Summer Fellowship Report

On

Integrated Circuit Design using Subcircuit feature of eSim

Submitted by

Shweta Umesh Khune
Electrical Engineering Department
SGGSIE&T

Under the guidance of

Prof.Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

March 12, 2025

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 and Mrs. Usha Vishwanathan, Mrs.Vineeta Ghavri 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 LM60 - Temperature Sensor

The LM60 is a precision temperature sensor IC that operates from 2.7V to 10V and provides an output voltage linearly proportional to temperature. It has a sensitivity of 6.25mV/°C with a 424mV offset at 0°C, allowing it to measure negative temperatures without a negative supply.

The LM60 temperature sensor IC is used in various applications where accurate temperature measurement is needed. It is used in mobile phones, laptops, and battery management systems to monitor temperature for safety. It is also used in power supply modules to prevent overheating. In home appliances and HVAC systems, it helps in temperature control.

The LM60 IC has low power consumption (70µA–110µA), making it ideal for battery-operated devices. It provides accurate temperature readings over a wide range (-40°C to 125°C) without needing a negative supply. Its linear output (6.25mV/°C) with a 424mV offset simplifies interfacing with microcontrollers and ADCs.

4.1.1 IC Layout

This figure represents the 3-Pin Package Diagram of the LM60 Temperature Sensor

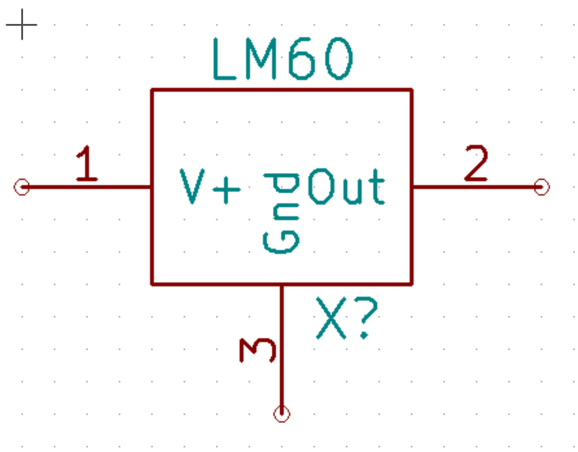

Figure 4.1: LM60 Temperature Sensor

4.1.2 Subcircuit Schematic Diagram

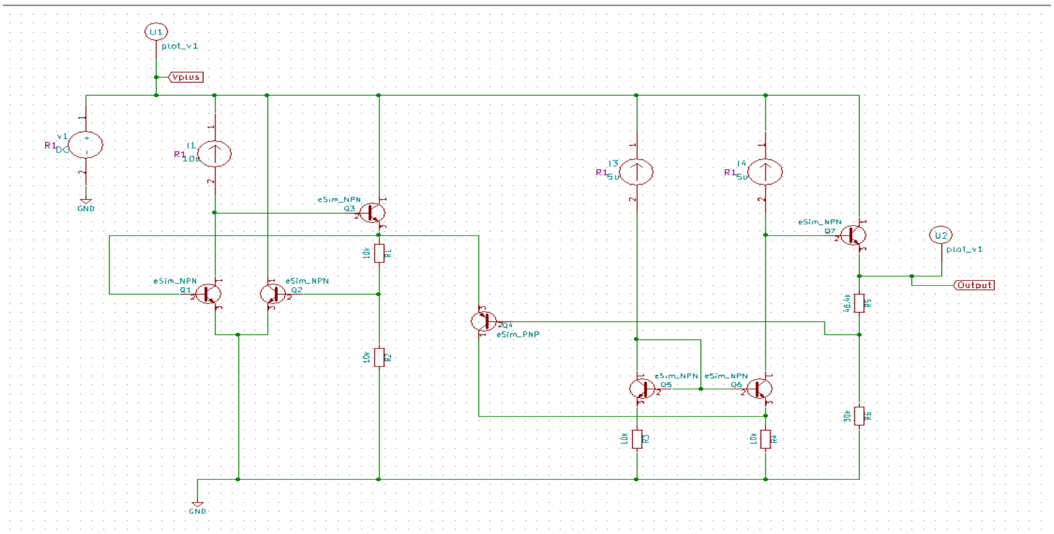

Figure 4.2: Subcircuit Schematic of LM60

4.1.3 Test Circuit

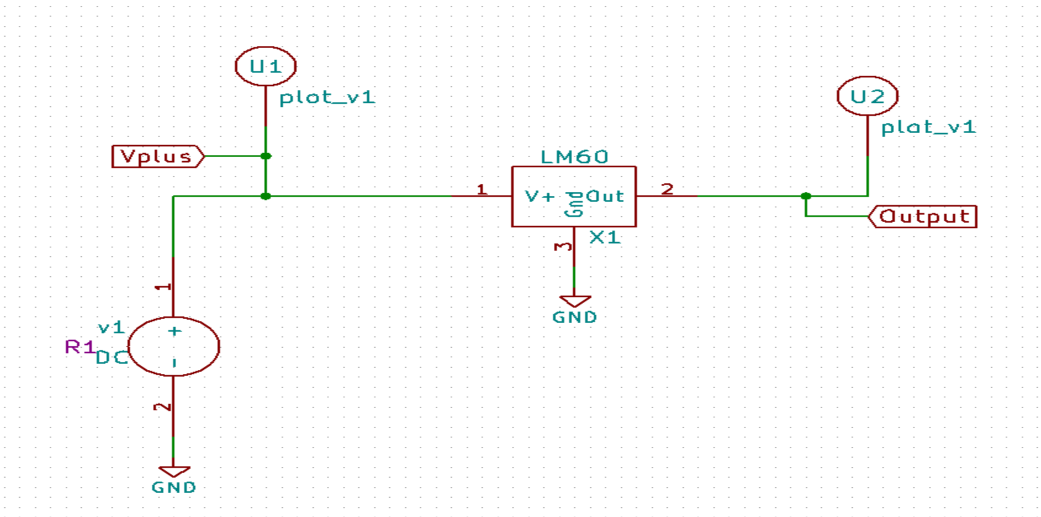

Figure 4.3: Test Circuit of LM60 IC

4.1.4 Input Plots

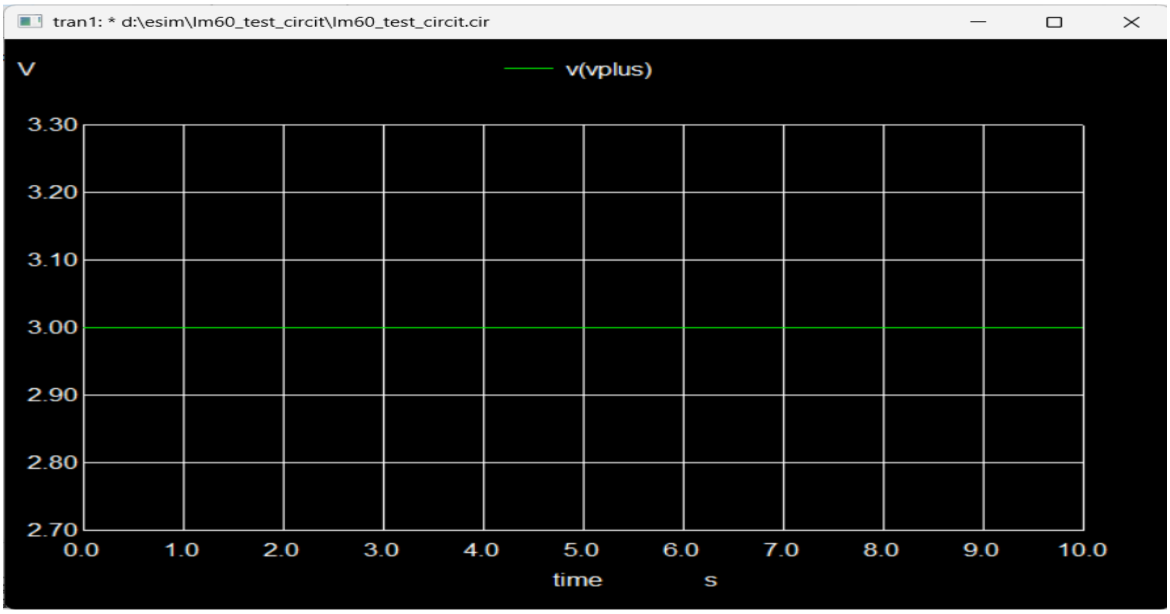

Figure 4.4: Input Voltage Waveform of LM60

4.1.5 Output Plots

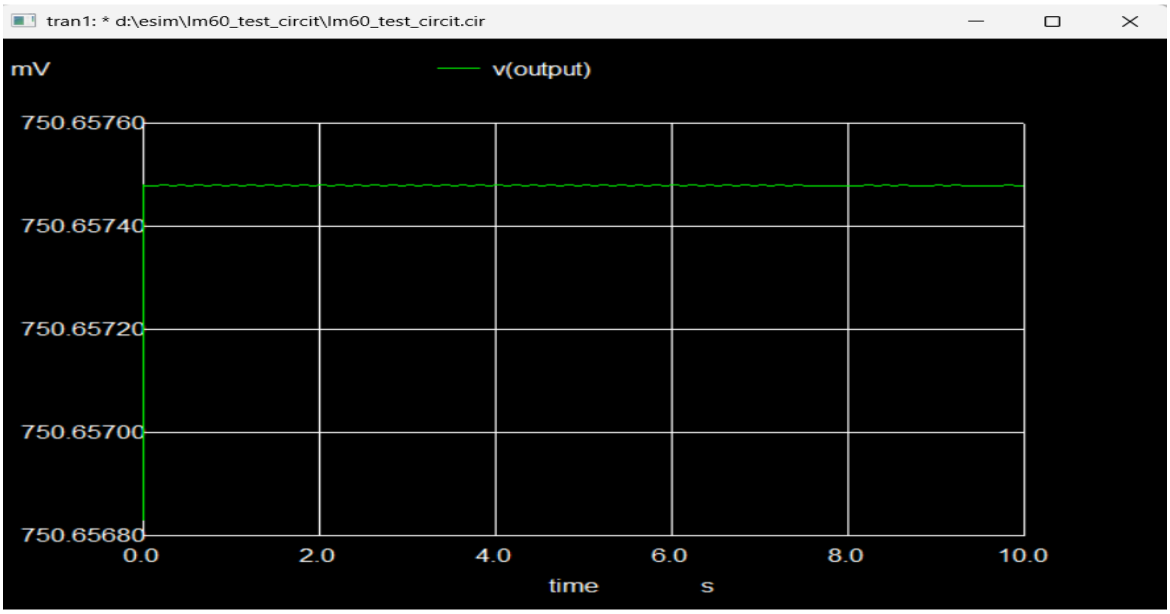

Figure 4.5: Output Voltage Waveform of LM60

4.2 LM324 - Quad Operational Amplifier

The LM324 is a quad operational amplifier (op-amp) IC that consists of four independent op-amps in a single package. It operates over a wide voltage range (3V to 36V) and is suitable for low-power applications with a low quiescent current of 240µA per amplifier.

LM324 is used in power supplies, chargers, and electronic devices like computers and printers. It is also found in air conditioners, refrigerators, and washing machines for control circuits. In industries, it is used in motor control, automation, and signal processing.

LM324 works with a wide voltage range (3V to 36V) and consumes very low power (240µA per amplifier), making it energy-efficient. It has built-in EMI and RF filters to reduce noise and improve stability. The input voltage can go down to ground (0V), allowing easy single-supply operation. It also has high ESD protection, making it reliable in harsh conditions. Additionally, it can directly replace older LM324 versions for better performance.

4.2.1 IC Layout

This figure represents the 14-Pin Package Diagram of the LM324 Quad Operational Amplifier

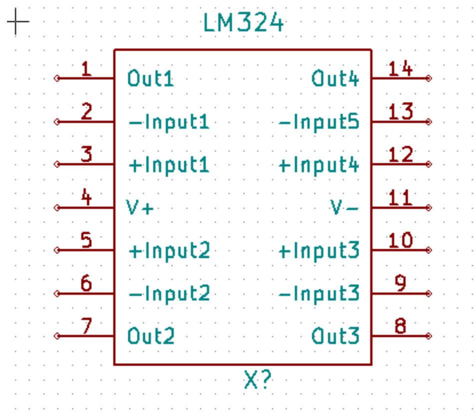

Figure 4.6: LM324 Quad Operational Amplifier

4.2.2 Subcircuit Schematic Diagram

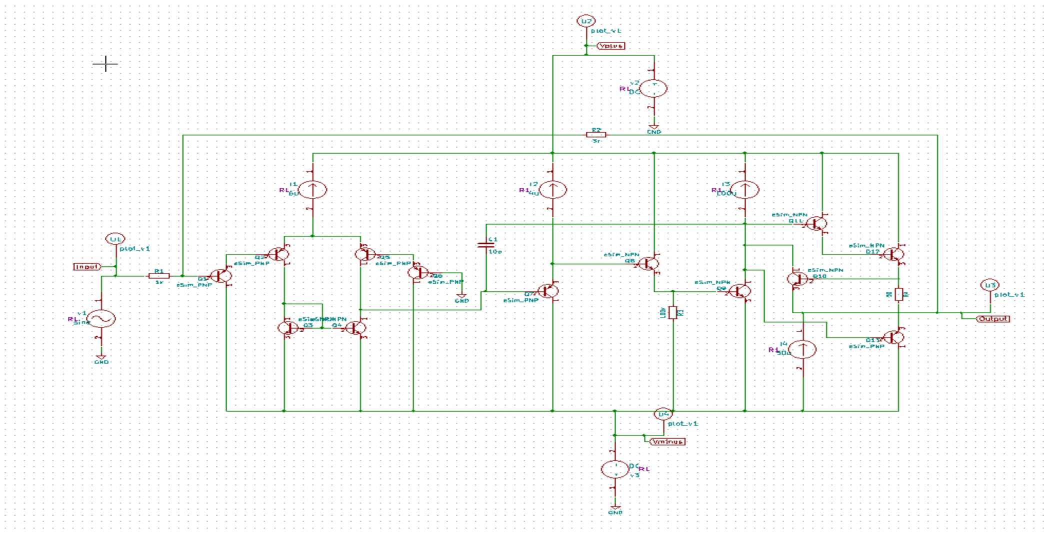

Figure 4.7: Subcircuit Schematic of LM324

4.2.3 Test Circuit

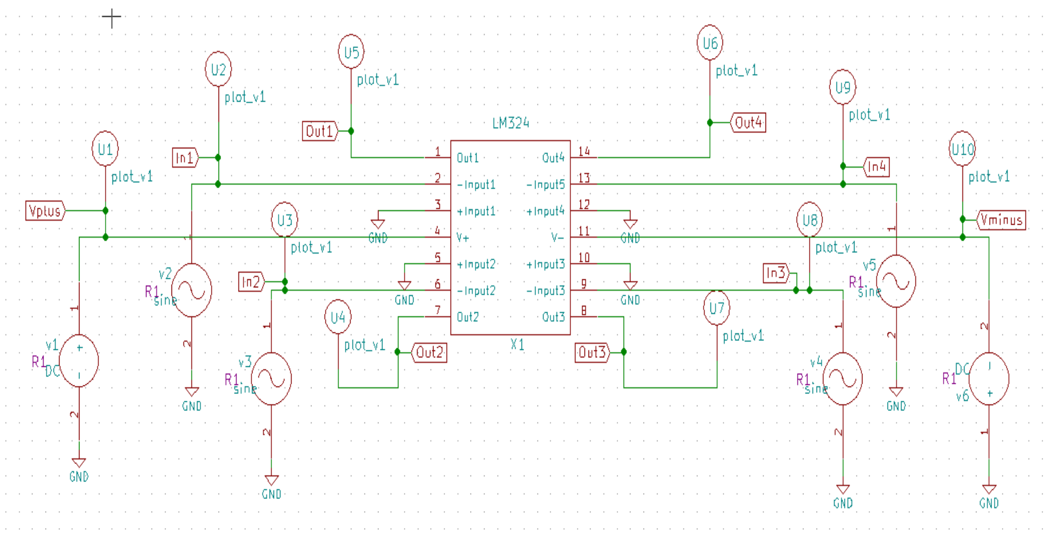

Figure 4.8: Test Circuit of LM324 IC

4.2.4 Input Plots

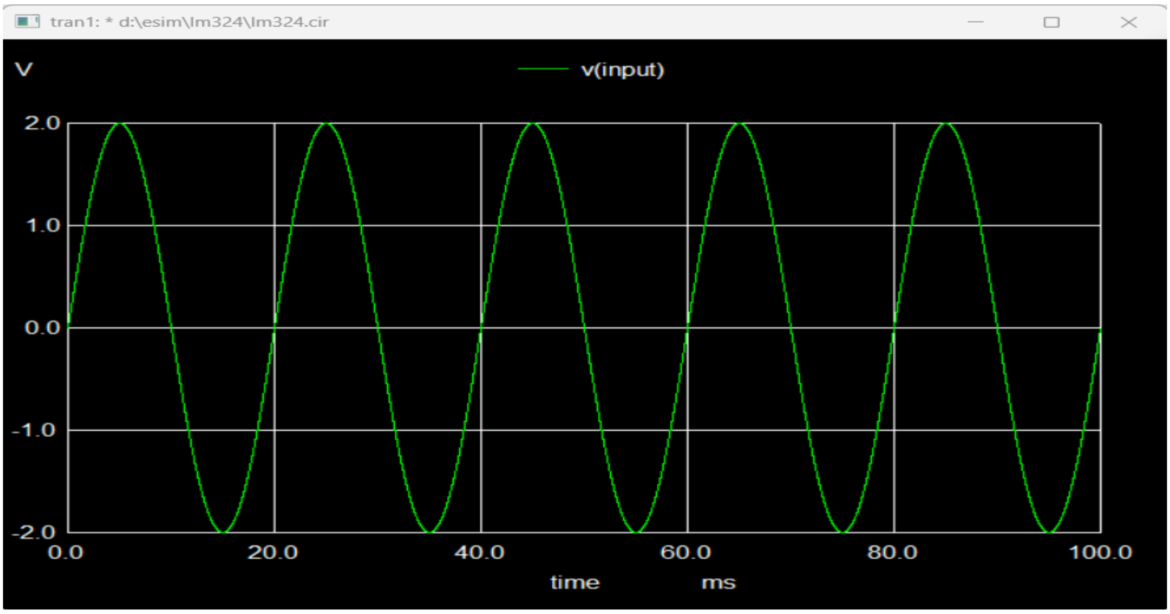

Figure 4.9: Input Voltage Waveform of LM324

4.2.5 Output Plots

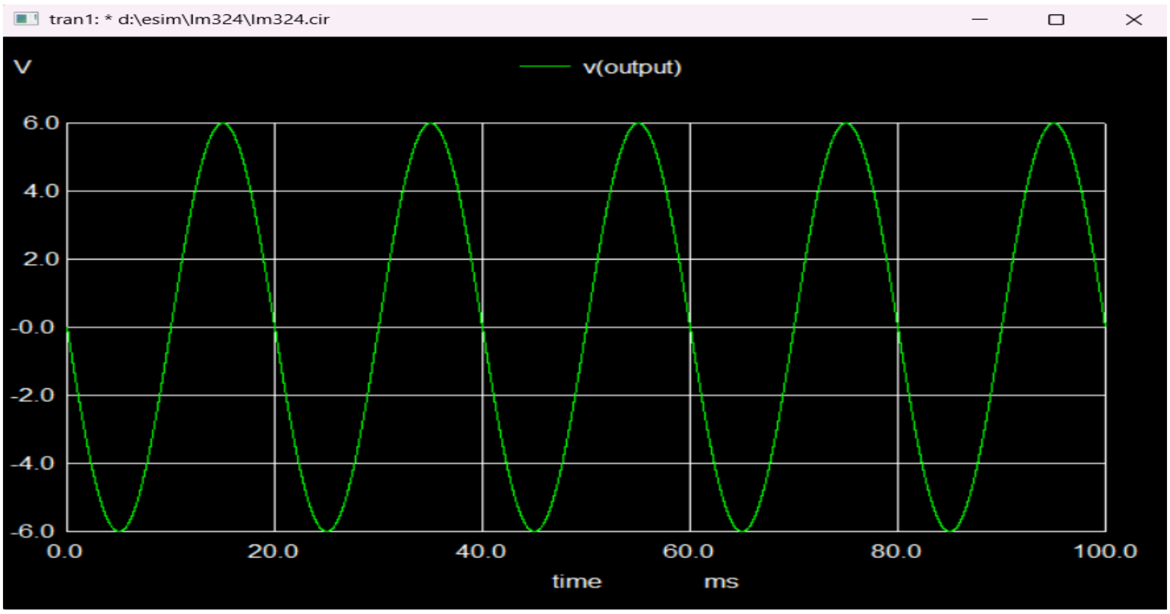

Figure 4.10: Output Voltage Waveform of LM324

4.3 TMP35 - Temperature Sensor

The TMP35 is a low-power, precision temperature sensor that gives an output voltage directly proportional to temperature in degrees Celsius. It operates from 2.7V to 5.5V and provides a 250 mV output at 25°C, covering a temperature range of 10°C to 125°C. The sensor requires no calibration, has low power consumption, and is ideal for temperature monitoring in various applications.

The TMP35 IC is used in various temperature-sensing applications. It helps in HVAC systems to regulate heating and cooling based on room temperature. In industrial machines, it ensures safe operation by preventing overheating. It is also used in computers and power systems to monitor and control temperature, improving efficiency and longevity.

The TMP35 IC has several advantages. It works on a low voltage (2.7V to 5.5V), making it energy-efficient. It provides a linear and accurate temperature output without needing extra calibration. The sensor has low power consumption and generates minimal heat, making it ideal for sensitive applications. Plus, it is easy to interface with microcontrollers and ADCs.

4.3.1 IC Layout

This figure represents the 3-Pin Package Diagram of the TMP35 Temperature Sensor

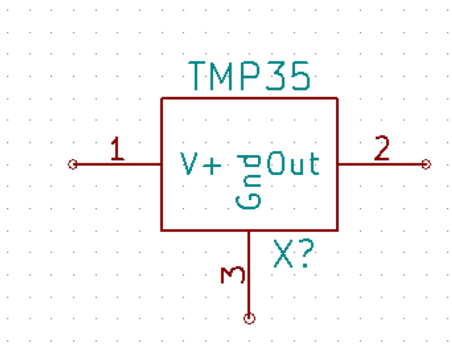

Figure 4.11: TMP35 Temperature Sensor

4.3.2 Subcircuit Schematic Diagram

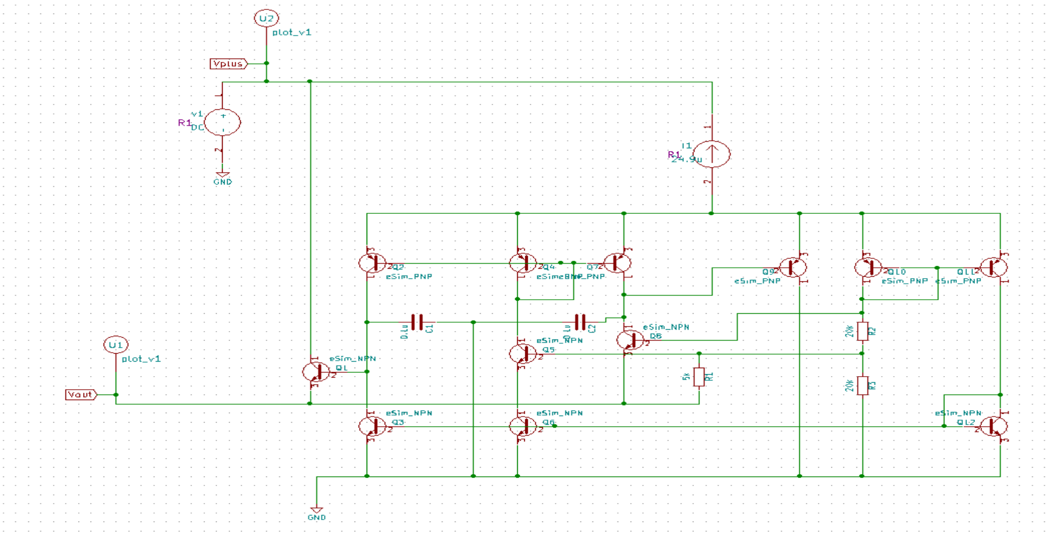

Figure 4.12: Subcircuit Schematic of TMP35

4.3.3 Test Circuit

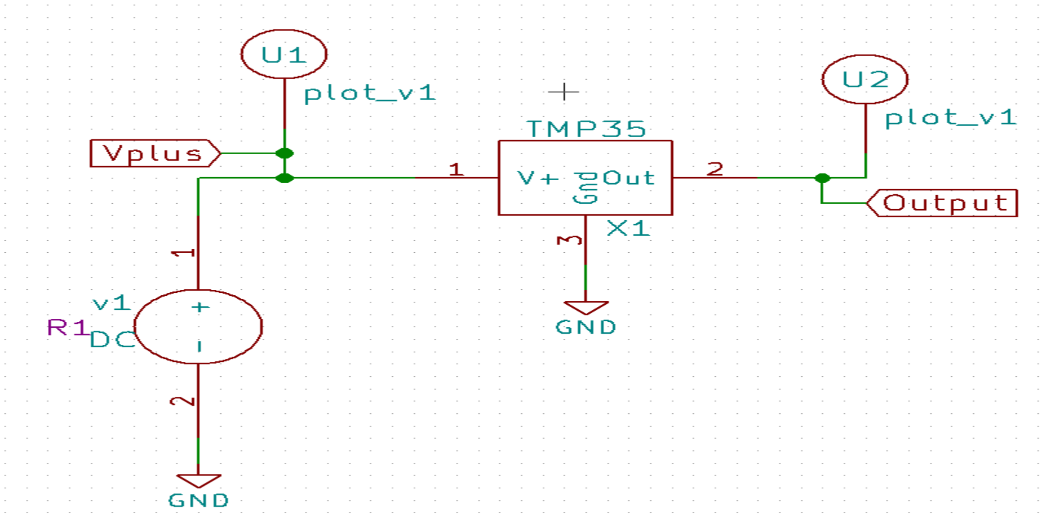

Figure 4.13: Test Circuit of TMP35 IC

4.3.4 Input Plots

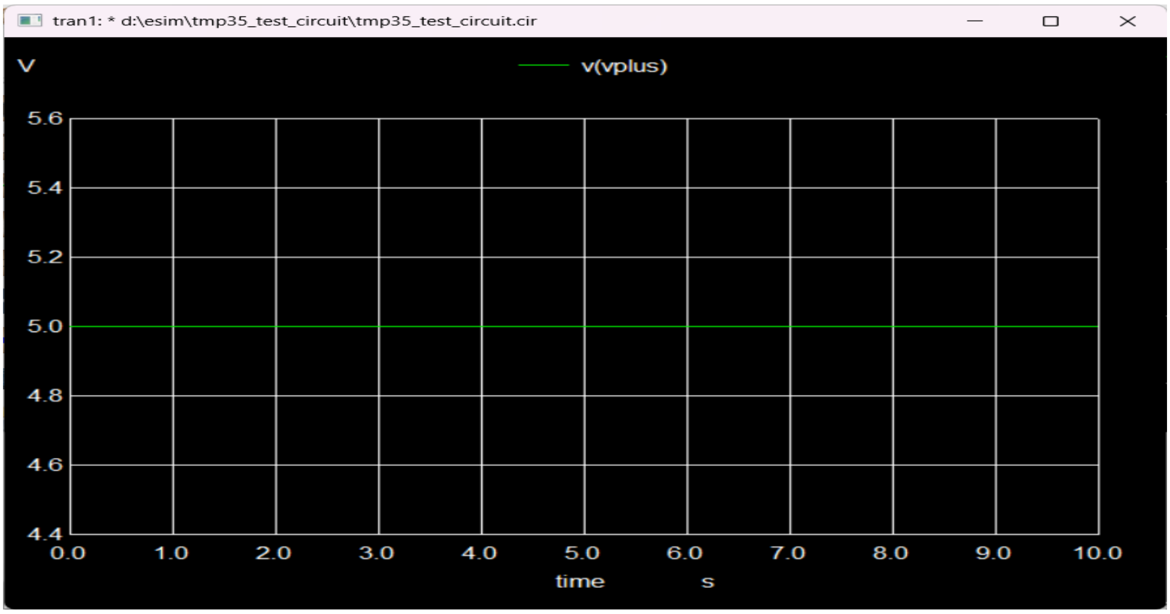

Figure 4.14: Input Voltage Waveform of TMP35

4.3.5 Output Plots

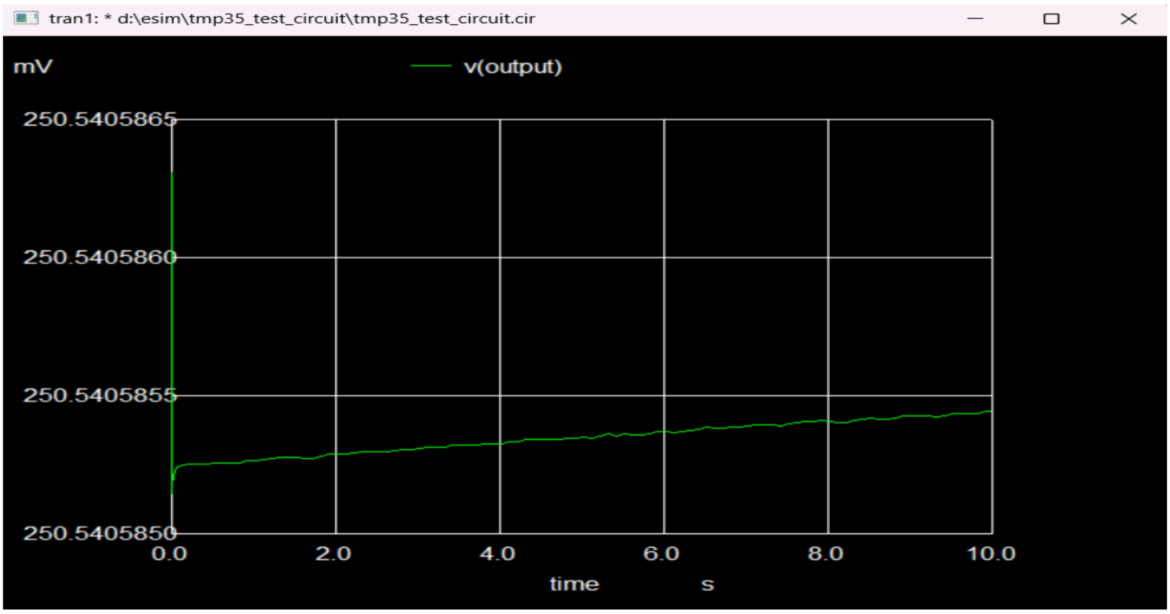

Figure 4.15: Output Voltage Waveform of TMP35

4.4 LM7815 - Voltage Regulator

The LM7815 is a fixed 15V voltage regulator that provides a stable output voltage with a maximum current of 1A. It has built-in protection against overheating, short circuits, and excessive power dissipation. The IC requires minimal external components and is commonly used in power supplies for electronic circuits.

The LM7815 is used in power supply circuits to provide a stable 15V output for electronic devices. It is commonly found in audio amplifiers, microcontroller circuits, industrial automation, and instrumentation systems. It helps regulate voltage for logic circuits and sensors, ensuring proper operation without fluctuations.

Advantages of the LM7815 voltage regulator provides a stable 15V output, ensuring reliable operation for various electronic circuits. It has built-in thermal shutdown, preventing overheating, and short-circuit protection, which safeguards connected components. Its current limiting feature prevents excessive current flow, reducing the risk of damage. The IC requires minimal external components, making circuit design simple and efficient. Additionally, it offers high efficiency and durability, making it ideal for long-term applications.

4.4.1 IC Layout

This figure represents the 3-Pin Package Diagram of the LM7815 voltage regulator

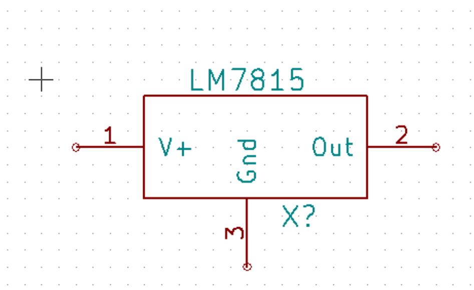

Figure 4.16: LM7815 voltage regulator

4.4.2 Subcircuit Schematic Diagram

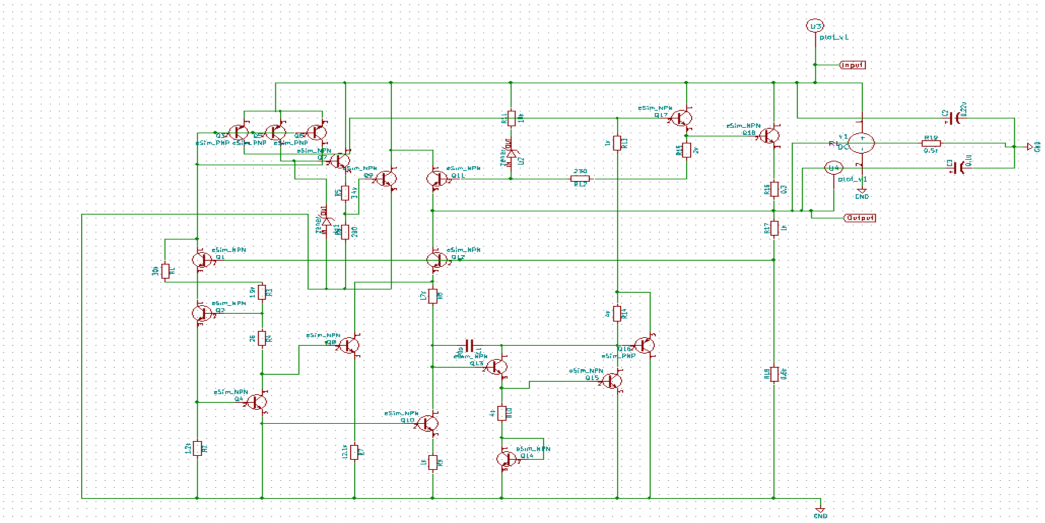

Figure 4.17: Subcircuit Schematic of LM7815

4.4.3 Test Circuit

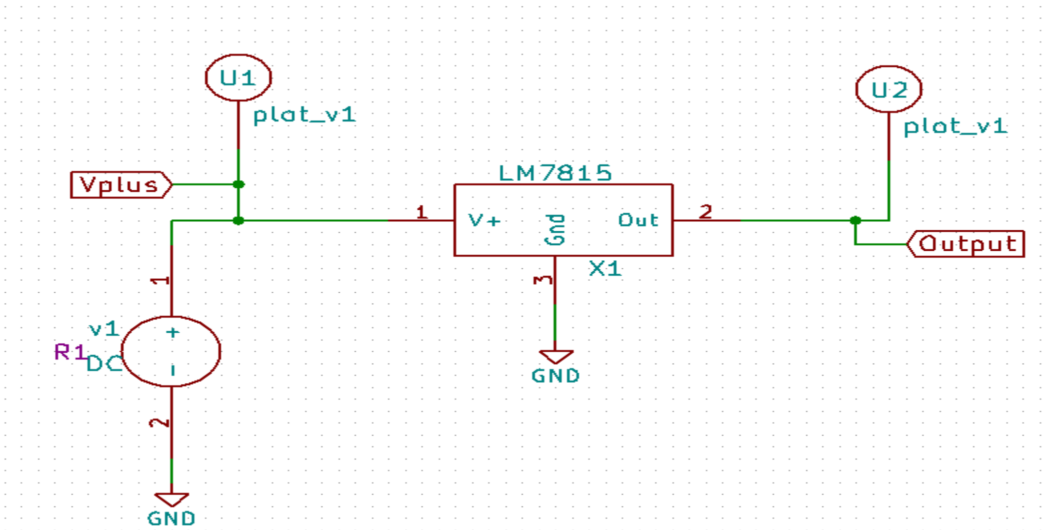

Figure 4.18: Test Circuit of LM7815 IC

4.4.4 Input Plots

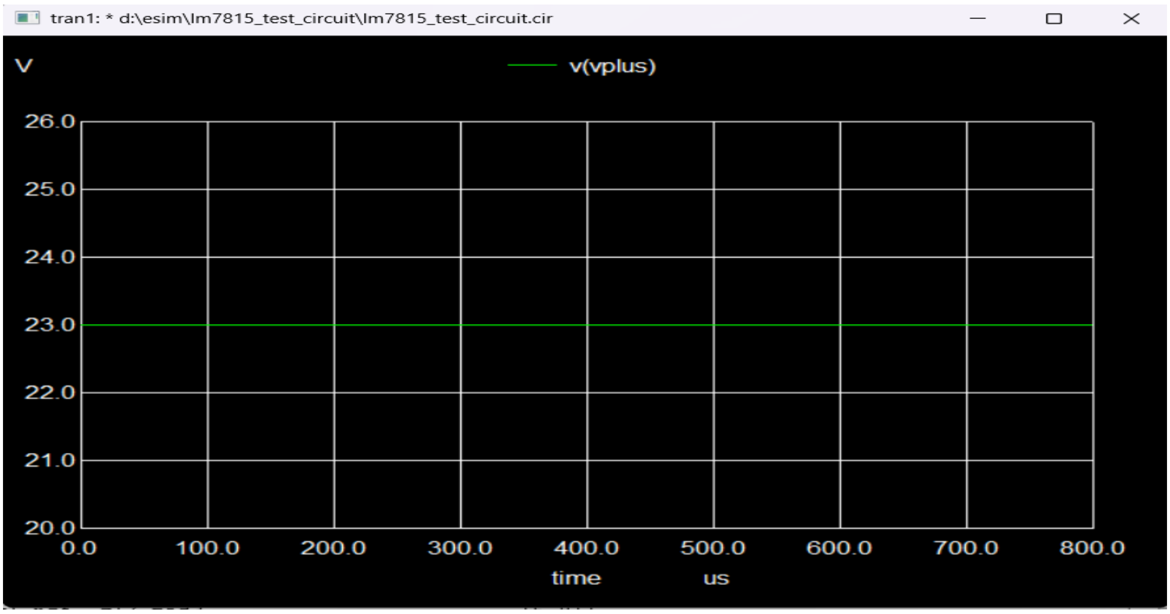

Figure 4.19: Input Voltage Waveform of LM7815

4.4.5 Output Plots

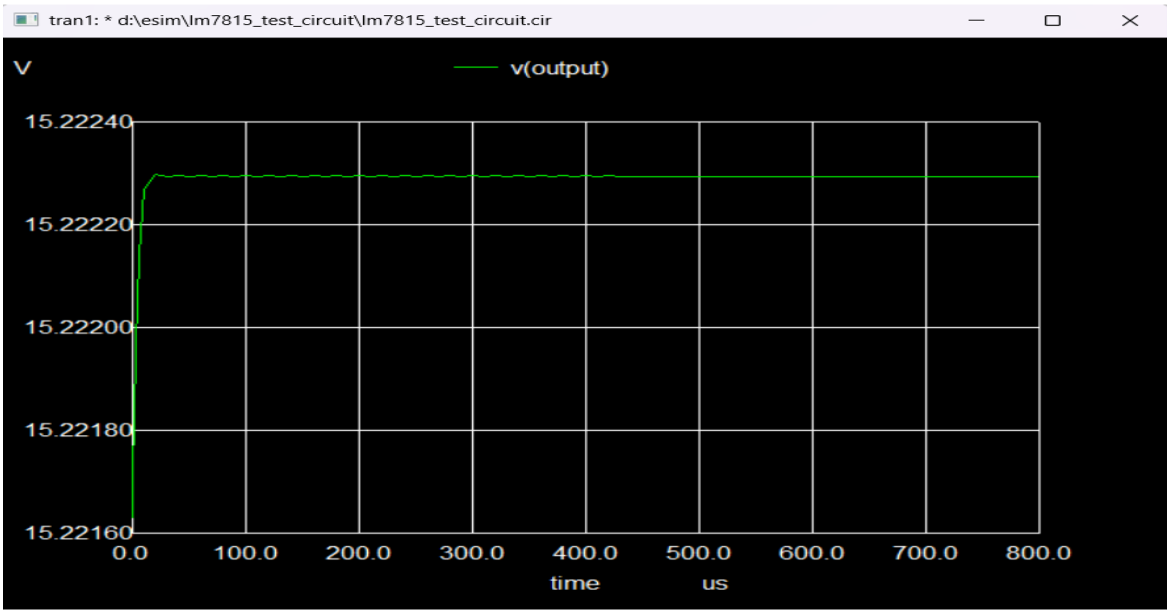

Figure 4.20: Output Voltage Waveform of LM7815

4.5 LM614 - Quad Operational Amplifier

The LM614 is an integrated circuit that includes four operational amplifiers (op-amps) and a programmable voltage reference in a single 16-pin package. It operates with a low current of 450µA and supports a wide supply voltage range from 4V to 36V. The op-amps provide high speed and bandwidth while consuming minimal power, making them suitable for single-supply applications. The built-in voltage reference can be adjusted from 1.2V to 5.0V, ensuring stable operation for signal processing, transducer systems, and power monitoring applications. It is designed to reduce cost and board space while maintaining high performance.

Applications of the LM614 IC is used in transducer bridge drivers and signal processing systems where stable and precise voltage is needed. It is also applied in process and mass flow control systems to monitor and regulate signals. Additionally, it is useful in power supply voltage monitoring and as a buffered voltage reference for A/D converters, ensuring accurate data acquisition in measurement systems.

Advantages of the LM614 IC has low power consumption (450µA), making it energy-efficient. It operates over a wide voltage range (4V to 36V) and provides a stable output even with large capacitive loads. Its built-in voltage reference (adjustable from 1.2V to 5V) ensures precise signal processing. Additionally, it integrates four high-performance op-amps, reducing the need for extra components and saving board space.

4.5.1 IC Layout

This figure represents the 14-Pin Package Diagram of the LM614 Quad Operational Amplifier

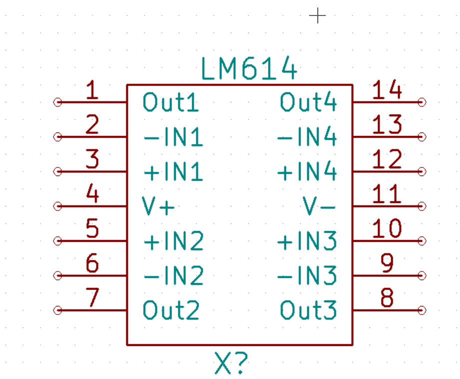

Figure 4.21: LM614 Quad Operational Amplifier

4.5.2 Subcircuit Schematic Diagram

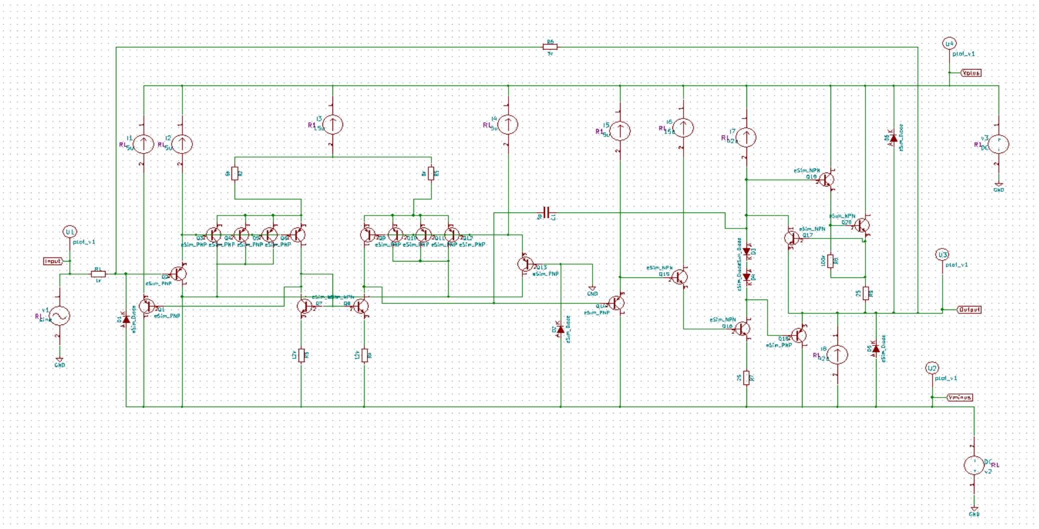

Figure 4.22: Subcircuit Schematic of LM614

4.5.3 Test Circuit

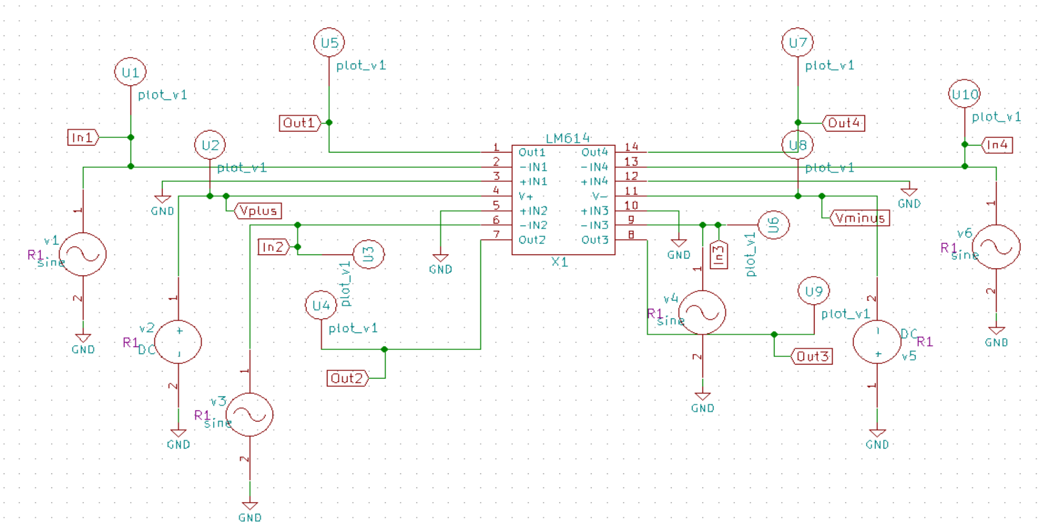

Figure 4.23: Test Circuit of LM614 IC

4.5.4 Input Plots

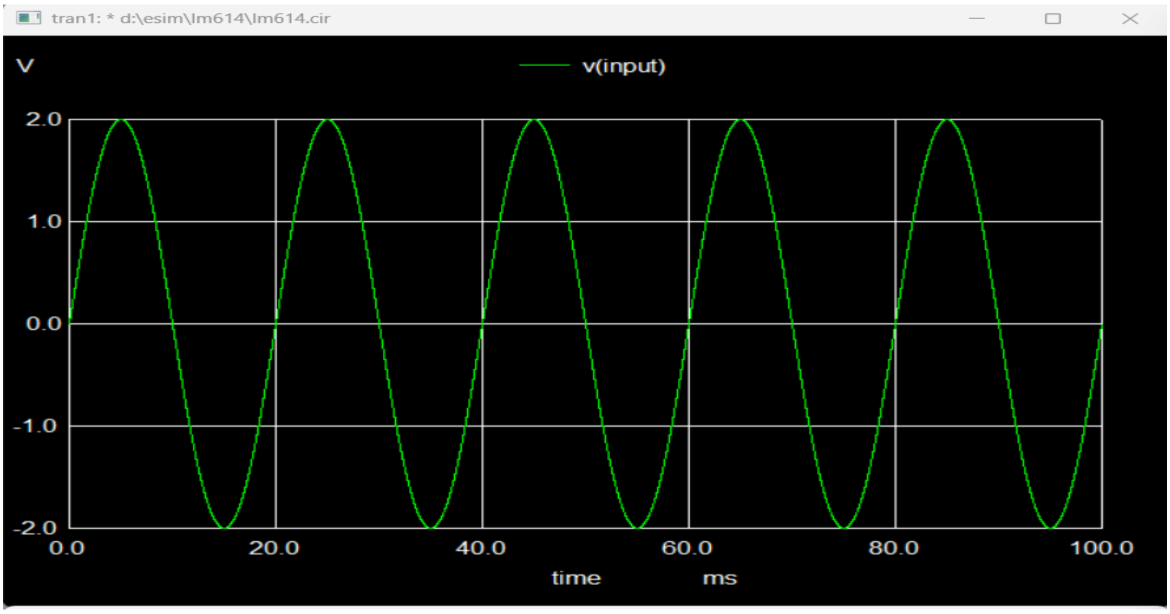

Figure 4.24: Input Voltage Waveform of LM614

4.5.5 Output Plots

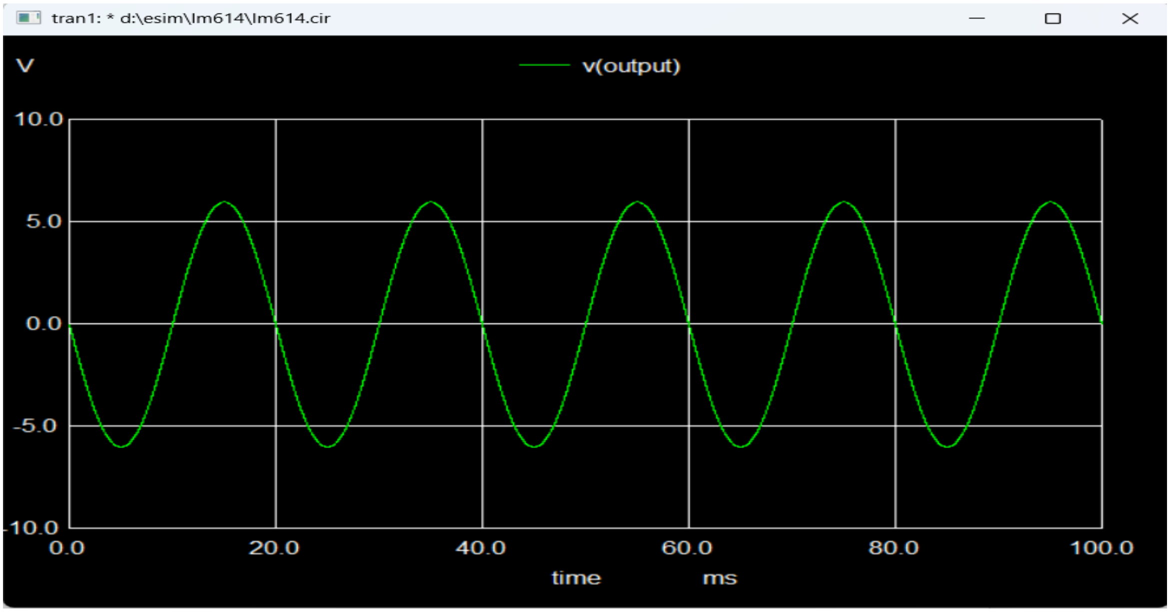

Figure 4.25: Output Voltage Waveform of LM614

4.6 LM833 - Dual Operational Amplifier

The LM833 is a high-performance dual op-amp designed mainly for audio applications. It has low noise (4.5nV/Hz), a high slew rate (7V/s), and a wide bandwidth (15MHz), making it ideal for HiFi and PCM systems. The IC provides low distortion and low offset voltage (0.3mV) for precise signal amplification. It is internally compensated, ensuring stable performance across different gain settings.

Applications of the LM833 is used in high-quality audio systems, such as HiFi amplifiers, preamplifiers, and mixers, due to its low noise and distortion. It is ideal for PCM systems, professional audio equipment, and active filters where clean and accurate signal amplification is needed. Its high-speed performance also makes it suitable for low-level signal processing in instrumentation and communication systems.

Advantages of the LM833 is a low-noise, low-distortion dual operational amplifier, making it perfect for high-fidelity audio applications. It has a high slew rate and wide bandwidth, ensuring fast and accurate signal processing. The low offset voltage improves precision in signal amplification. It is internally compensated, making it stable and easy to use without extra components.

4.6.1 IC Layout

This figure represents the 8 -Pin Package Diagram of the LM833 Dual Operational Amplifier

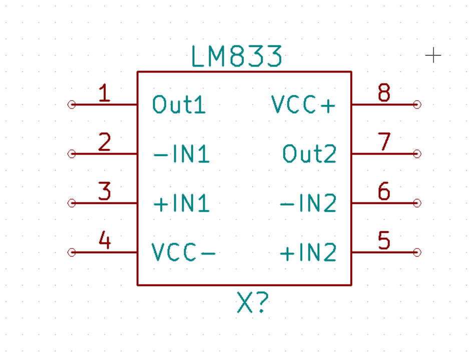

Figure 4.26: LM833 Dual Operational Amplifier

4.6.2 Subcircuit Schematic Diagram

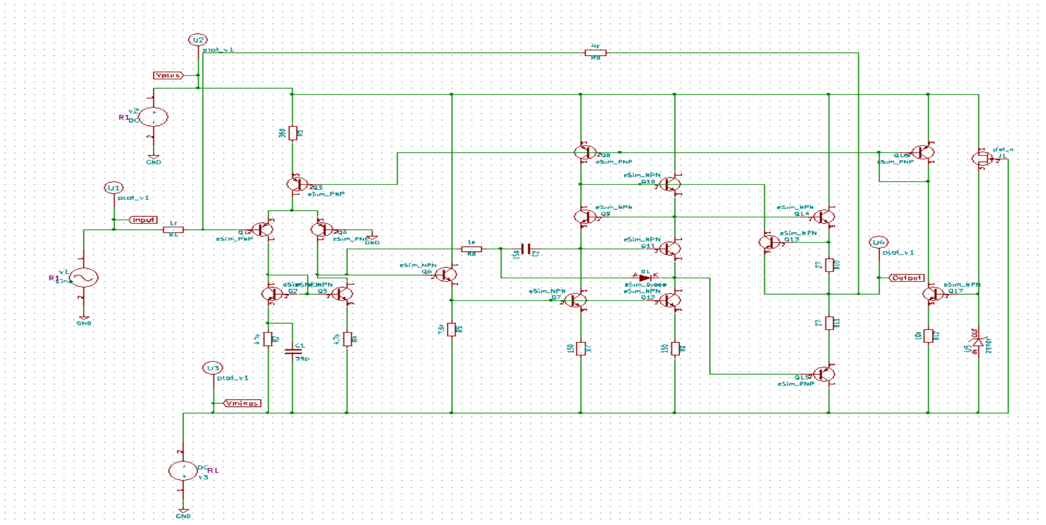

Figure 4.27: Subcircuit Schematic of LM833

4.6.3 Test Circuit

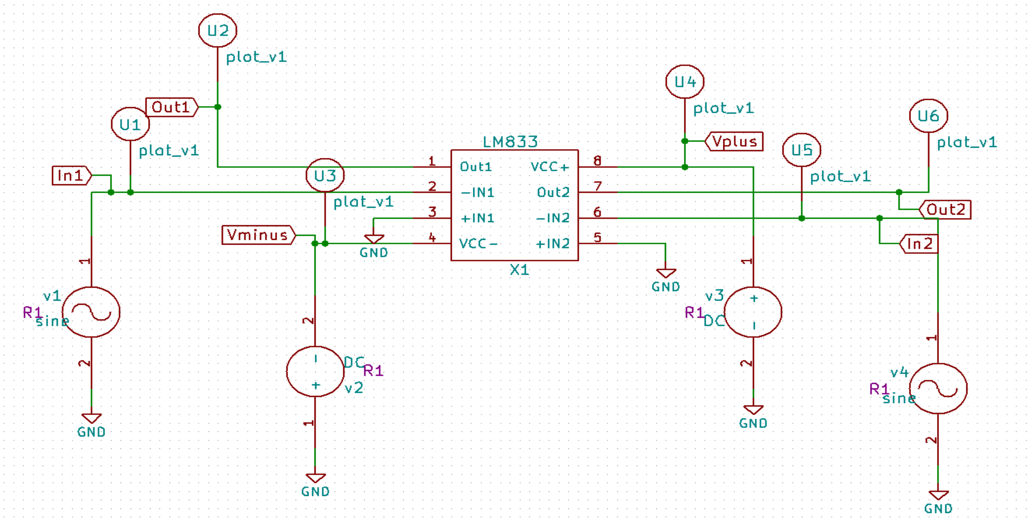

Figure 4.28: Test Circuit of LM833 IC

4.6.4 Input Plots

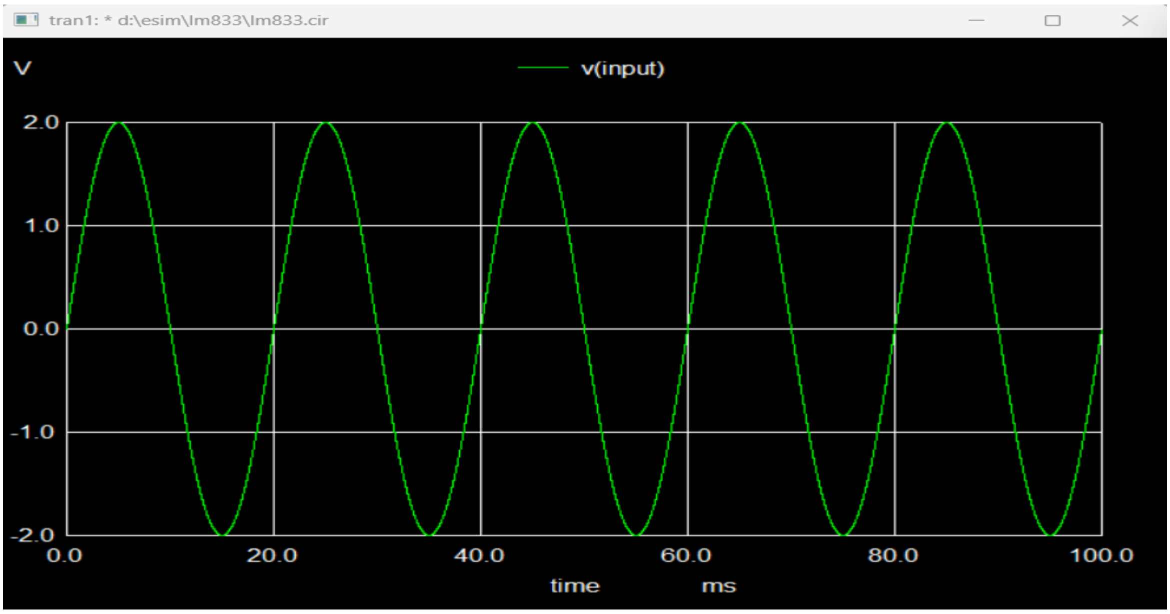

Figure 4.29: Input Voltage Waveform of LM833

4.6.5 Output Plots

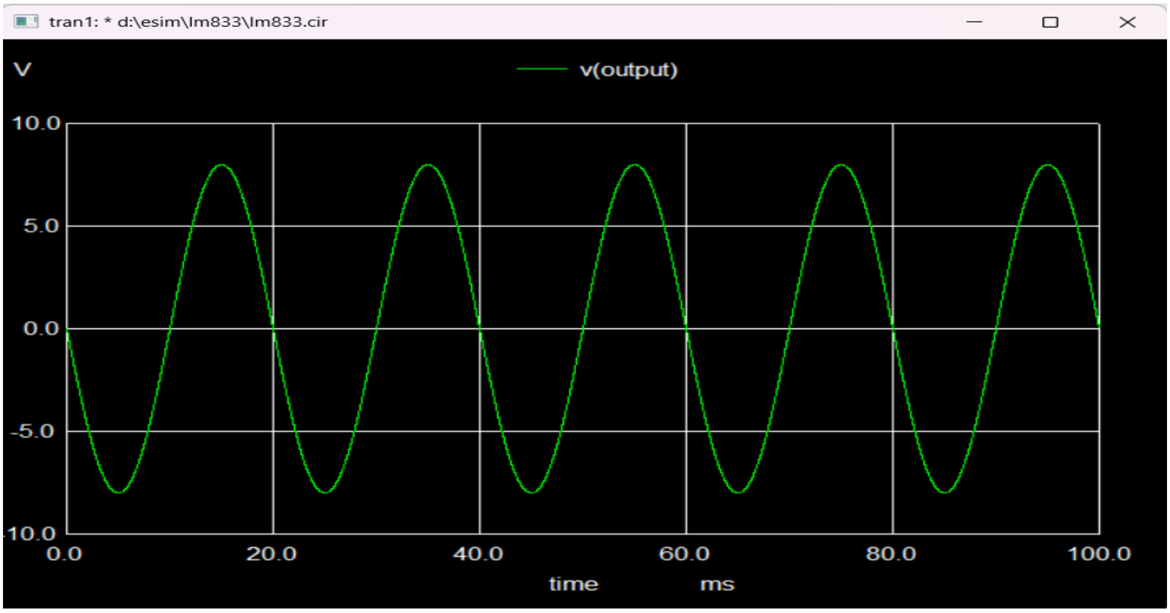

Figure 4.30: Output Voltage Waveform of LM833

4.7 LM338 - Adjustable Regulator

The LM338 is an adjustable voltage regulator IC that can provide up to 5A of output current with an output voltage range of 1.2V to 32V. It requires only two external resistors to set the output voltage and includes thermal overload and short-circuit protection. This IC is commonly used in adjustable power supplies, battery chargers, and constant current regulators.

The LM338 is used in adjustable power supplies to provide a stable voltage output for different electronic circuits. It is commonly used in battery chargers to regulate charging voltage and current. The IC is also ideal for constant current regulators, ensuring a steady current supply for LED drivers and motor control. Additionally, it is used in high-power DC voltage regulators for industrial and automotive applications.

Advantages of the LM338 offers a high output current capacity of up to 5A, making it suitable for power-hungry applications. It provides an adjustable output voltage from 1.2V to 32V with just two external resistors. The IC has thermal overload and short-circuit protection, ensuring safe operation. It also features excellent load and line regulation, making it reliable for stable power supply applications.

4.7.1 IC Layout

This figure represents the 3-Pin Package Diagram of the LM338 Adjustable Regulator

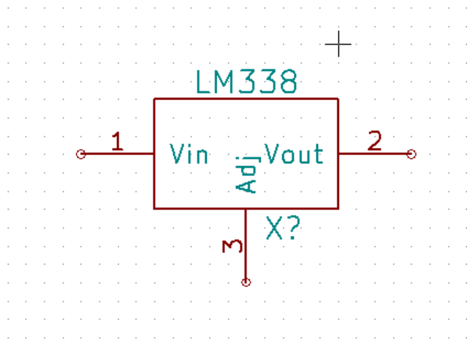

Figure 4.31: LM338 Adjustable Regulator

4.7.2 Subcircuit Schematic Diagram

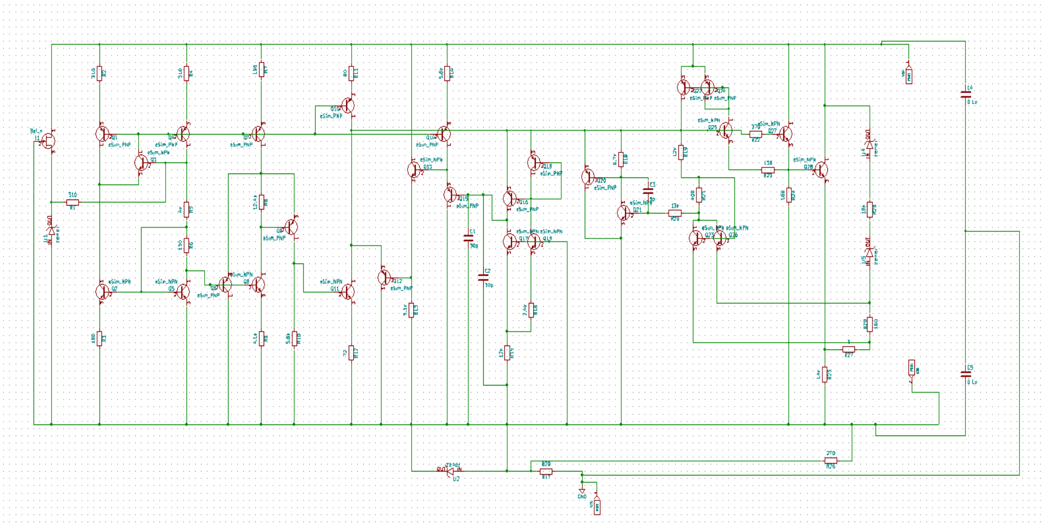

Figure 4.32: Subcircuit Schematic of LM338

4.7.3 Test Circuit

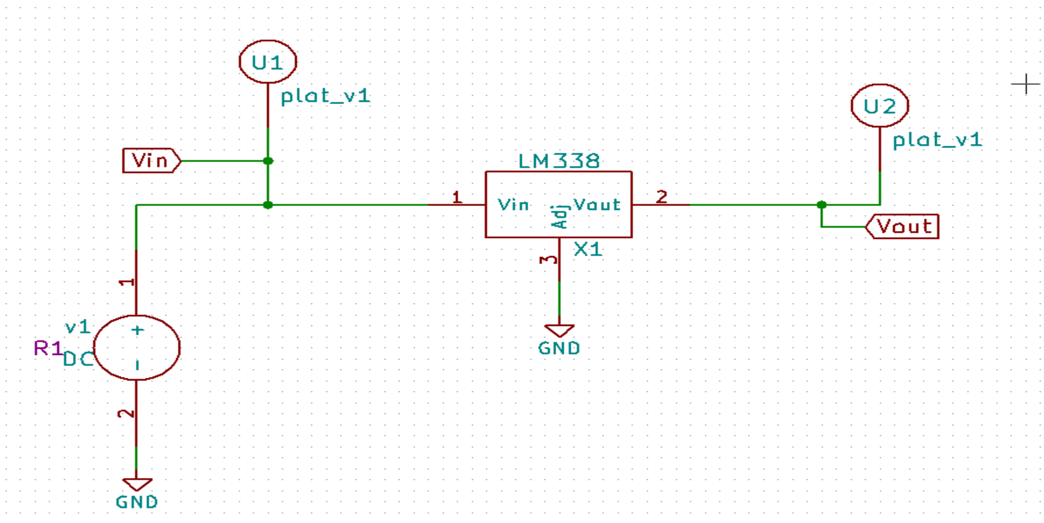

Figure 4.33: Test Circuit of LM338 IC

4.7.4 Input Plots

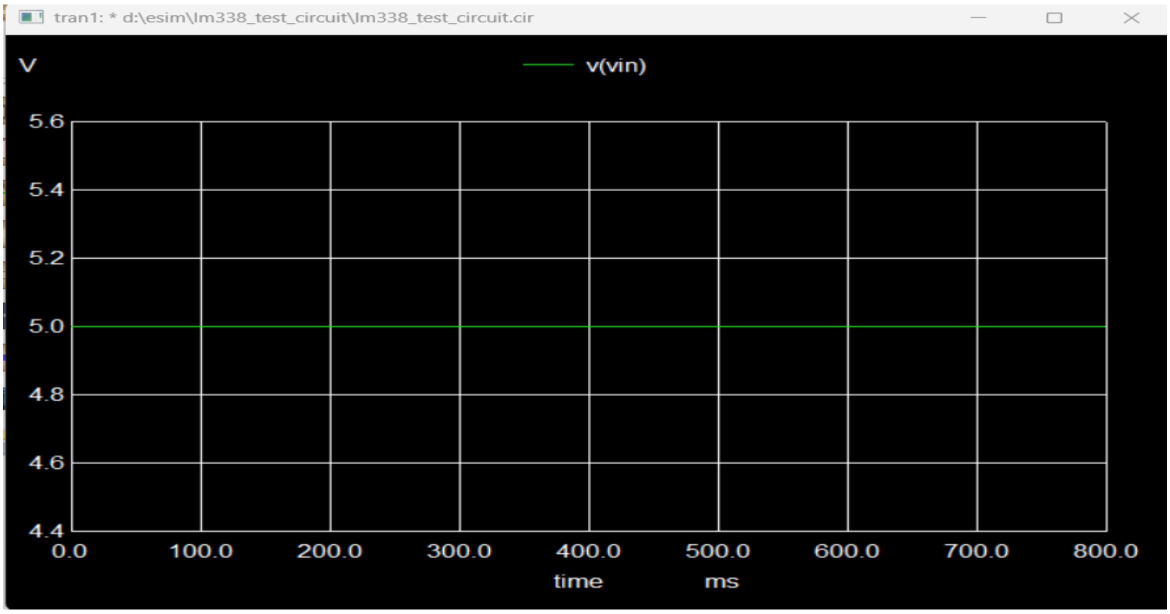

Figure 4.34: Input Voltage Waveform ofLM338

4.7.5 Output Plots

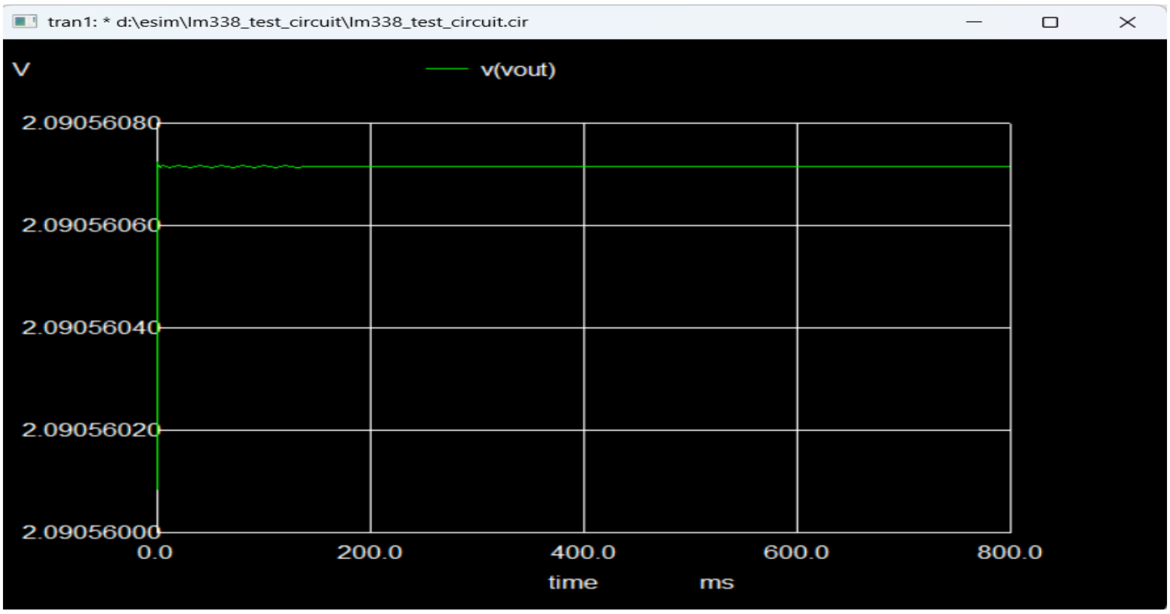

Figure 4.35: Output Voltage Waveform of LM338

4.8 LM231 - Voltage to Frequency Converter

The LM231 is a voltage-to-frequency converter that generates a pulse output proportional to the input voltage. It works with both single and dual power supplies (as low as 5V) and is highly accurate with a 0.01 linearity. It is temperature stable consumes low power (15 mW at 5V), and supports a wide frequency range (1 Hz to 100 kHz). Common applications include ADC conversion, frequency-to-voltage conversion, tachometers, and remote sensing.

The LM231 is used in voltage-to-frequency and frequency-to-voltage conversion applications. It is commonly used in analog-to-digital conversion (ADC), tachometers, and remote sensor monitoring. This IC helps convert sensor signals into digital pulses, making data transmission and processing easier. It is also used in frequency modulation/demodulation and long-term signal integration for precise measurements.

Advantages of the LM231 has high accuracy with low non-linearity (0.01), ensuring precise voltage-to-frequency conversion. It works with both single and dual power supplies, making it flexible for different applications. It has low power consumption (15mW at 5V) and excellent temperature stability (±50 ppm/°C). The IC supports a wide frequency range (1 Hz to 100 kHz) and produces pulse outputs compatible with all logic circuits.

4.8.1 IC Layout

This figure represents the 8-Pin Package Diagram of the LM231 Voltage to Frequency Converter

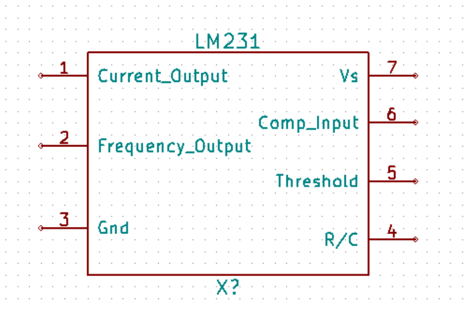

Figure 4.36: LM231 Voltage to Frequency Converter

4.8.2 Subcircuit Schematic Diagram

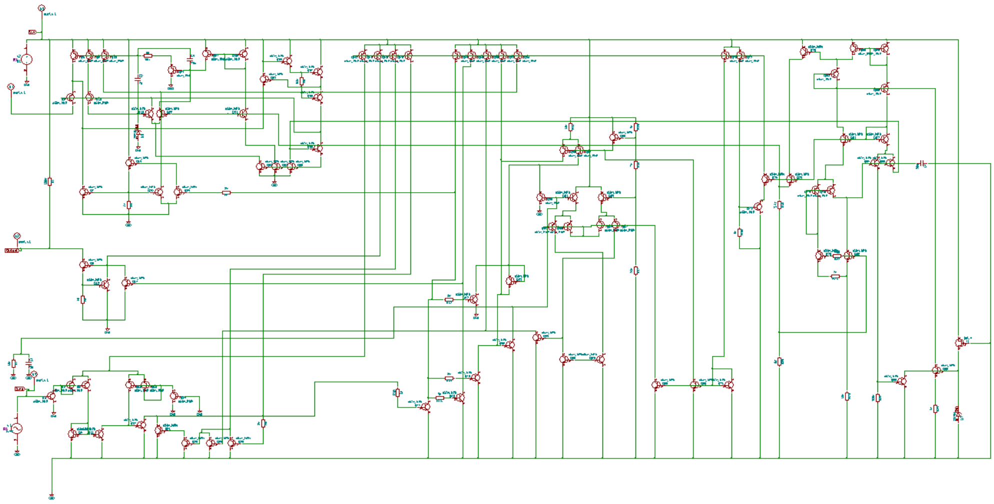

Figure 4.37: Subcircuit Schematic of LM231

4.8.3 Test Circuit

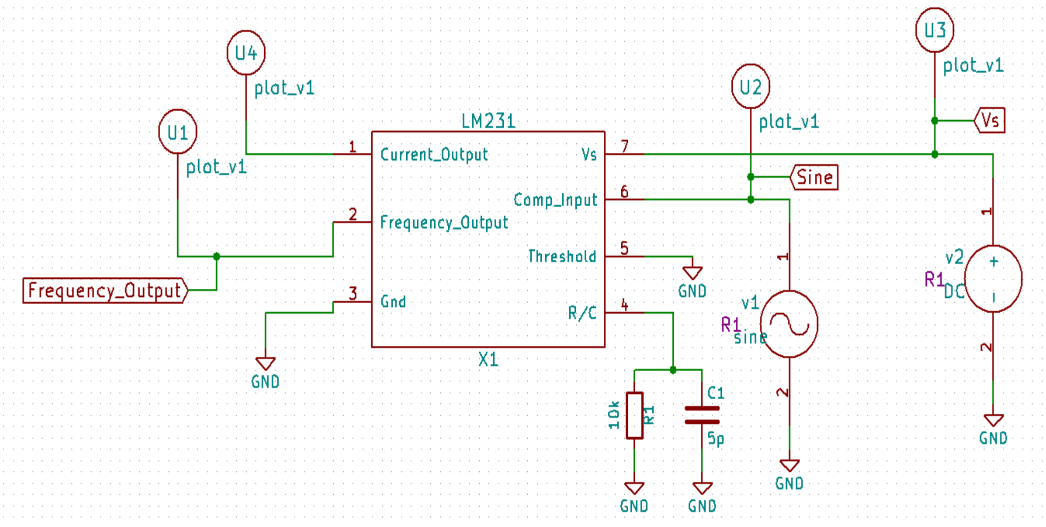

Figure 4.38: Test Circuit of LM231 IC

4.8.4 Input Plots

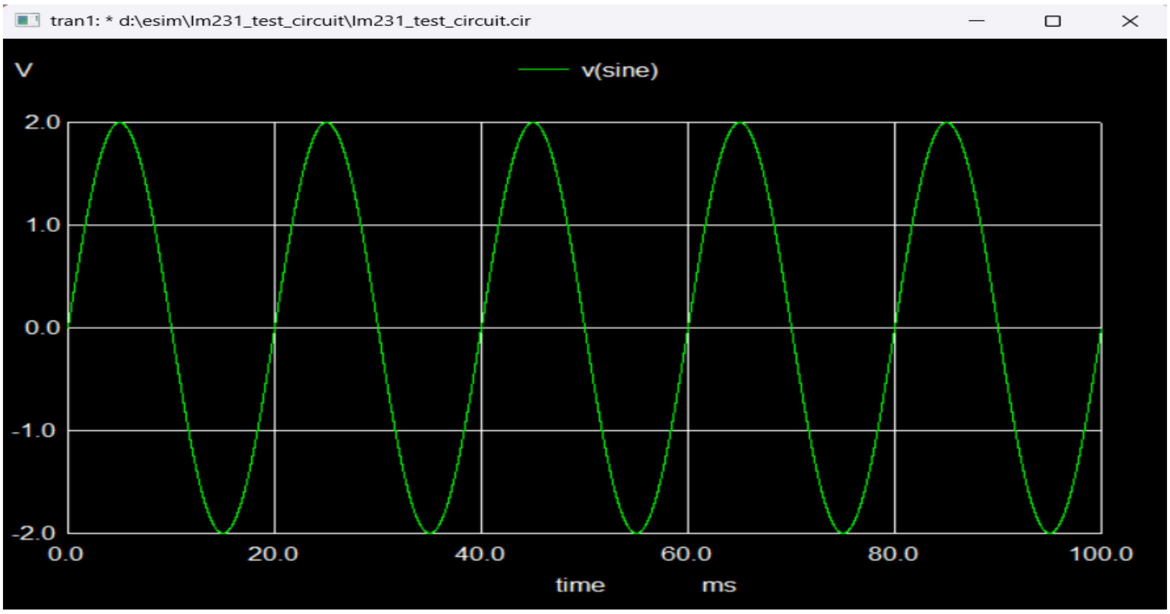

Figure 4.39: Input Voltage Waveform ofLM231

4.8.5 Output Plots

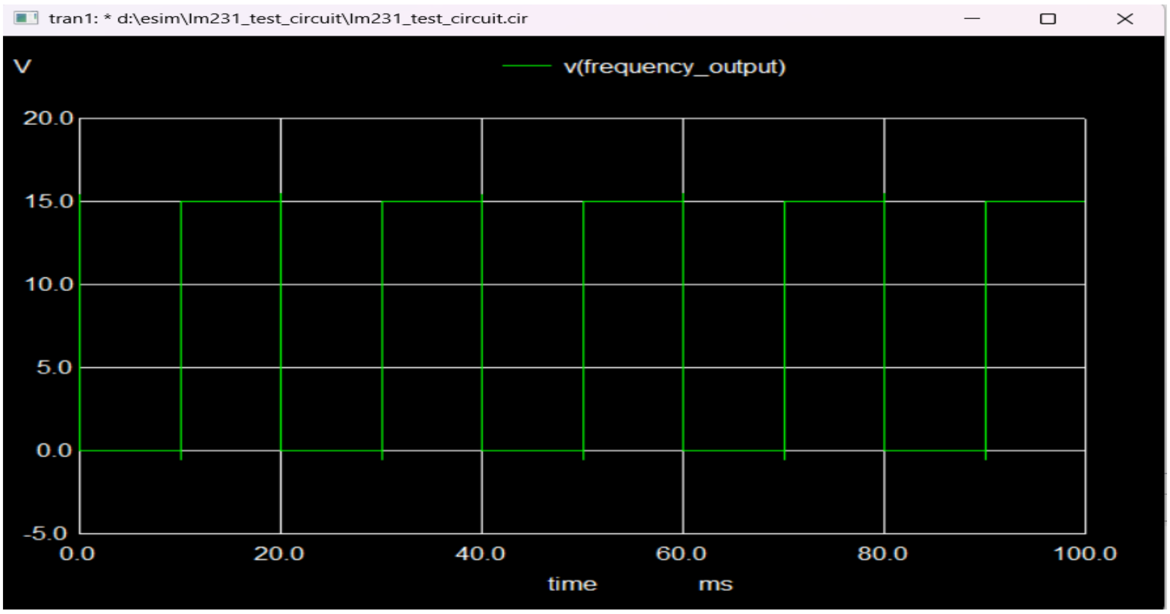

Figure 4.40: Output Voltage Waveform of LM231

Chapter 5

Conclusion and Future Scope

We were successful to achieve the target of developing various subcircuits for Analog Integrated Circuits. Each Integrated Circuit Model was developed strictly according to the information contained in their official data-sheets.The output of each IC was verified and tested successfully with the help of their test circuits. All of these IC Models, developed under this Fellowship are very basic circuit units, such as Op-Amps, Voltage Regulators, Precision Rectifier, Voltage to frequency converter, Differential Amplifier, Instrumentation Amplifier and Comparator ICs. Each of these ICs is ready to be integrated in the subcircuit library of eSim. Developers and Students can use these ICs in their projects and circuit models as units. With the development and expansion of the device model library in eSim, We expect more such ready to use IC models be developed to be used in eSim.

Bibliography

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

[2] eSim Official Website.
https://esim.fossee.in/

[3] LM60 IC.
https://www.ti.com/lit/gpn/LM60.pdf

[4] LM324 IC.
https://www.ti.com/lit/ds/symlink/lm324.pdf

[5] TMP35 IC.
https://www.alldatasheet.com/datasheet-pdf/pdf/49107/AD/TMP35.html

[6] LM7815 IC.
https://www.alldatasheet.com/datasheet-pdf/view/9046/NSC/7815.html

[7] LM614 IC.
https://www.ti.com/lit/gpn/LM614-MIL

[8] LM833 IC.
https://www.ti.com/lit/gpn/LM833.pdf

[9] LM338 IC.
https://www.ti.com/lit/ds/symlink/lm338.pdf

[10] LM231 IC.
https://www.ti.com/lit/gpn/LM331.pdf