.

4.10.1 IC Layout

This figure represents the 5-Pin Package Diagram of the Low Power Single Operational Amplifier

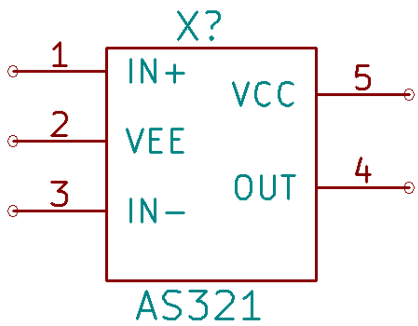

Figure 4.46: AS321-Low Power Single Operational Amplifier

4.10.2 Subcircuit Schematic Diagram

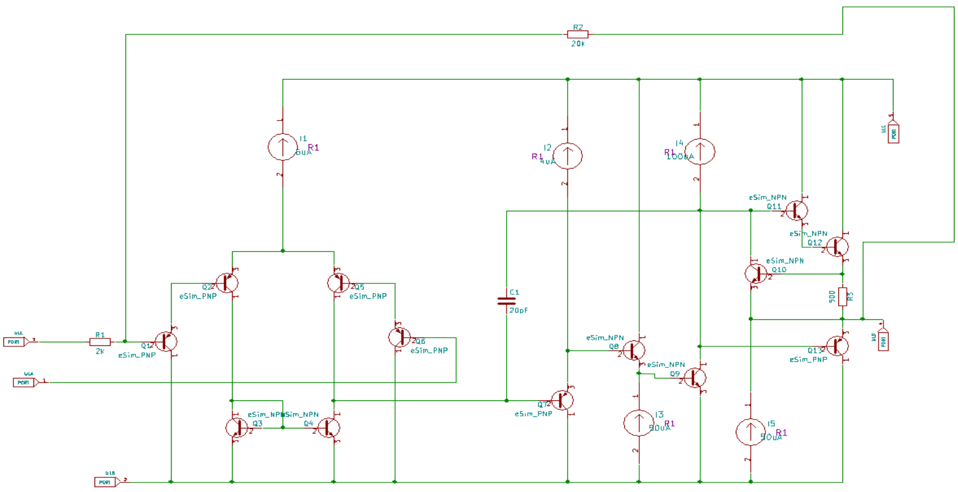

Figure 4.47: Subcircuit Schematic of AS321

4.10.3 Test Circuit

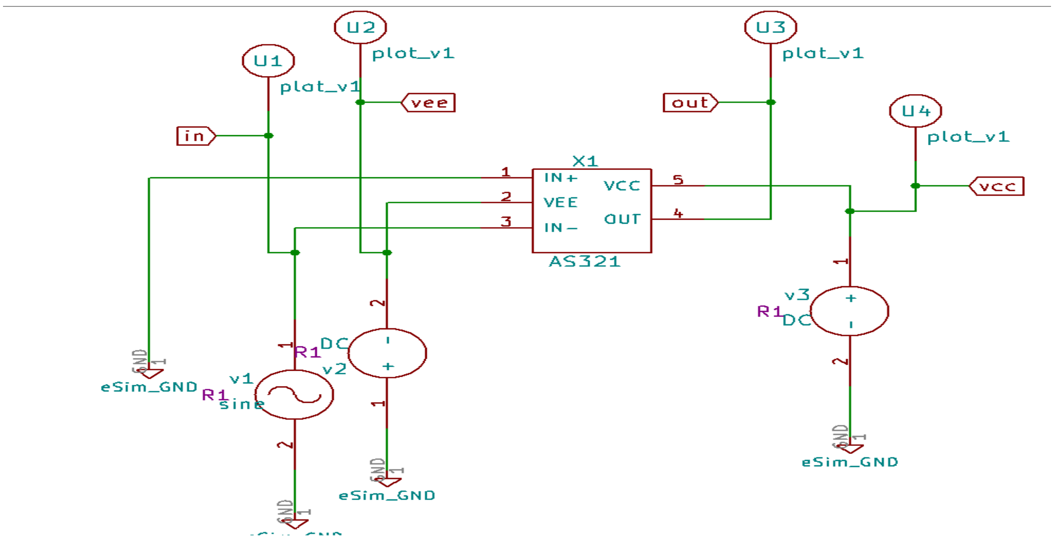

Figure 4.48: Test Circuit of AS321 IC

4.10.4 Input Plots

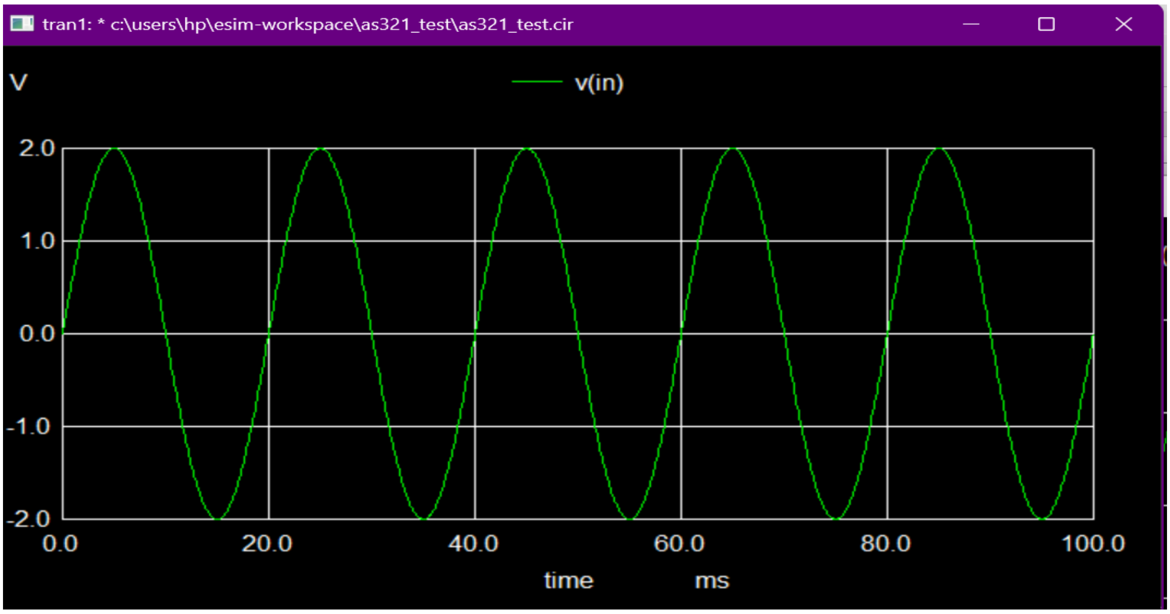

Figure 4.49: Input Voltage Waveform of AS321

4.10.5 Output Plots

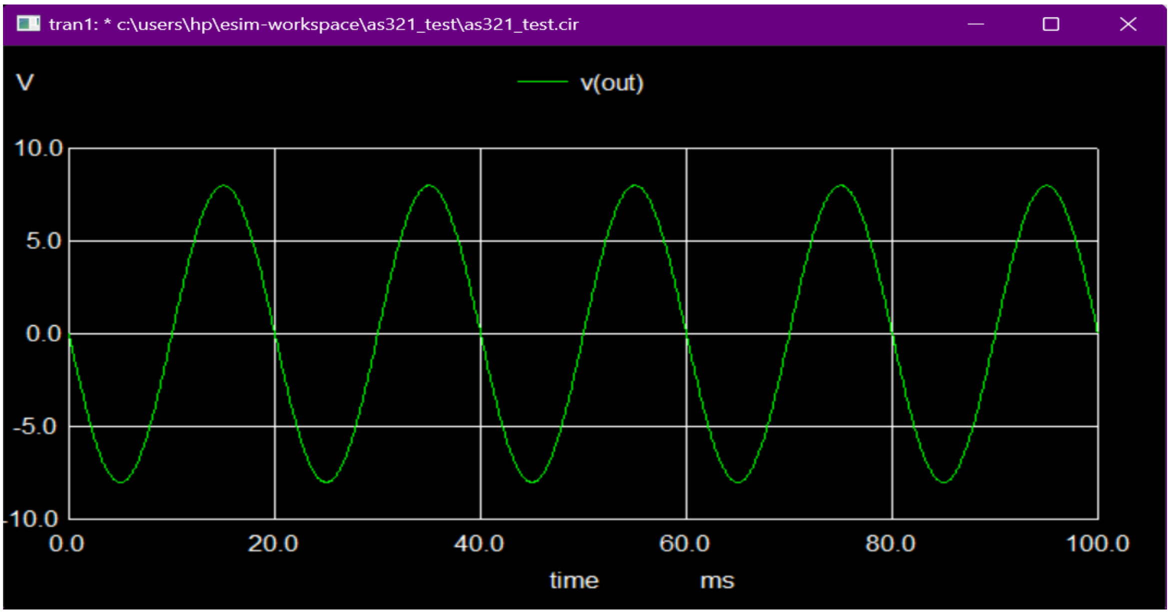

Figure 4.50: Output Voltage Waveform of AS321

Chapter 5

Conclusion and Future Scope

We successfully met our goal of developing various subcircuits for both Analog and Digital Integrated Circuits. Each IC model was meticulously created based on the specifications provided in their official datasheets. The output of each IC was thoroughly verified and tested using their respective test circuits. The IC models developed during this fellowship include fundamental circuit components such as Op-Amps, Voltage Regulators, Precision Rectifiers, Schmitt Triggers, Differential Amplifiers, Instrumentation Amplifiers, Comparators, Multiplexers, De-Multiplexers, and various Logic Gate ICs. These models are now ready to be incorporated into the subcircuit library of eSim, allowing developers and students to utilize them as building blocks in their projects and circuit designs. As the device model library in eSim continues to grow, we anticipate the development of more such ready-to-use IC models for use in eSim.

Bibliography

[1] FOSSEE Official Website
https://fossee.in/

2] eSim Official website.
URL:https://esim.fossee.in/

[3] Datasheet Catalog.
URL:https://www.datasheetcatalog.com/

[4] ALLDATASHEET.COM.
URL:https://www.alldatasheet.com/

[5] Mouser.in.
URL:https://www.mouser.in/

[6] Texas Instruments.
URL:https://www.ti.com/

[7] NXP.
URL:https://www.nxp.com/design/documentation:DOCUMENTATION#/

[8] Analog Devices.
URL:https://www.analog.com/en/product-category.html

[9] Datasheet4u National Semiconductor.
URL:https://www.datasheet4u.com/Manufacture/National%20Semiconductor.php?c=7

[10] STMicroelectronics .
URL:https://www.st.com/resource/en/datasheet/