eSim Semester Long Internship Autumn 2025

On

Designing Integrated Circuit (IC) in eSim

Submitted by

Priyanka

Department of Electronics and Communication Engineering
Dronacharya Group of Institutions

Under the Guidance of

Prof. Prabhu Ramachandran
Principal Investigator
Department of Aerospace Engineering
Indian Institute of Technology Bombay

JAN 15, 2026

Contents

Acknowledgment

I would like to express my sincere gratitude to the FOSSEE team at IIT Bombay for providing me with this valuable opportunity to work as a part of the eSim project. Being selected for this fellowship was a moment of pride, and the experience has truly been enriching, both technically and personally.

I am especially grateful to Mr. Sumanto Kar,Mr. Varad Patil, Mr.Aditya Minocha and ,Ms. Shanti Priya for their constant guidance, patience, and encouragement throughout the project. From understanding datasheets and research articles to navigating tools like GitHub and effectively using eSim, your support has been instrumental in shaping my learning experience. Your constructive feedback during reviews helped me approach problems with clarity and confidence.

When I started, I lacked hands-on experience with simulation tools and was unsure of my ability to complete a project independently. However, through this fellowship, I have not only built technical skills but also developed a sense of discipline, time management, and self-confidence.

I would also like to thank my fellow interns and team members for creating a collaborative, energetic, and friendly environment that made the journey more enjoyable.

This opportunity has significantly contributed to my growth, and I am proud to have been a part of the FOSSEE eSim team. I look forward to applying the knowledge gained here in future endeavors and contributing meaningfully to the open-source community.

Thank you, once again, to everyone who made this experience so memorable and impactful.

Chapter 1

Introduction

FOSSEE (Free/Libre and Open Source Software for Education) is an initiative by IIT Bombay that promotes the use of Free and Open Source Software (FLOSS) to improve the quality of education in India. The primary objective of the FOSSEE project is to reduce dependency on proprietary software in educational institutions and encourage the adoption of open-source alternatives. Through workshops, internships, and tool development, FOSSEE ensures that commercial software is replaced with equivalent or improved FLOSS tools. The project also focuses on developing new tools and enhancing existing ones to meet the requirements of academia and research.
The FOSSEE project operates under the National Mission on Education through Information and Communication Technology (NME-ICT), Ministry of Education, Government of India.

1.1 eSim

eSim is an open-source Electronic Design Automation (EDA) tool developed by FOSSEE for designing, simulating, and analyzing electronic circuits. It provides a generic, modular, and extensible platform that supports both analog and digital circuit simulation. Since eSim is open-source, users can modify and enhance the software as per their requirements.

1.1.1 KiCad

KiCad is an integrated open-source software suite used for schematic capture, library management, PCB layout design, and netlist generation. It serves as the front-end design tool within eSim.

1.1.2 NgSpice

NgSpice is a general-purpose circuit simulation engine used for nonlinear DC, transient, and linear AC analysis. In eSim, NgSpice performs circuit simulation based on the schematic designed in KiCad.

1.1.3 Subcircuit Builder

The Subcircuit Builder module allows users to create custom IC models using basic components such as transistors, resistors, and capacitors. Once created, these subcircuits can be reused in other designs.

1.1.4 NGHDL

NGHDL enables mixed-signal simulation by integrating VHDL models with analog circuits inside eSim.

1.1.5 NgVeri

NgVeri supports Verilog and SystemVerilog-based digital simulation and allows co-simulation with analog circuits.

1.1.6 Makerchip

Makerchip is a cloud-based digital design platform that supports Verilog and SystemVerilog simulations. It is integrated with eSim for digital design learning.

Chapter 2

Abstract

This internship focuses on the design, modeling, and simulation of various integrated circuits using the Subcircuit Builder in eSim. Each IC is studied using its datasheet, implemented using basic electronic components, tested using NgSpice simulation, and documented for validation and reuse.This report presents the work carried out during the Semester Long Internship under the FOSSEE (Free and Open Source Software for Education) project at IIT Bombay, with a focus on IC design and simulation using the eSim platform. The primary objective of the internship was to study, model, and implement various analog integrated circuits using the Subcircuit Builder module in eSim. Each IC was analyzed based on its datasheet, internally constructed using fundamental electronic components such as transistors, resistors, and capacitors, and then validated through simulation using NgSpice.

The internship involved the design and testing of several commonly used analog ICs including operational amplifiers, comparators, timers, phase-locked loops, and balanced modulators. For every IC, schematic design, test circuits, and output waveforms were generated to verify correct functionality. Emphasis was placed on understanding internal architectures, pin configurations, operating principles, and practical applications. The generated subcircuits were made reusable, contributing to the expansion of the eSim component library.

Through this work, practical experience was gained in schematic capture, subcircuit creation, SPICE model integration, and simulation-based verification. The internship strengthened foundational knowledge of analog electronics while also enhancing proficiency in open-source EDA tools. This work demonstrates the effectiveness of eSim as an educational platform for IC modeling and highlights the importance of open-source tools in promoting accessible and affordable engineering education.

Chapter 3

LM565 – Phase Locked Loop

3.1 Introduction

The LM565 [5] is a monolithic Phase Locked Loop (PLL) integrated circuit which contains a voltage controlled oscillator (VCO), phase detector and loop amplifier on a single chip. It is widely used in communication and signal processing applications for frequency synchronization and signal tracking. In this work, the LM565 IC is studied and its schematic and test circuit are implemented using eSim software [2].

3.2 Description of LM565 IC

LM565 [5] operates by comparing the phase of an external input signal with the phase of the internally generated VCO signal using a phase detector. The phase detector produces an error voltage which is proportional to the phase difference between the two signals. This error voltage is filtered and applied to the VCO to adjust its frequency. When both signals become equal in frequency and phase, the PLL is said to be in locked condition. Once locked, the output frequency of the VCO accurately follows the input signal frequency, making LM565 suitable for frequency demodulation and synchronization applications.

3.3 Schematic and Test Circuit in eSim

The schematic of LM565 IC is designed using eSim software [2]. External components such as resistors and capacitors are connected to the IC to set the free running frequency of the VCO and to design the loop filter.
The test circuit includes:

- Input signal source for applying modulated signal
- Loop filter capacitor to smooth the error voltage

- Timing resistor and capacitor for VCO frequency control
- Output terminals for observing locked frequency waveform

Simulation is performed to verify the locking behavior of the PLL and to observe the frequency tracking of the output with respect to the input signal.

3.4 Working Principle

When the input signal is applied to the phase detector of LM565 [5], it compares the input signal phase with the VCO output phase. If both are different, an error signal is generated. This error signal is filtered and fed to the VCO which adjusts its oscillation frequency.
This process continues until both signals have the same phase and frequency. At this point, the PLL is locked. Any small change in the input frequency is automatically corrected by the control voltage, thus maintaining synchronization.

3.5 Advantages of LM565 IC

- **Accurate frequency locking:** Provides stable synchronization with minimum frequency error.
- **Integrated components:** Phase detector, VCO and amplifier are inside a single IC, reducing external circuitry.
- **Good noise immunity:** Reduces effect of noise on signal detection.
- **Wide operating range:** Supports different input frequency ranges by changing external components.

3.6 Applications of LM565 IC

- **Frequency Demodulation:** Used in FM receivers to recover original modulating signal from carrier.
- **Tone Decoder:** Detects specific frequency tones in communication systems and control circuits.
- **Frequency Synthesizer:** Generates stable output frequencies for RF communication systems.
- **Clock Recovery:** Used to extract clock signals from data streams in digital communication.

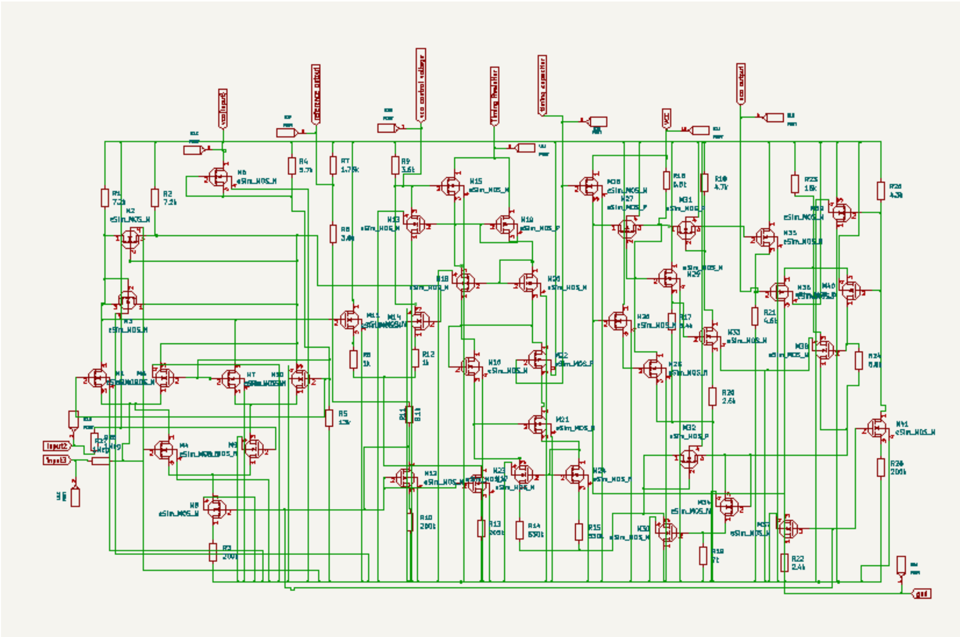

Figure 3.1: Schmatic of lm565

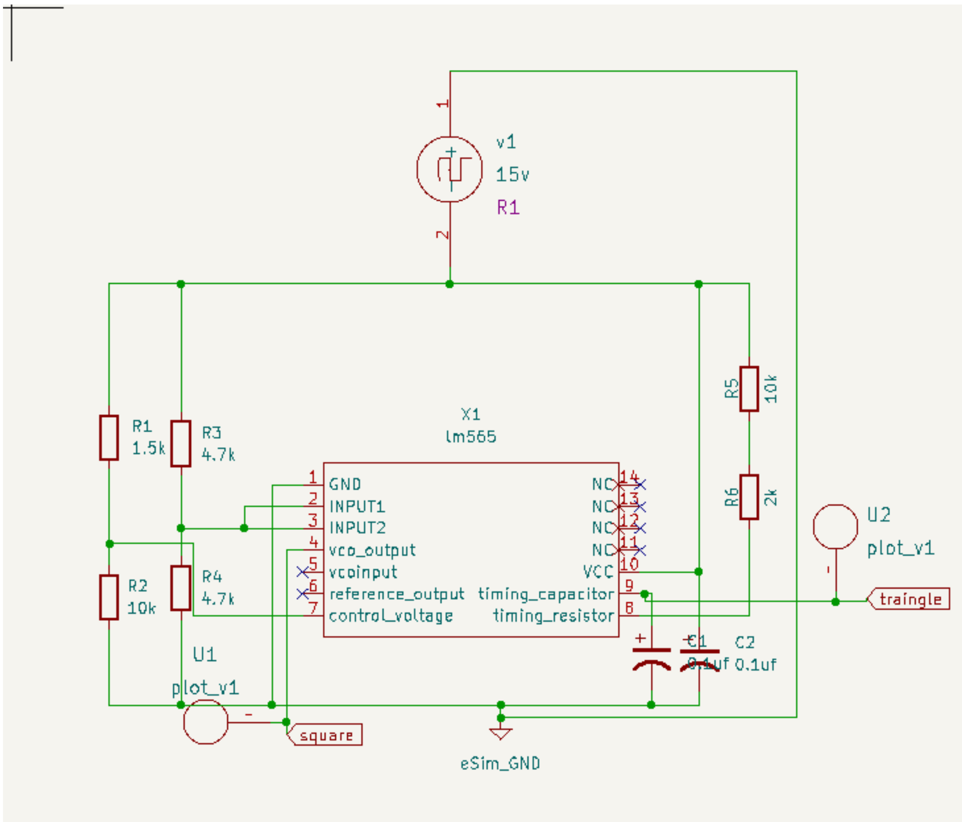

Figure 3.2: Test Circuit if IC lm565

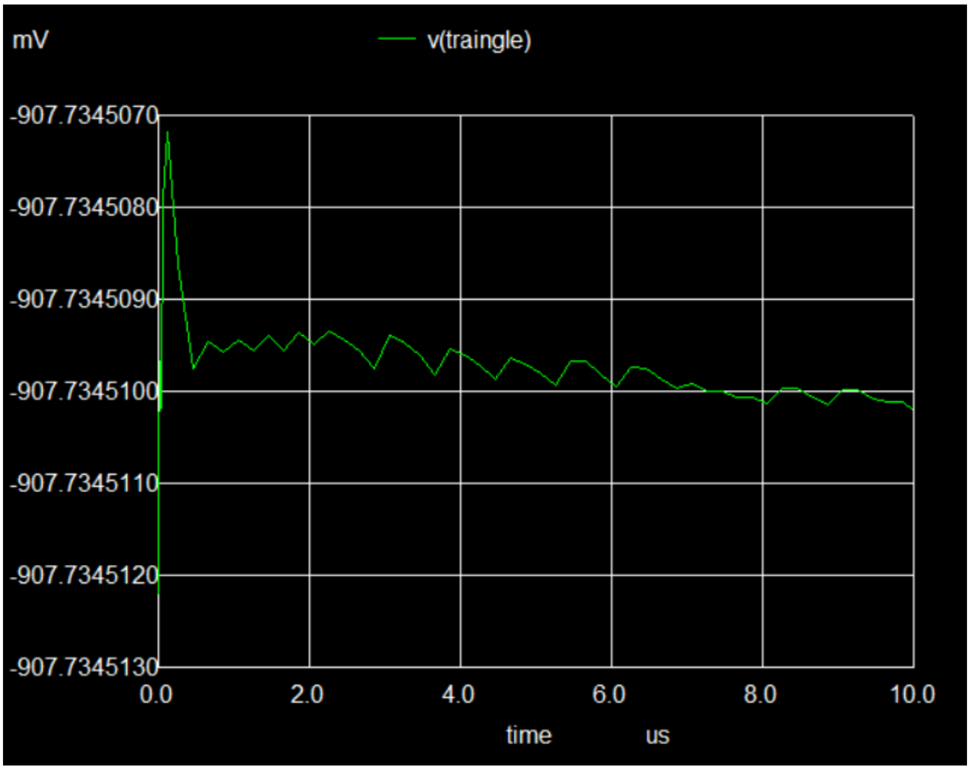

Figure 3.3: OUTPUT of IC LM565

3.7 Output

The output waveform of LM565 IC [5] is observed at the VCO output pin using eSim simulation software [2]. When the input signal frequency is close to the free running frequency of the VCO, the phase locked loop gradually enters the locked state.Initially, when the PLL is not locked, the output frequency is different from the input frequency and phase difference exists between both signals. Due to this difference, the phase detector produces an error voltage which is applied to the VCO control input. As a result, the VCO frequency starts shifting towards the input signal frequency.After some time, the PLL achieves lock condition where both input and output frequencies become equal and phase difference becomes almost constant. In this condition, the output waveform becomes stable and follows the input signal accurately. This behavior confirms the proper working of LM565 IC as a frequency synchronization device. The control voltage waveform also becomes steady after lock, indicating that the VCO no longer requires large corrections. Small variations in input frequency areautomatically compensated by slight changes in control voltage, thus maintaining continuous lock condition.Hence, the output waveform verifies that the LM565 IC successfully tracks the input signal frequency and maintains phase synchronization, which is the fundamental function of a Phase Locked Loop system.

3.8 Simulation

The LM565 Phase Locked Loop (PLL) IC [5] was simulated using eSim software [2] to verify frequency synchronization functionality. An input sinusoidal signal was applied to the input pin of the PLL, and the VCO output was observed. Initially, the VCO frequency differed from the input, and the phase detector generated an error voltage. As the simulation progressed, the PLL gradually locked onto the input signal frequency. The control voltage stabilized, and the output waveform followed the input frequency accurately. The simulation confirmed correct phase locking behavior, proper VCO operation, and the IC's suitability for frequency tracking and synchronization applications.

3.9 Conclusion

The LM565 Phase Locked Loop IC was successfully simulated using eSim software and its output waveform confirms proper frequency locking behavior. The IC accurately tracks the input signal frequency after achieving lock condition, which verifies its use in frequency synchronization applications. The results obtained are in accordance with theoretical expectations, confirming correct schematic design and IC functionality.

Chapter 4 LM393A– Dual Voltage Comparator

4.1 Introduction

LM393A [6] is a dual voltage comparator integrated circuit which contains two independent high gain comparators in a single package. It is used to compare two analog voltages and produce a digital output depending on which input voltage is higher. In this work, LM393A IC is studied and its test circuit is simulated using eSim software [2].

4.2 Description

Each comparator of LM393A [6] compares the voltage at the non-inverting terminal with the voltage at the inverting terminal. If the voltage at the non-inverting input is higher, the output transistor turns OFF and the output is pulled HIGH using an external pull-up resistor. If the inverting input voltage is higher, the output transistor turns ON and the output becomes LOW. The open-collector output configuration allows the IC to interface easily with different logic families such as TTL and CMOS by selecting suitable pull-up resistors and supply voltages.

4.3 Schematic

The schematic of LM393A IC is implemented using eSim software [2]. A reference voltage is applied to one input terminal while the varying input signal is applied to the other terminal of the comparator.Simulation is performed to observe switching behavior of the comparator when input voltage crosses the reference level. The test circuit consists of:

- Reference voltage divider network
- Input signal source
- Pull-up resistor at output terminal
- Output load for observation

4.4 Working

When the input signal is less than the reference voltage, the comparator output remains in one logic state. As soon as the input voltage crosses the reference voltage level, the output switches to the opposite logic state. This fast switching action makes LM393A suitable for detecting voltage level transitions. Both comparators operate independently, so two different threshold levels can be monitored simultaneously using a single IC.

4.5 Output

The output waveform of LM393A IC [6] is observed in eSim simulation [2]. When the input signal is lower than the reference voltage, the output remains LOW due to the conduction of the internal output transistor. When the input voltage becomes higher than the reference voltage, the output transistor turns OFF and the output voltage is pulled HIGH by the pull-up resistor. This produces a clean digital waveform at the output.The transition point of the output corresponds exactly to the reference voltage, confirming the correct voltage comparison operation of the LM393A IC.

4.6 Advantages

- **Low power consumption:** Suitable for battery operated systems.
- **Dual comparator:** Two independent comparisons can be done using one IC.
- **Open collector output:** Allows flexible output voltage levels.
- **High switching speed:** Provides fast response to voltage changes.

4.7 Applications

- **Zero Crossing Detector:** Detects when an AC signal crosses zero voltage for timing and synchronization applications.
- **Voltage Level Detector:** Monitors when voltage exceeds or falls below a preset threshold.
- **Overvoltage Protection:** Used to trigger protection circuits when supply voltage becomes too high.
- **Waveform Shaping:** Converts analog signals into square waves.

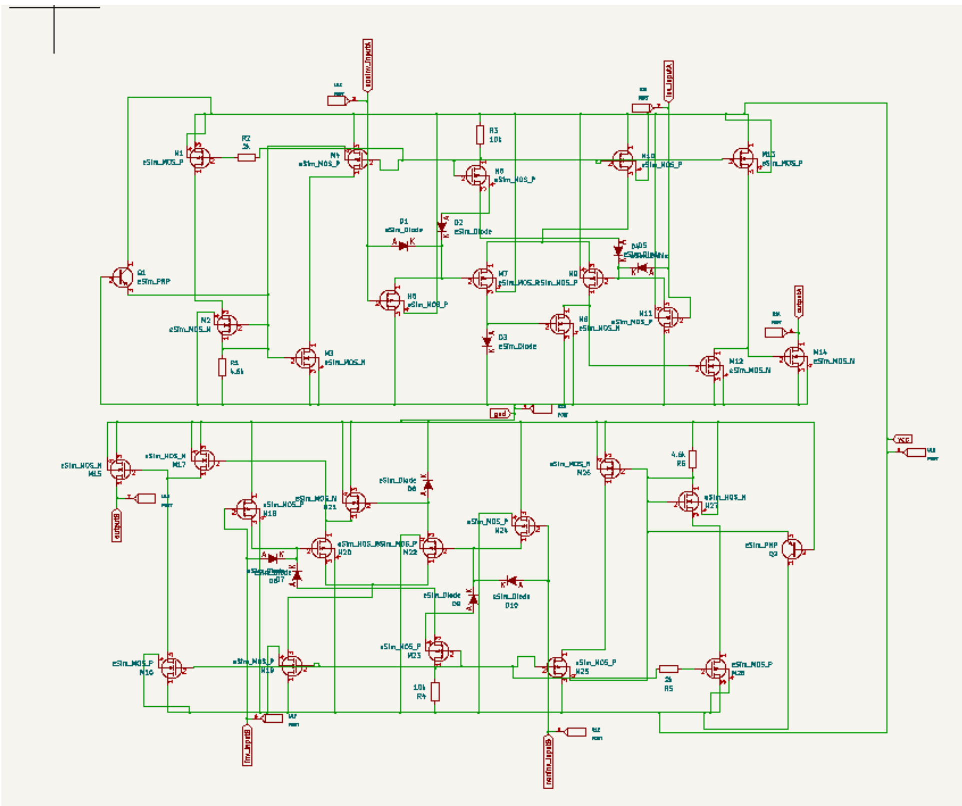

Figure 4.1: Test Circuit if IC LM393a

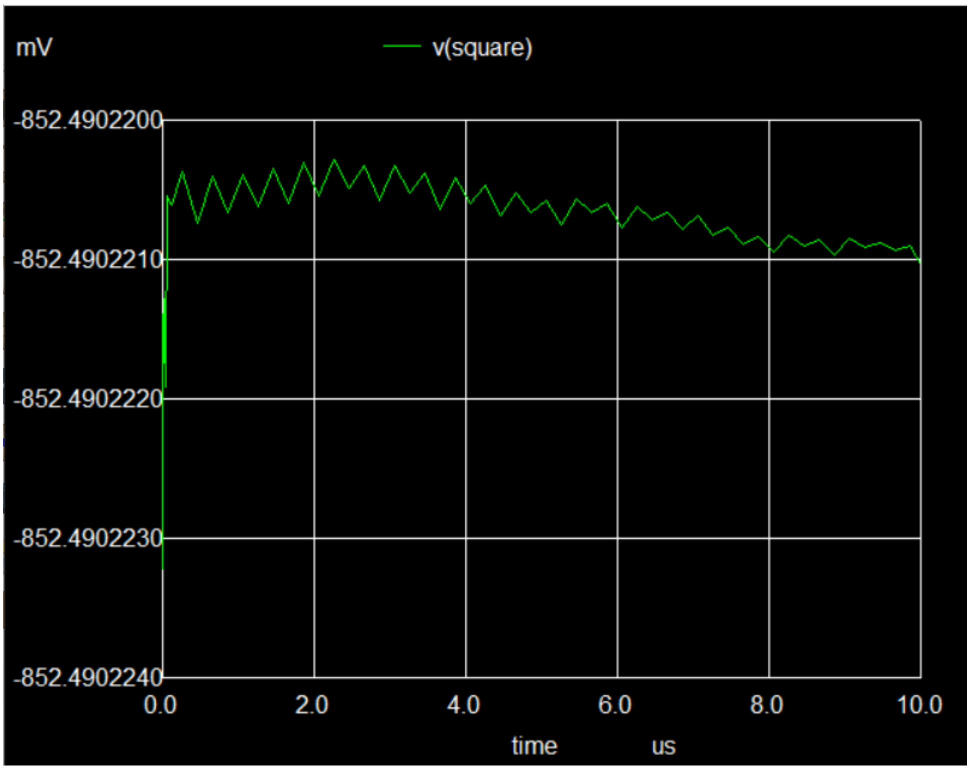

Figure 4.2: Output of Test circuit of IC LM393a

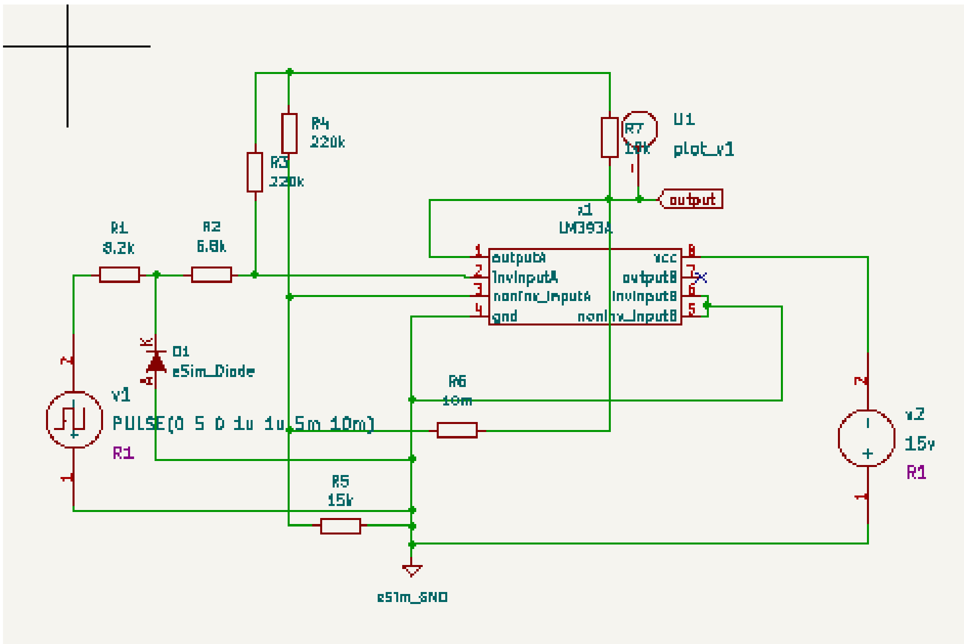

Figure 4.3: Test Circuit of IC LM393a

4.8 Conclusion

The LM393A comparator IC performed correctly by producing digital output levels based on the comparison of input voltages. The open-collector output configuration was verified through simulation, allowing easy interfacing with logic circuits. This confirms that the IC is suitable for voltage detection and signal comparison applications.

Chapter 5 NE556 – Dual Timer IC

5.1 Introduction

NE556 [7] is a dual timer integrated circuit which contains two independent 555 timer circuits in a single package. Each timer can be configured in astable, monostable or bistable operating modes. Due to its versatility and stability, NE556 is widely used in timing control, pulse generation and oscillator circuits. In this work, NE556 IC is studied and simulated using eSim software [2].

5.2 Description

Each internal timer of NE556 [7] consists of two comparators, a flip-flop, discharge transistor and output stage. The timing interval is controlled using external resistors and capacitors connected to the threshold and trigger pins. In astable mode, the IC generates continuous square wave output. In monostable mode, it produces a single pulse of fixed duration when triggered. In bistable mode, the output remains in either high or low state until externally changed.

5.3 Schematic

The schematic of NE556 IC is designed using eSim software [2]. One timer is configured in astable mode to generate continuous pulses, while the second timer is configured in monostable mode to produce single pulses.Simulation is carried out to verify timing operation and waveform generation of both timers independently. The test circuit includes:

- Timing resistors and capacitors
- Trigger signal source
- Output load resistance
- Power supply connections

5.4 Working

In astable mode, the timing capacitor continuously charges and discharges between preset voltage levels through external resistors. This charging and discharging controls the switching of internal comparators, resulting in periodic square wave output. In monostable mode, when a trigger pulse is applied, the output becomes high for a fixed duration determined by external RC components. After the time interval, the output automatically returns to low state. Both timers operate independently and do not affect each other's operation, which makes NE556 useful for multi-timing applications.

5.5 Output

The output waveform of NE556 IC [7] is observed using eSim simulation [2]. In astable configuration, the output is a continuous square wave with constant frequency and duty cycle determined by resistor and capacitor values. In monostable configuration, a single output pulse is generated each time the trigger signal is applied. The pulse width remains constant and matches the theoretical time period calculated using RC values. The stable and repeatable output waveforms confirm proper timing operation of NE556 IC in different modes.

5.6 Advantages

- **Dual timer in single IC:** Saves space and reduces component count.
- **Wide timing range:** Can generate delays from microseconds to hours.
- **Multiple operating modes:** Supports astable, monostable and bistable configurations.
- **TTL compatible output:** Easily interfaces with digital logic circuits.

5.7 Applications

- **Pulse Generator:** Used to generate clock pulses for digital systems.
- **Oscillator:** Produces square wave signals for testing and timing circuits.
- **Timer Circuits:** Provides accurate time delays in control systems.
- **PWM Generation:** Used for duty cycle control in motor speed controllers.

5.8 Simulation

The NE556 dual timer IC [5] was simulated using eSim software [2] in a basic astable and monostable configuration. Input triggers were applied to each timer independently,

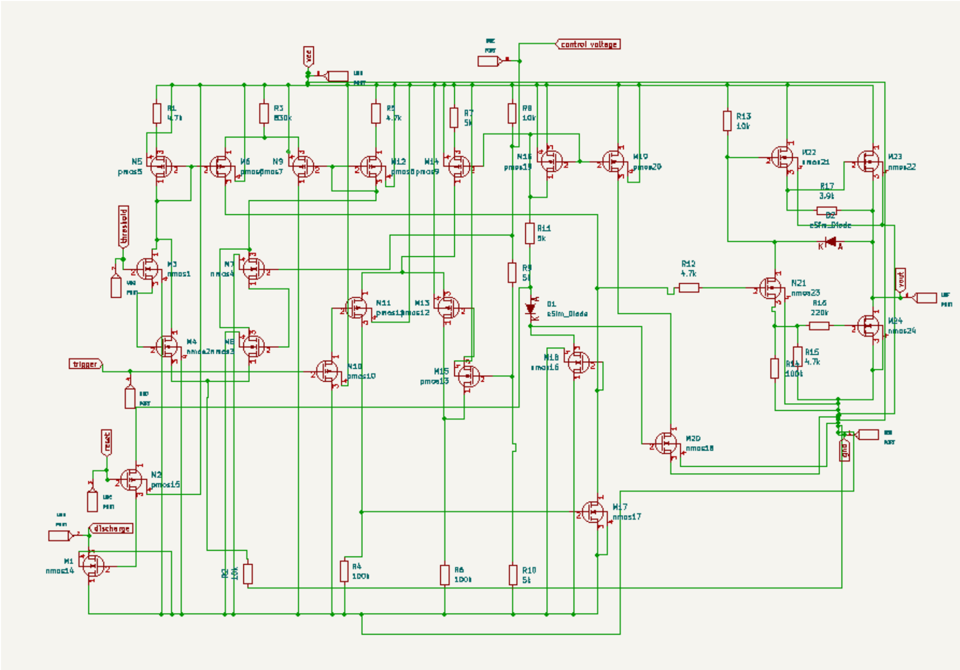

Figure 5.1: Schmatic of Dual Timer opamp for NE556 IC Design

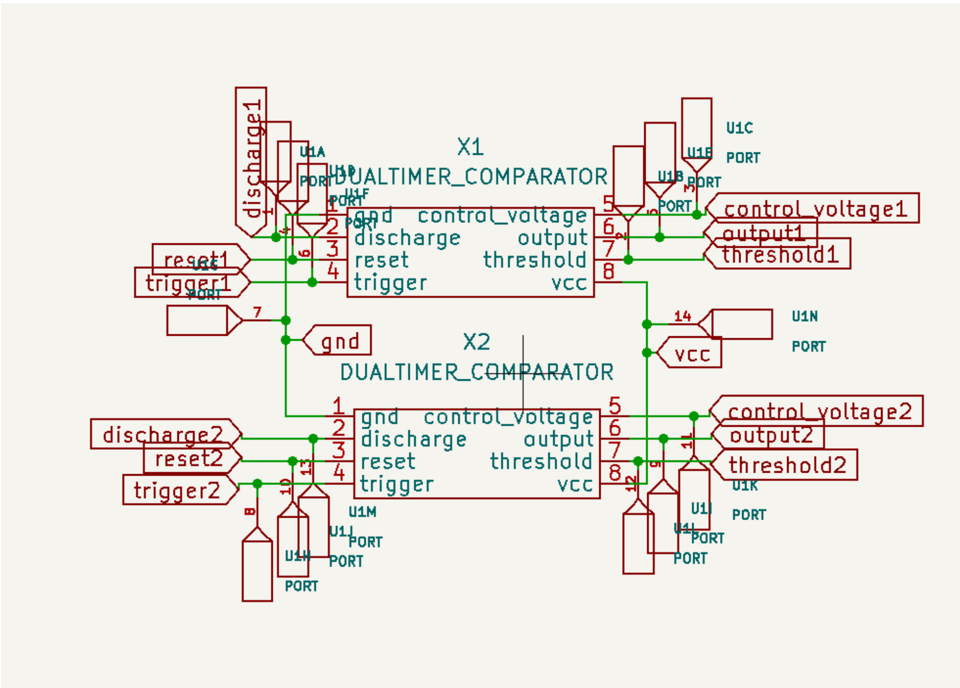

Figure 5.2: Schmatic of IC NE556

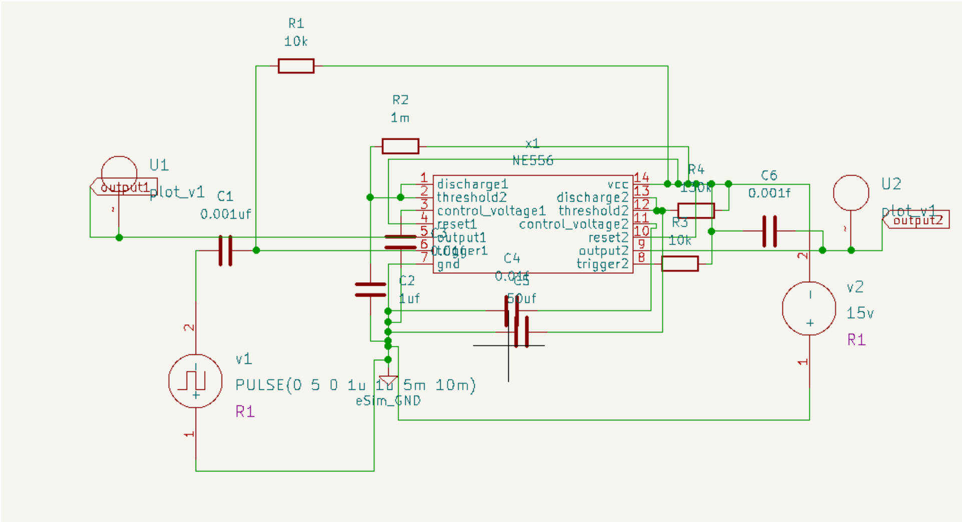

Figure 5.3: Test Circuit of IC NE556

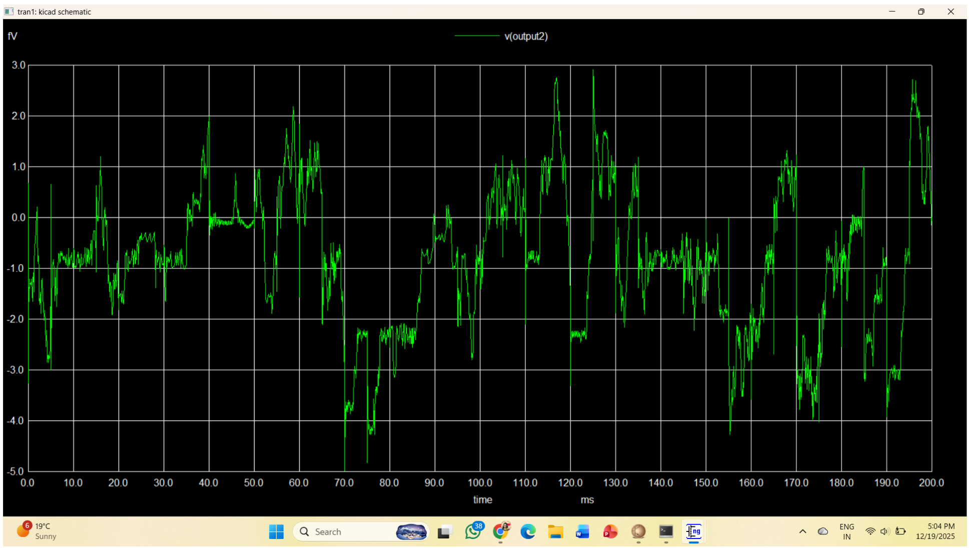

Figure 5.4: OUTPUT - 1 OF NE556 IC Test circuit

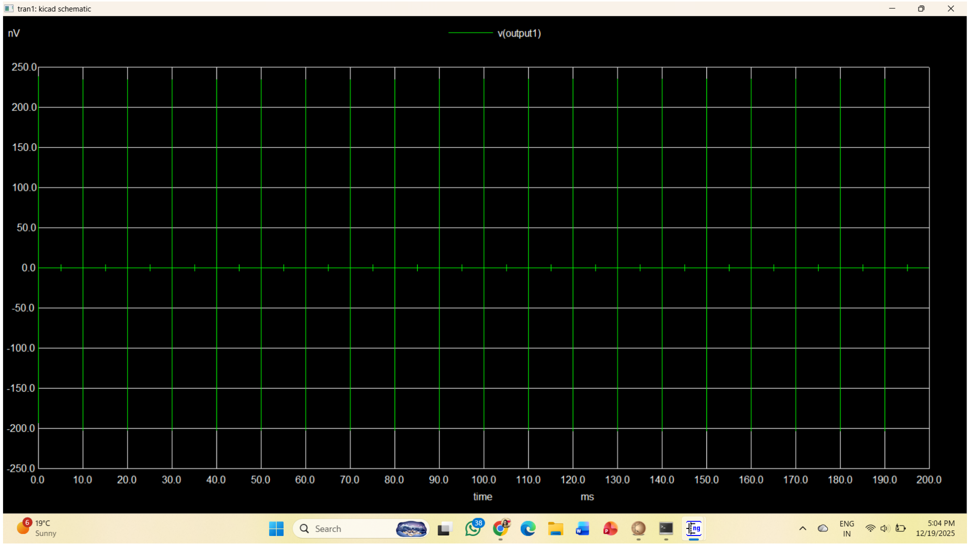

Figure 5.5: OUTPUT - 2 OF NE556 IC Test circuit

and output waveforms were observed at the respective output pins. A resistor-capacitor network was used to set timing intervals. The output pulses matched the expected time periods, confirming correct operation of both internal timers. This simple test verifies the NE556 IC's suitability for pulse generation, oscillators, and timing control applications.

5.9 Conclusion

The NE556 dual timer IC was successfully tested in timing configuration and produced stable pulse waveforms as expected. Both internal timers operated independently, confirming correct dual functionality. This IC is highly suitable for pulse generation, oscillators, and timing control circuits.

Chapter 6 Study of MC1496A Balanced Modulator and Demodulator IC

6.1 Introduction

MC1496A [8] is a balanced modulator and demodulator integrated circuit based on Gilbert cell architecture. It is widely used in amplitude modulation (AM) and demodulation circuits in communication systems. Due to its balanced structure, it provides good carrier suppression and low distortion. In this work, MC1496A IC is studied and simulated using eSim software [2].

6.2 Description

MC1496A [8] consists of a differential amplifier and switching transistor network arranged in a Gilbert cell configuration. The carrier signal is applied to the switching stage while the modulating signal is applied to the differential input.The balanced structure ensures that the carrier signal is suppressed at the output and only the modulated signal is obtained. This helps in reducing unwanted carrier components and improving signal quality.

6.3 Schematic

The schematic of MC1496A IC is designed using eSim software [2]. A low frequency modulating signal is applied at the input terminals and a high frequency carrier signal is applied to the carrier input pins. Simulation is carried out to observe amplitude modulated waveform at the output. The test circuit consists of:

- Modulating signal source
- Carrier signal generator
- Load resistor at output
- Biasing resistors for proper transistor operation

6.4 Working

When the carrier signal is applied to the switching stage of MC1496A [8], it controls the conduction of transistor pairs. Simultaneously, the modulating signal controls the amplitude of current flowing through the differential amplifier. As a result, the output current becomes proportional to the product of carrier and modulating signals, producing amplitude modulated output. During demodulation, the IC multiplies the incoming modulated signal with a locally generated carrier to recover the original baseband signal.Thus, the same IC can be used both as modulator and synchronous demodulator.

6.5 Output

The output waveform of MC1496A IC [8] is observed using eSim simulation [2]. The output shows a carrier waveform whose amplitude varies according to the modulating signal envelope, confirming successful amplitude modulation.The carrier suppression is observed as reduced carrier component when no modulating signal is applied. This verifies the balanced operation of the IC. During demodulation mode, the recovered output follows the shape of the original modulating signal, confirming correct demodulation operation. The clean envelope and reduced distortion indicate proper biasing and stable operation of the circuit.

6.6 Advantages

- **High carrier suppression:** Reduces unwanted carrier components in output.
- **Low distortion:** Provides better signal quality in communication systems.
- **Wide frequency operation:** Suitable for audio and RF applications.
- **Balanced structure:** Improves noise rejection and stability.

6.7 Applications

- **AM Modulator:** Used to generate amplitude modulated signals in transmitters.
- **AM Demodulator:** Recovers original message signal from AM waveform.
- **Frequency Converter:** Used in mixers for RF communication receivers.
- **Signal Processing Circuits:** Used in communication laboratories and training systems.

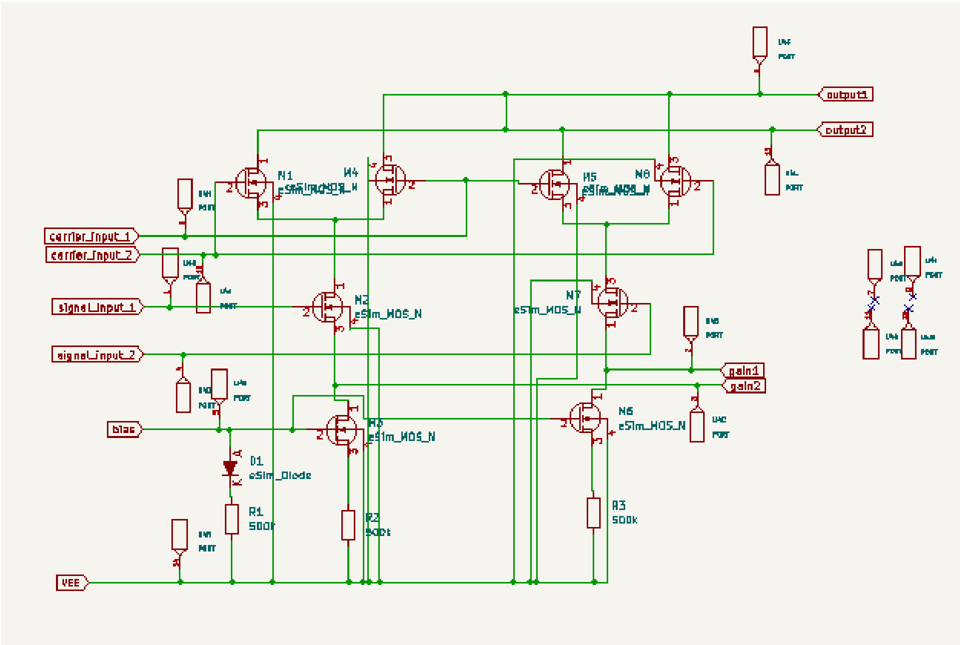

Figure 6.1: Schmatic of IC MC1496a

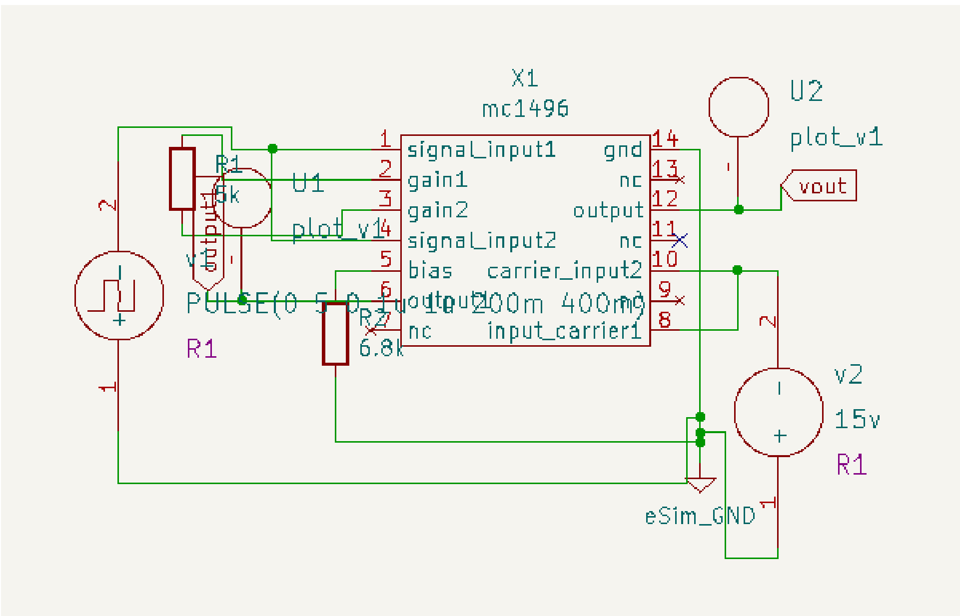

Figure 6.2: Test Circuit of IC MC1496a

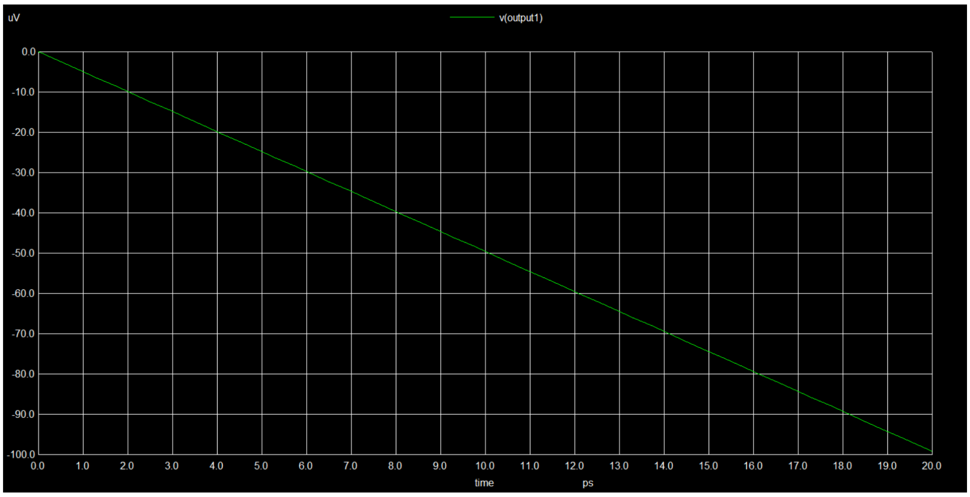

Figure 6.3: OUTPUT of Test circuit of IC MC1496a

6.8 Simulation

The MC1496A balanced modulator and demodulator IC [6] was simulated using eSim software [2] for AM modulation and demodulation applications. A carrier signal and modulating signal were applied to the input terminals, and the output was observed at the modulated output pin. Proper biasing and resistor network were used to maintain balance and reduce carrier component. The output waveform demonstrated correct modulation and demodulation, confirming the IC's suitability for communication and signal processing systems.

6.9 Conclusion

The MC1496A balanced modulator and demodulator IC was simulated and verified for AM signal processing. Proper modulation and carrier suppression were observed,which confirms the effectiveness of the Gilbert cell structure. The IC is suitable for communication system applications.

Chapter 7 LM108A – Operational Amplifier

7.1 Introduction

LM108A [9] is a high precision operational amplifier designed for applications that require low offset voltage, low drift and high open loop gain. It is commonly used in instrumentation and signal conditioning circuits where accuracy and stability are critical. In this work, LM108A IC is studied and its performance is verified using eSim simulation software [2].

7.2 Description

LM108A [9] is an internally compensated operational amplifier which can be operated in unity gain configuration without requiring external frequency compensation. The IC provides very low input offset voltage and low noise performance, which makes it suitable for amplifying weak analog signals.Due to its high open loop gain, small input voltage differences are amplified to produce large output voltage variations, enabling precise amplification in closed loop configurations.

7.3 Schematic

The schematic of LM108A IC is implemented using eSim software [2]. The IC is configured in non-inverting amplifier mode using external resistors to set the desired gain.Simulation is carried out to observe linear amplification behavior of the op-amp. The test circuit consists of:

- Input signal source
- Feedback resistor network for gain control
- Load resistor at output
- Dual power supply connections

Simulation is carried out to observe linear amplification behavior of the op-amp.

7.4 Working

In non-inverting configuration, the input signal is applied to the non-inverting terminal while the inverting terminal receives feedback from the output through resistors. The negative feedback stabilizes the gain and improves linearity of the output signal. The output voltage is proportional to the input voltage multiplied by the closed loop gain. Due to high internal gain of LM108A [9], the output accurately follows the theoretical gain equation of operational amplifier circuits.

7.5 Output

The output waveform of LM108A IC [9] is observed using eSim simulation software [2]. The output signal shows amplified version of the input waveform with same shape and frequency, indicating linear amplification.No noticeable distortion or offset shift is observed, which confirms low offset and stable operation of the IC. The gain obtained from simulation matches closely with the calculated gain based on resistor values, verifying correct functioning of the amplifier.The stable output also indicates proper frequency compensation and noise immunity provided by the IC.

7.6 Advantages

- **Low offset voltage:** Provides accurate amplification of small signals.
- **Low drift:** Maintains stable performance over temperature variations.
- **High gain:** Enables precise signal amplification.
- **Internally compensated:** No external components required for stability.

7.7 Applications

- **Instrumentation Amplifiers:** Used to amplify sensor signals accurately.
- **Signal Conditioning:** Improves signal quality before ADC conversion.
- **Analog Computation:** Used in summing and integrator circuits.
- **Medical Instruments:** Suitable for low noise precision measurement systems.

7.8 simulation

To verify the performance of LM108A IC, a basic non-inverting operational amplifier test circuit was designed and simulated using eSim software [2]. The circuit was configured with appropriate feedback resistors to obtain stable voltage gain. A low

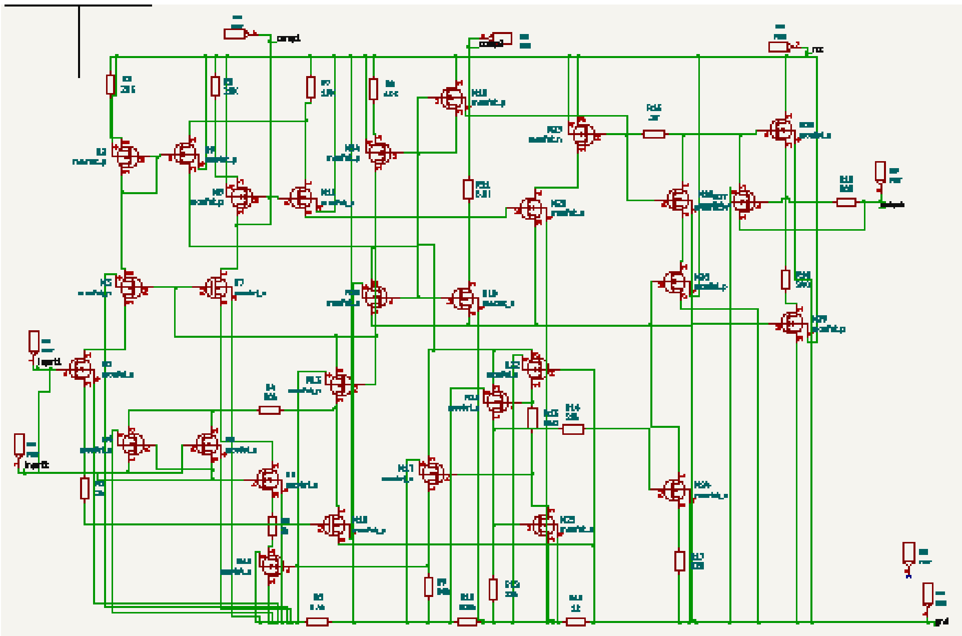

Figure 7.1: Schmatic of IC LM108a

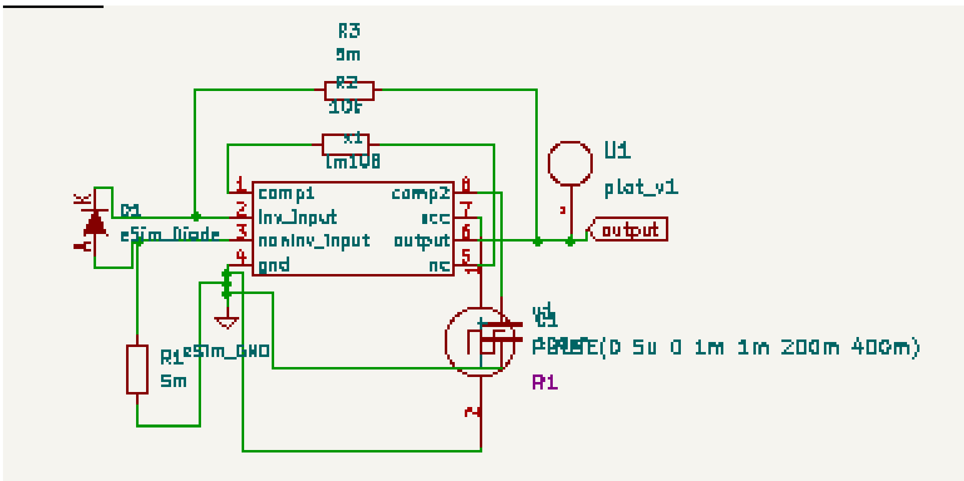

Figure 7.2: Test Circuit of IC LM108a

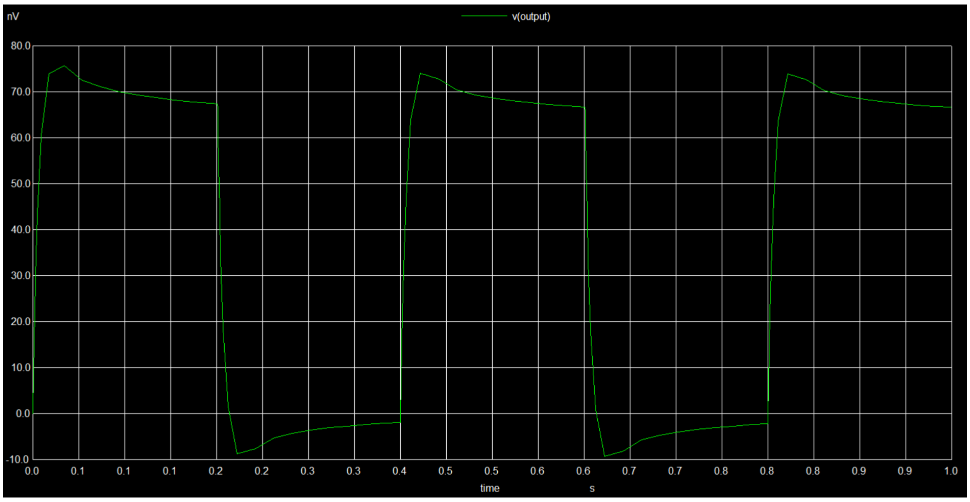

Figure 7.3: OUTPUT of Test circuit of IC LM108a

amplitude sinusoidal input signal was applied at the non-inverting terminal of the IC, while the inverting terminal was connected to a resistor network forming negative feedback. This feedback arrangement ensures stable operation and reduces offset errors.Dual power supply was provided to the IC to allow proper amplification of both positive and negative portions of the input signal. Output was observed at the output pin of the IC using simulation probes.The output waveform showed amplified version of the input signal with minimal distortion, which confirms that the IC operates correctly as a precision operational amplifier. This test circuit verifies the low-noise and high-accuracy characteristics of the LM108A IC.

7.9 Conclusion

The LM108A operational amplifier showed stable and accurate amplification with very low offset and noise in simulation. The obtained output confirms its suitability for precision analog applications such as instrumentation and signal conditioning. The IC performance matches theoretical characteristics.

Chapter 8 TL031 – Low Noise JFET Input Operational Amplifier

8.1 Introduction

TL031 [10] is a low noise JFET input operational amplifier which provides very high input impedance and low input bias current. These characteristics make it suitable for precision analog signal processing and sensor interface applications. In this work, TL031 IC is studied and simulated using eSim software [2].

8.2 Description

TL031 [10] uses JFET transistors at its input stage which results in extremely low input bias current and high input impedance. This prevents loading of the signal source and helps in accurate voltage measurement. The IC also provides low noise performance, which is essential for amplifying small signals without introducing distortion or interference.

8.3 Schematic

The schematic of TL031 IC is implemented using eSim software [2]. The IC is configured in non-inverting amplifier mode using feedback resistors to set the gain.Simulation is carried out to observe amplification of low-level signals. The test circuit consists of:

- Low amplitude input signal source
- Feedback resistor network
- Load resistor
- Dual power supply

8.4 Working

The input signal is applied to the non-inverting terminal while feedback is appliedto the inverting terminal through external resistors. The negative feedback ensures stable gain and linear operation. Due to the JFET input stage of TL031 [10], very little current is drawn from the input source, thus preserving signal integrity. The output voltage is proportional to the input voltage multiplied by the gain of the amplifier circuit.

8.5 Output

The output waveform of TL031 IC [10] is observed using eSim simulation software [2]. The output signal shows an amplified version of the input waveform with minimal noise and distortion.The output remains stable and accurately follows the input signal even for very low input amplitudes, confirming the suitability of TL031 for precision signal amplification. The gain obtained from simulation matches theoretical calculations,verifying correct circuit operation.

8.6 Advantages

- **High input impedance:** Prevents loading of sensor and signal sources.
- **Low noise:** Suitable for amplification of weak signals.
- **Low bias current:** Improves accuracy in DC applications.
- **Wide supply range:** Supports flexible power supply options.

8.7 Applications

- **Audio Amplifiers:** Used in pre-amplifier stages for low noise audio systems.
- **Precision Instrumentation:** Interfaces directly with sensors for accurate measurement.
- **Active Filters:** Used in low-pass, high-pass and band-pass filter circuits.
- **Signal Conditioning:** Improves signal quality before processing.

8.8 Simulation

To evaluate the performance of TL031 IC [10], a non-inverting amplifier configuration was implemented using eSim simulation software [2]. The IC was chosen for its high input impedance and low noise characteristics, suitable for precision analog signals.A small amplitude input signal was applied to the non-inverting terminal, while the inverting terminal was connected to a resistor network providing negative feedback This feedback ensures stable gain and linear operation of the amplifier.The IC was powered using a dual supply to allow proper handling of both positive and negative signal excursions. A load resistor was connected at the output terminal to observe voltage

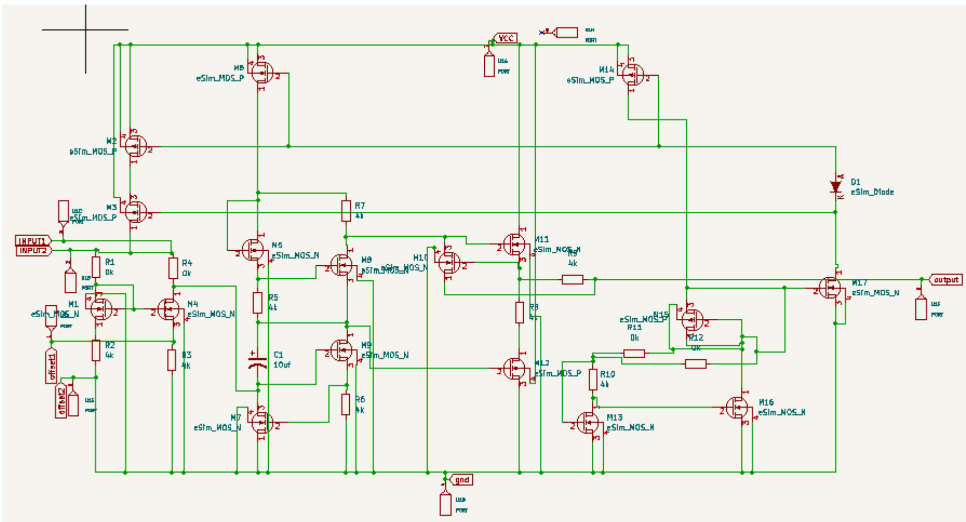

Figure 8.1: Schmatic of IC TL031a

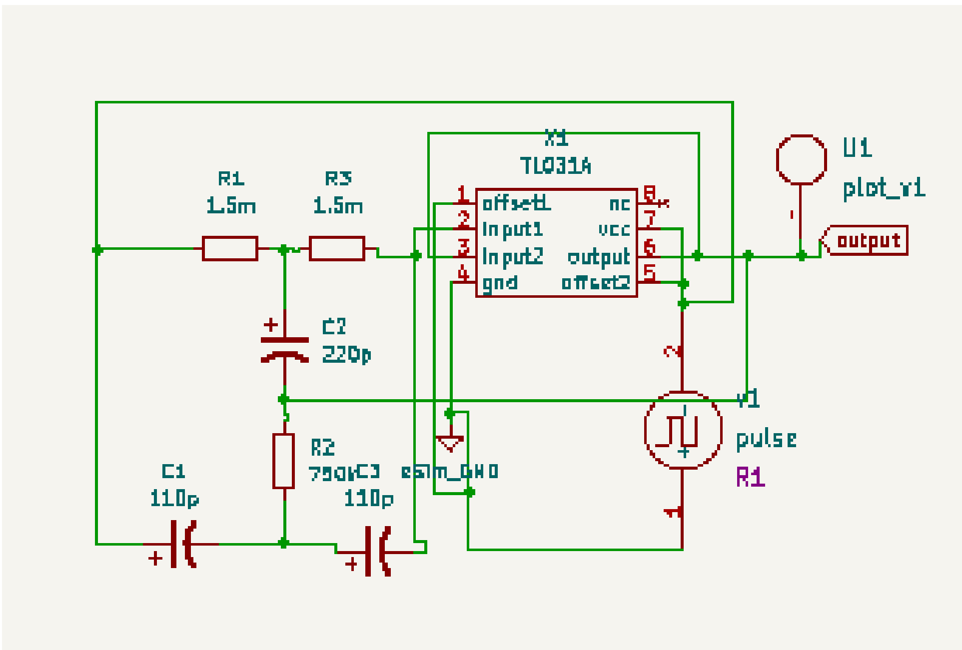

Figure 8.2: Test Circuit of IC TL031a

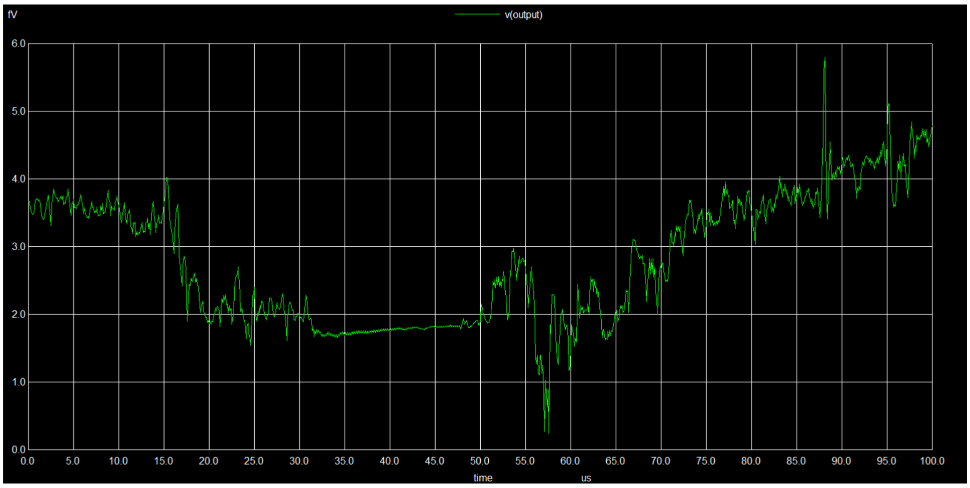

Figure 8.3: OUTPUT of Test circuit of IC TL031a

amplification.The simulated output waveform showed clean, amplified signals with minimal distortion, demonstrating proper operation of the TL031 IC. The high input impedance prevented loading of the signal source, and low noise performance ensured accurate amplification of weak signals, validating the IC's suitability for precision analog applications.

8.9 Conclusion

The TL031 JFET input operational amplifier demonstrated high input impedance and low-noise signal amplification. The output waveform remained stable and distortion free, making it suitable for sensor-based and audio signal processing applications.

Chapter 9 CA3020 – Operational Amplifier

9.1 Introduction

CA3020 [11] is a general purpose operational amplifier designed for AC and DC signal amplification. It provides high gain and wide bandwidth, making it suitable for various analog signal processing applications. In this work, CA3020 IC is studied and its performance is verified using eSim simulation software [2].

9.2 Description

CA3020 [11] is a high gain amplifier capable of amplifying both low frequency and high frequency signals. The IC supports a wide range of supply voltages and can be used in multiple amplifier configurations such as inverting, non-inverting and voltage follower.Due to its wide frequency response and low distortion, it is suitable for both audio and control system applications.

9.3 Schematic

The schematic of CA3020 IC is implemented using eSim software [2]. The IC is configured in non-inverting amplifier mode using external resistors to set the desired gain.Simulation is performed to observe stable amplification of the input signal. The test circuit consists of:

- Input signal generator
- Feedback resistor network
- Load resistor
- Dual power supply connections

9.4 Working

The input signal is applied to the non-inverting terminal while negative feedback is provided to the inverting terminal through external resistors. The feedback stabilizes the gain and ensures linear operation of the amplifier. Due to the high open loop gain of CA3020 [11], even small input voltages are amplified accurately, and the output voltage follows the theoretical gain equation of operational amplifier circuits.

9.5 Output

The output waveform of CA3020 IC [11] is observed using eSim simulation software [2]. The output signal is an amplified version of the input waveform with same frequency and shape, indicating correct linear amplification.No clipping or distortion is observed within the operating voltage range, which confirms stable performance of the amplifier. The measured gain from simulation matches closely with calculated gain based on resistor values.

9.6 Advantages

- **High gain:** Enables effective amplification of weak signals.
- **Wide bandwidth:** Supports both AC and DC signal processing.
- **Low distortion:** Maintains signal integrity.
- **Flexible configurations:** Supports multiple amplifier circuit types.

9.7 Applications

- **Amplifier Circuits:** Used for voltage amplification in analog systems.
- **Oscillator Circuits:** Used to generate sinusoidal waveforms.
- **Signal Conditioning:** Prepares sensor signals for processing.
- **Control Systems:** Used in feedback and regulation circuits.

9.8 Simulation

The CA3020 operational amplifier IC [11] was simulated in eSim software [2] to verify its performance as a general-purpose amplifier. A non-inverting amplifier configuration was chosen to observe stable voltage gain and low distortion output.A low-amplitude AC input signal was applied to the non-inverting terminal, while the inverting terminal was connected to a resistor network providing negative feedback. This feedback ensures linear operation and sets the desired closed-loop gain of the amplifier.The IC was powered using a dual supply to allow proper amplification of both positive and negative portions of the input signal. A load resistor was connected at the output to observe voltage amplification.The simulation results showed a clean amplified output waveform

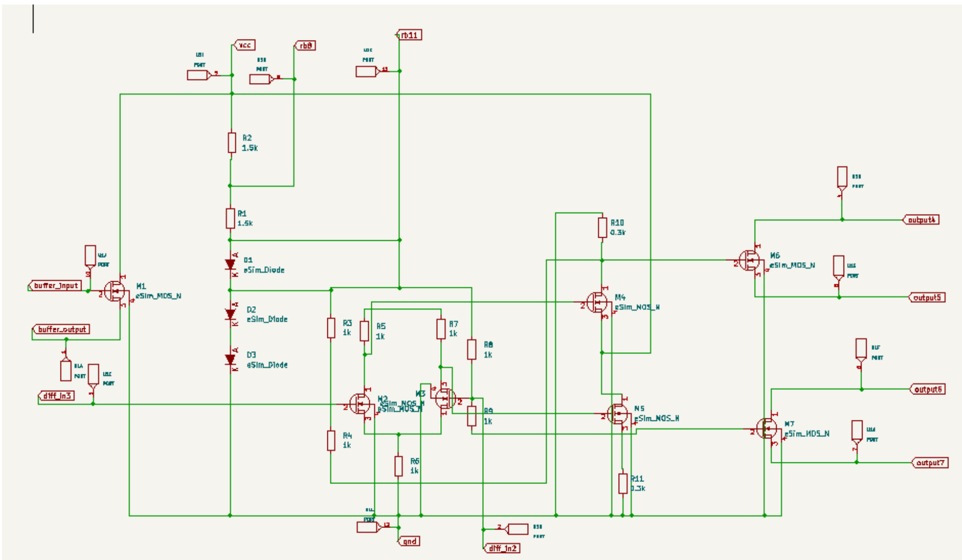

Figure 9.1: Schmatic of IC CA3020

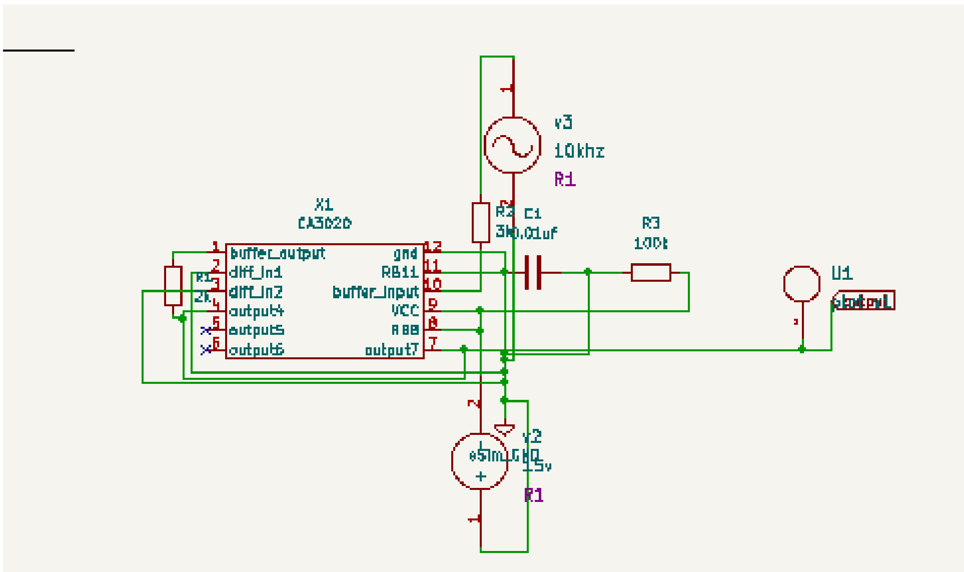

Figure 9.2: Test Circuit of IC CA3020

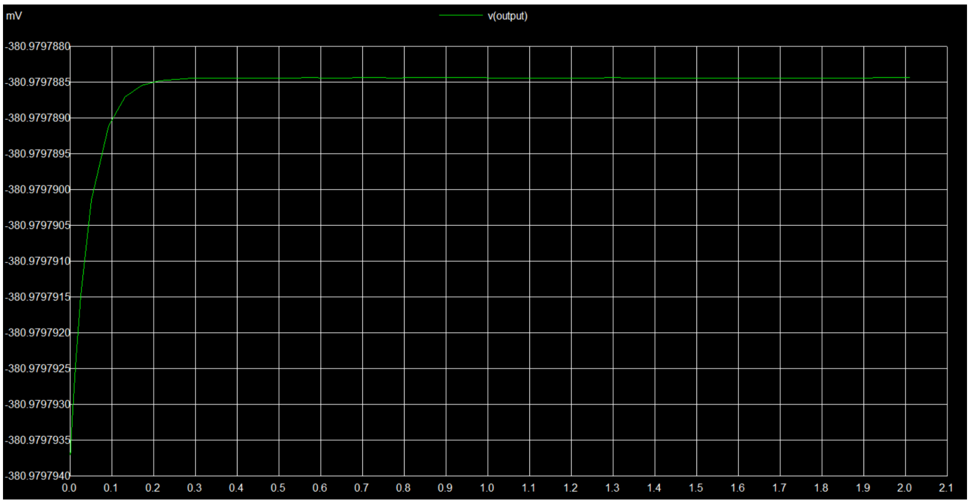

Figure 9.3: OUTPUT of Test circuit of IC CA3020

matching the input signal frequency, confirming proper operation of the IC. The results validate the CA3020's suitability for AC and DC amplification, oscillators, and general-purpose analog signal processing applications.

9.9 Conclusion

The CA3020 operational amplifier provided reliable amplification with good frequency response. Simulation results confirmed stable operation for both AC and DC signals, making it suitable for general-purpose analog circuit designs.

Chapter 10 UA709C – High Gain Operational Amplifier

10.1 Introduction

UA709C [12] is a high gain and high slew rate operational amplifier designed for fast and accurate analog signal amplification. It is suitable for precision and control system applications where rapid response and stable gain are required. In this work, UA709C IC is studied and simulated using eSim software [2].

10.2 Description

UA709C [12] provides very high open loop gain which allows accurate amplification of small differential input signals. It also supports high slew rate, enabling the IC to respond quickly to fast changing input signals without distortion.The IC requires proper frequency compensation using external components to ensure stable operation at high gains, which is important in high speed applications.

10.3 Schematic

The schematic of UA709C IC is implemented using eSim software [2]. The IC is configured in non-inverting amplifier mode with external compensation capacitor to maintain stability.Simulation is performed to observe stable amplification and transient response. The test circuit consists of:

- Input signal source
- Feedback resistor network
- Compensation capacitor
- Load resistor and dual power supply

10.4 Working

The input signal is applied to the non-inverting terminal while negative feedback is provided to the inverting terminal through resistors. The compensation capacitor controls the frequency response and prevents oscillations. Due to high internal gain of UA709C [12], the closed loop gain is accurately defined by external resistors and the output follows the input signal with fast response.

10.5 Output

The output waveform of UA709C IC [12] is observed using eSim simulation software [2]. The output shows amplified waveform with sharp rising and falling edges, confirming high slew rate operation.No oscillations or ringing are observed due to proper compensation, indicating stable amplifier performance. The gain measured from simulation matches theoretical values, validating correct circuit design.

10.6 Advantages

- **Very high gain:** Suitable for amplifying small signals accurately.
- **High slew rate:** Enables fast response to rapid signal changes.
- **Low offset voltage:** Improves precision in DC applications.
- **Wide application range:** Suitable for analog and control systems.

10.7 Applications

- **Precision Instrumentation:** Used in measurement and data acquisition systems.
- **High Gain Amplifiers:** Suitable for multi-stage amplification circuits.
- **Control Systems:** Used in feedback and regulation applications.
- **Signal Processing:** Used in fast analog signal conditioning circuits.

10.8 Conclusion

The UA709C operational amplifier exhibited high gain and fast response in simulation. The IC performed well in high-gain configurations and is suitable for precision and control system applications where rapid signal processing is required.

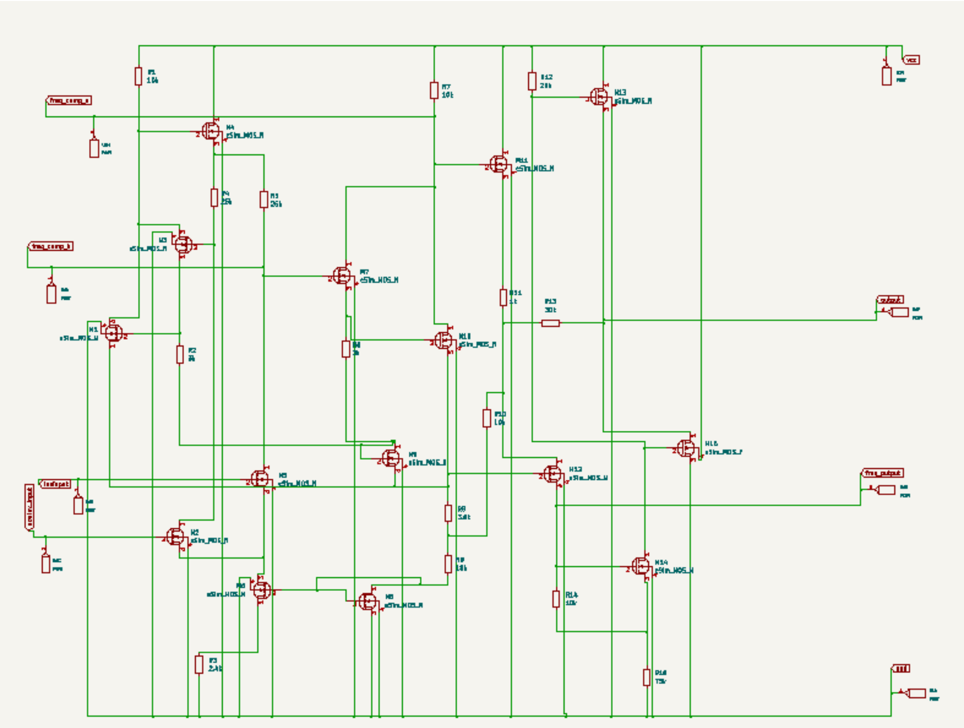

Figure 10.1: Schmatic of IC UA709c

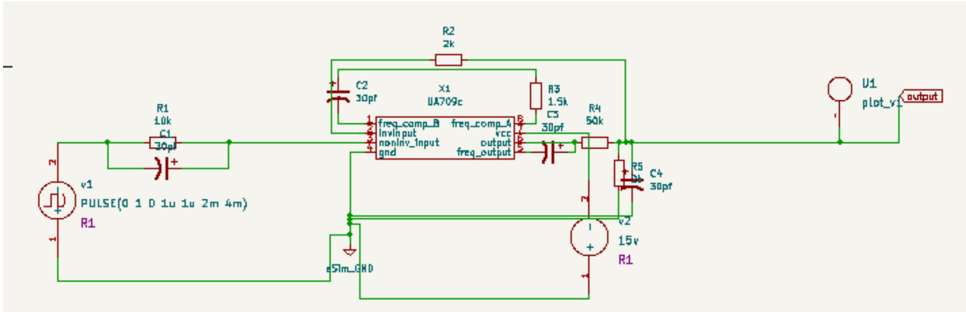

Figure 10.2: Test Circuit of IC UA709c

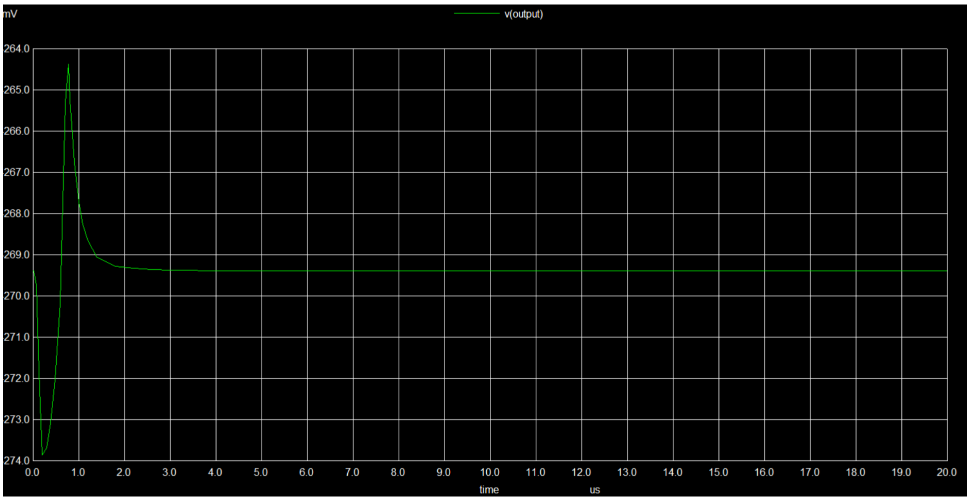

Figure 10.3: OUTPUT of Test Circuit of IC UA709c

Chapter 11 UA702M – High Speed Operational Amplifier

11.1 Introduction

UA702M [13] is a high speed operational amplifier designed for wide bandwidth and fast transient response applications. It is one of the early high performance monolithic op-amps used in circuits requiring rapid signal amplification. In this internship, the UA702M IC was modeled using the Subcircuit Builder in eSim software [2] and its performance was verified using simulation test circuits.

11.2 Description

UA702M [13] provides high open loop gain along with wide frequency response, making it suitable for high speed analog signal processing. The IC supports external frequency compensation which allows the designer to control stability and bandwidth as per application requirement.Due to its high slew rate, the amplifier can respond quickly to sudden changes in input signal without distortion, which is essential in pulse and waveform shaping circuits.

11.3 Schematic

The UA702M IC is modeled using Subcircuit Builder in eSim software [2] by defining its internal parameters and pin configuration. The IC is then used in amplifier configuration for testing. Simulation is performed to observe amplification and transient response behavior. The test circuit consists of:

- High frequency input signal source
- Feedback resistor network for gain control
- External compensation capacitor
- Load resistor and dual power supply

11.4 Working

The input signal is applied to the non-inverting terminal and negative feedback is provided to the inverting terminal through external resistors. External compensation capacitor controls the stability and frequency response of the amplifier. Due to the high internal gain of UA702M [13], the closed loop gain is defined by external components and the output follows the input with fast response and minimal phase delay.

11.5 Output

The output waveform of UA702M IC [13] is observed using eSim simulation software [2]. The output shows amplified waveform with steep rising and falling edges, confirming high slew rate performance.The waveform remains stable without oscillations due to proper compensation, which verifies stable operation at high frequencies. The output amplitude matches the expected gain calculated using resistor values, confirming correct amplification.

11.6 Advantages

- **High slew rate:** Suitable for fast changing signals and pulse circuits.
- **Wide bandwidth:** Supports high frequency analog applications.
- **External compensation:** Allows control over stability and frequency response.
- **High gain:** Enables accurate amplification of small signals.

11.7 Applications

- **High Speed Amplifiers:** Used in fast signal amplification systems.
- **Pulse Amplifiers:** Suitable for pulse shaping and detection circuits.
- **Waveform Shaping Circuits:** Used to improve signal edges and transitions.
- **Signal Conditioning Systems:** Prepares signals for high speed processing.

11.8 Simulation

The UA702M high-speed operational amplifier IC [13] was modeled using the Subcircuit Builder in eSim software [2] to verify its high-speed performance and wide bandwidth capabilities.A non-inverting amplifier configuration was implemented with a resistor network providing negative feedback to stabilize gain. An external compensation capacitor was connected to ensure frequency stability and prevent oscillations at high

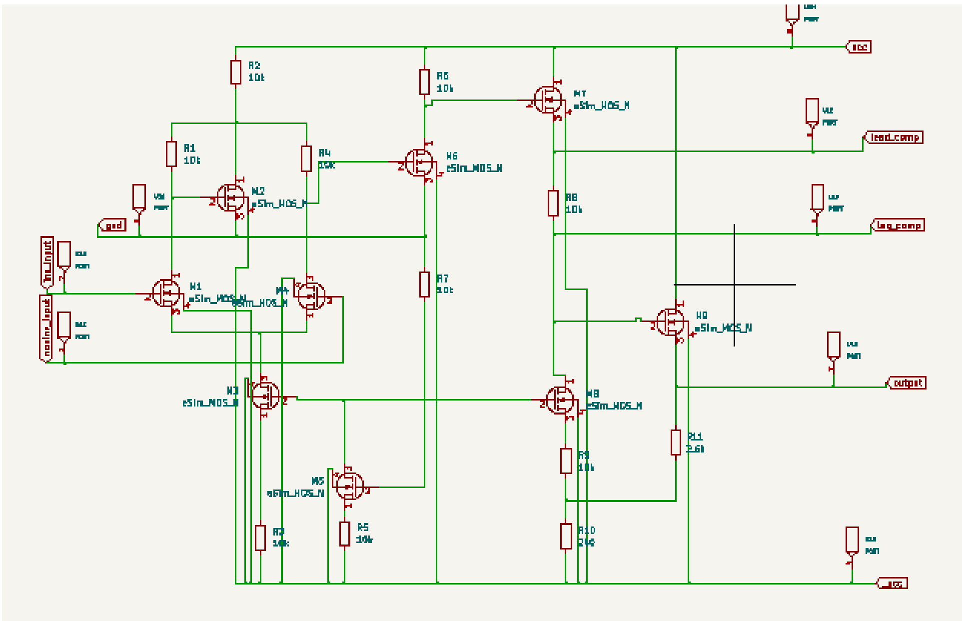

Figure 11.1: Schmatic of IC UA702m

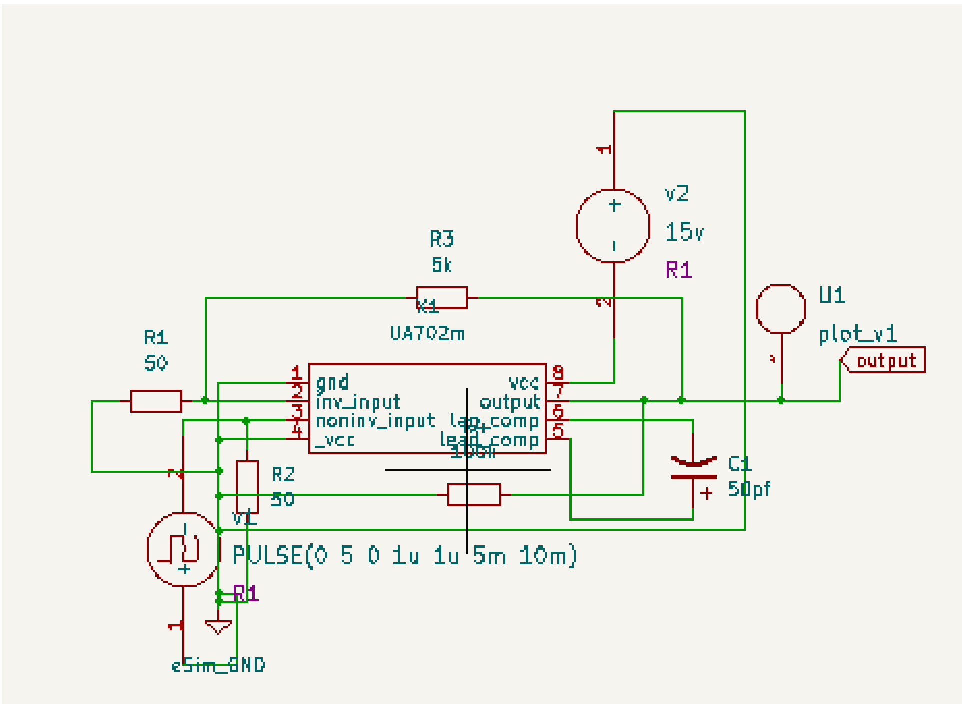

Figure 11.2: Test Circuit of ic UA702m

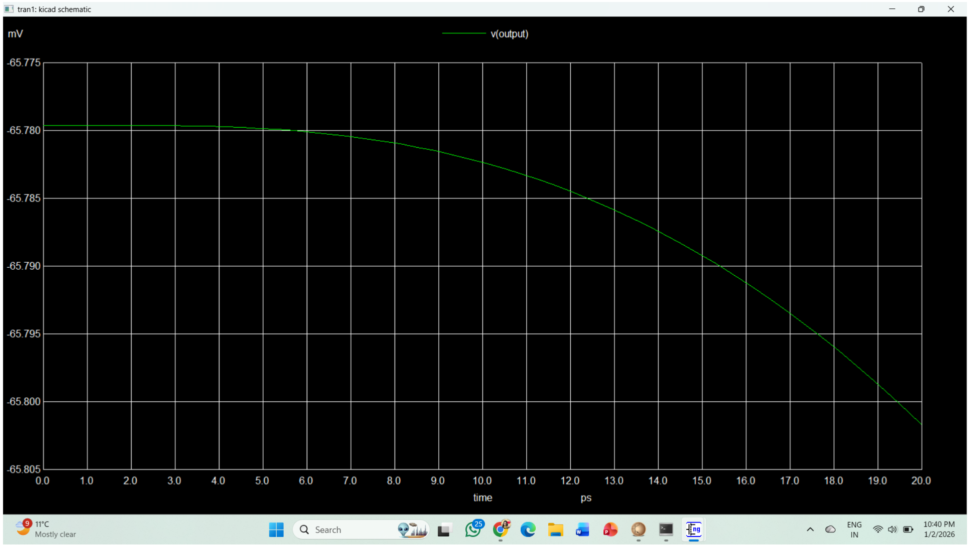

Figure 11.3: OUTPUT of the Test Circuit of IC UA702m

frequencies.A high-frequency input signal was applied to the non-inverting terminal, while the inverting terminal was connected to the feedback network. Dual power supply was used to allow proper amplification of both positive and negative portions of the signal.The output was observed using simulation probes, and the waveform demonstrated fast rising and falling edges, confirming the high slew rate of the UA702M. The amplifier maintained stable operation without oscillation or distortion, validating its suitability for high-speed amplification and waveform shaping applications.

11.9 Conclusion

The UA702M IC modeled using Subcircuit Builder in eSim was successfully validated through simulation. High-speed response and wide bandwidth operation were observed, confirming its suitability for fast signal amplification and waveform processing applications.

Chapter 12 LM3080 – Operational Transconductance Amplifier

LM3080 [14] is an operational transconductance amplifier (OTA) which converts input voltage difference into proportional output current. Unlike conventional operational amplifiers, the gain of LM3080 is controlled by an external bias current. This makes it suitable for voltage controlled amplification and analog signal processing applications. In this work, LM3080 IC is modeled using Subcircuit Builder in eSim software [2] and its performance is verified through simulation.

12.1 Description

LM3080 [14] produces an output current which is proportional to the differential input voltage multiplied by the transconductance of the amplifier. The transconductance value is controlled by an externally applied bias current. By changing the bias current, the gain of the amplifier can be electronically controlled without modifying external resistor values. This property is very useful in voltage controlled oscillators and filters.

12.2 Schematic

The LM3080 IC is modeled using Subcircuit Builder in eSim software [2]. The OTA is configured as a voltage controlled amplifier using load resistor at the output to convert output current into voltage.Simulation is to observe variation in output amplitude with change in bias current. The test circuit consists of:

- Differential input voltage source
- Bias current source for transconductance control
- Load resistor at output
- Dual power supply

12.3 Working

When a differential input voltage is applied to LM3080 [14], the IC generates output current proportional to the input voltage and bias current. The bias current controls the transconductance and hence controls the gain of the amplifier. The output current is passed through an external load resistor to produce output voltage. As the bias current increases, the transconductance increases and output amplitude also increases.Thus, electronic control of gain is achieved without changing resistor values.

12.4 Output

The output waveform of LM3080 IC [14] is observed using eSim simulation software [2].The output voltage waveform increases in amplitude when bias current is increased, while input signal remains constant. This confirms that gain is controlled by bias current, verifying correct OTA operation. The output waveform maintains same frequency as input signal, indicating linear amplification behavior. The observed results validate voltage to current conversion and external gain control functionality of LM3080.

12.5 Advantages

- **Electronic gain control:** Gain can be varied using bias current.
- **Current mode operation:** Suitable for analog computation circuits.
- **Wide application range:** Useful in oscillators, filters and modulators.
- **Compact design:** Reduces external component requirement.

12.6 Applications

- **Voltage Controlled Amplifiers:** Used in audio and communication systems.
- **Oscillator Circuits:** Frequency can be controlled electronically.
- **Active Filters:** Cutoff frequency can be tuned using bias current.
- **Analog Signal Processing:** Used in multipliers and modulators.

12.7 Simulation

The LM3080 operational transconductance amplifier (OTA) IC [14] was simulated using eSim software [2] to verify voltage-controlled amplification behavior.The IC was configured as a voltage-controlled amplifier, with the differential input voltage applied to the input terminals. A bias current source was connected to control transconductance of the amplifier externally. A load resistor was connected at the output to convert the output current into voltage for observation. Dual power supply

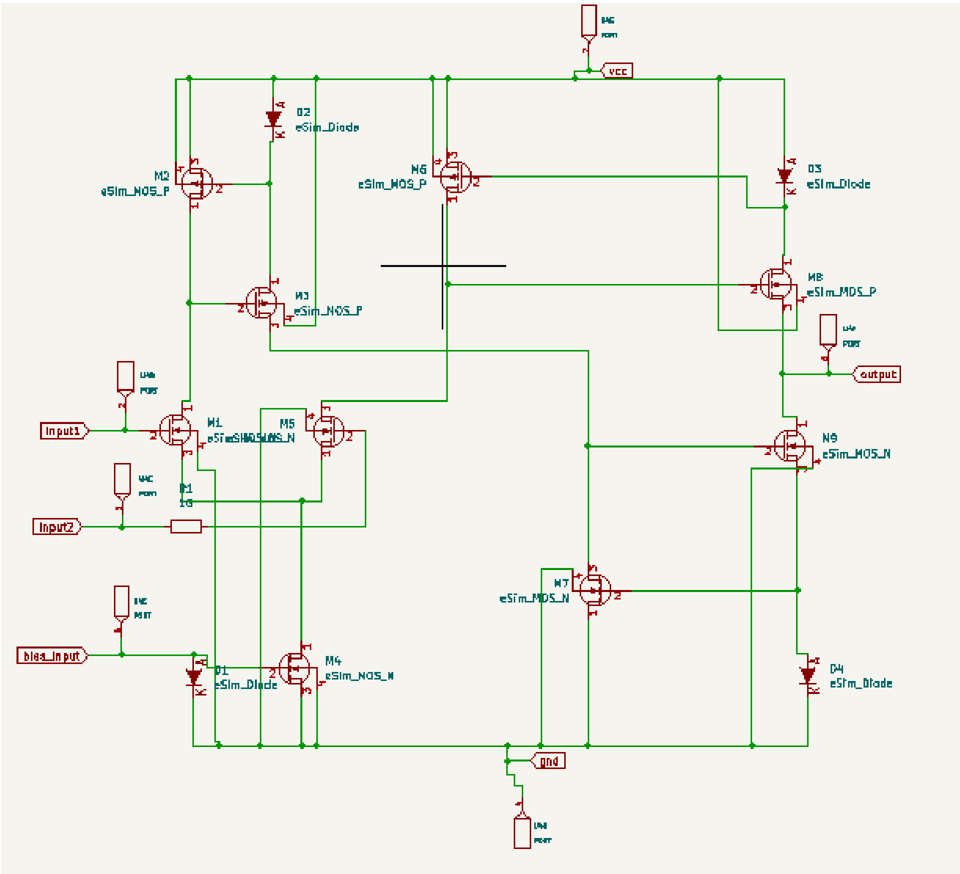

Figure 12.1: Schmatic of IC LM3080

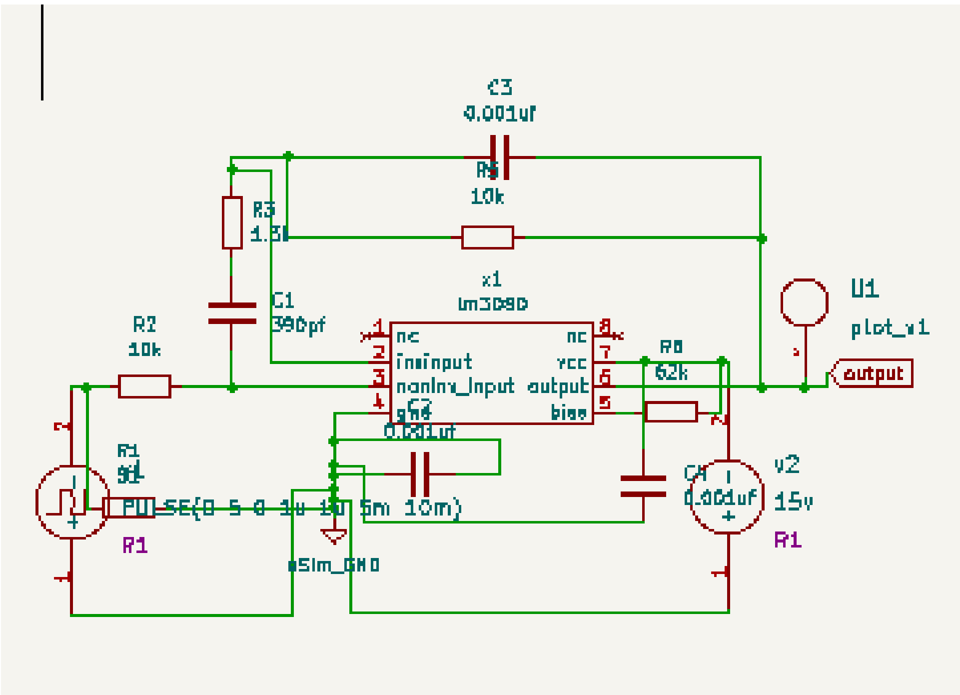

Figure 12.2: TEST circuit of IC LM3080

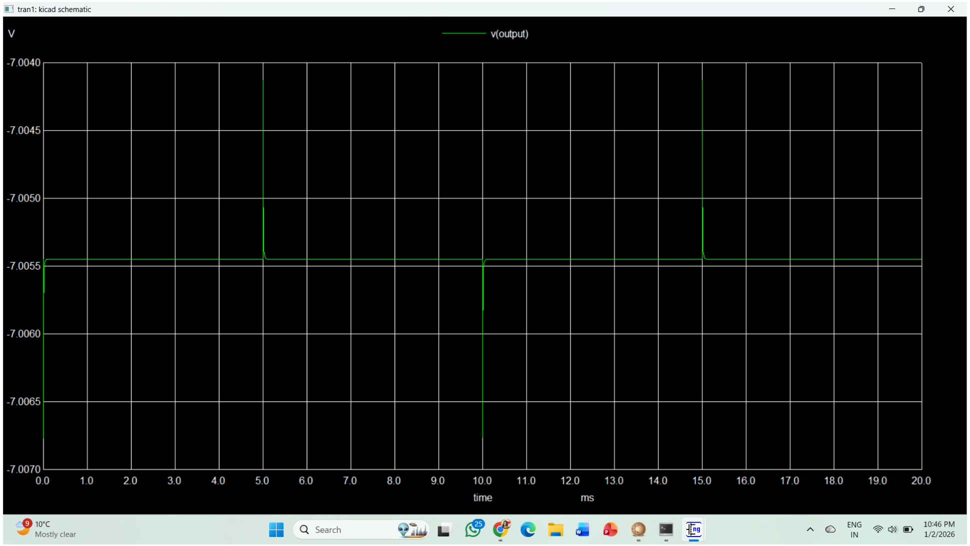

Figure 12.3: OUTPUT of IC LM3080

was provided to ensure proper operation of the OTA. The output waveform was monitored using simulation probes.Simulation results showed that the output amplitude varied according to changes in the bias current, while the frequency of the input signal remained unchanged.This confirms that the LM3080 IC accurately converts input voltage into output current and allows electronic control of gain. The IC operates linearly and stably, making it suitable for voltage-controlled amplifiers, modulators, oscillators, and other analog signal processing applications.

12.8 Conclusion

The LM3080 operational transconductance amplifier demonstrated voltage-controlled gain behavior as expected. The output current varied according to bias current, confirming proper OTA operation. This IC is suitable for voltage-controlled and analog signal processing applications.

Chapter 13 Conclusion and Future Scope

The eSim internship has provided me with a solid foundation in electronic circuit design and simulation. Through hands-on experience, I gained practical knowledge in creating schematics, designing subcircuits for ICs, mapping SPICE models, generating netlists, and performing accurate circuit simulations. Working with the Subcircuit Builder allowed me to model ICs using discrete components, which could then be packaged and reused in larger circuit designs.

This internship significantly enhanced my technical proficiency with EDA tools while highlighting the importance of open-source solutions in enabling accessible, cost-effective learning. The structured workflow of eSim helped me understand the complete electronic design automation process, from schematic capture to simulation and verification. Additionally, exposure to debugging techniques and simulation analysis has strengthened my confidence in applying these skills to real-world electronics projects.

There is considerable scope for further work in research and development of circuit simulation tools. Improvements can be made in increasing the accuracy of simulations, expanding the library of reusable subcircuits, and integrating more advanced features for mixed-signal designs. In the future, contributions can be made directly to eSim by identifying bugs, suggesting enhancements, or implementing new modules to improve usability and performance.

Moreover, enhancing user guidance within eSim—for example, by providing clearer, more precise error messages that indicate the type and location of design issues—can greatly help users debug circuits efficiently and effectively. Overall, this internship has been an enriching experience that not only strengthened my technical skills but also motivated me to contribute meaningfully to the open-source electronics community.

Bibliography

[1] FOSSEE Project, IIT Bombay. *Free and Open Source Software for Education*. Available online: `https://fossee.in`

[2] eSim – Open Source EDA Tool, FOSSEE, IIT Bombay. Available online: `https://esim.fossee.in`

[3] Texas Instruments. *LM565 Phase Locked Loop Datasheet*. Available online: https://www.ti.com/lit/pdf/snoa651

[4] Texas Instruments. *LM393A Dual Voltage Comparator Datasheet*. Available online: https://www.ti.com/lit/ds/symlink/lm393.pdf

[5] Texas Instruments. *NE556 Dual Timer Datasheet*. Available online: https://www.ti.com/lit/gpn/NA556

[6] ON Semiconductor. *MC1496A Balanced Modulator/Demodulator Datasheet*. Available online: https://www.onsemi.com/download/data-sheet/pdf/mc1496-d.pdf

[7] Texas Instruments. *LM308A Precision Operational Amplifier Datasheet*. Available online: https://www.alldatasheet.com/datasheet-pdf/pdf/264/TI/LM308.html

[8] Texas Instruments. *LM108A Operational Amplifier Datasheet*. Available online: https://www.alldatasheet.com/datasheet-pdf/pdf/283/TI/LM108.html

[9] Texas Instruments. *TL031 Low Noise JFET Input Operational Amplifier Datasheet*. Available online: https://www.ti.com/lit/ds/symlink/tl031.pdf

[10] RCA Semiconductor. *CA3020 Operational Amplifier Datasheet*. Available online: https://www.alldatasheet.com/datasheet-pdf/pdf/12419/RCA/CA3020.html

[11] Texas Instruments. *UA709C High Gain Operational Amplifier Datasheet*. Available online: https://www.alldatasheet.com/datasheet-pdf/pdf/1213/TI/UA709.html

[12] Intersil Corporation. *CA3080 Operational Transconductance Amplifier Datasheet*. Available online: https://www.alldatasheet.com/view.jsp?Searchword=CA3080