eSim Internship Report

IC Subcircuit Development and Simulation

Submitted by

Pandiya Rajan S

B.E Electronics and Communication Engineering
Chennai Institute of Technology, Chennai

Under the guidance of

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

June 2026

Acknowledgment

I express my sincere gratitude to Prof. Prabhu Ramachandran for providing me with the opportunity to be part of the FOSSEE internship program and for his continued efforts in promoting open-source engineering tool development. His leadership and vision have been instrumental in fostering meaningful student participation in the open-source ecosystem.

I also acknowledge Prof. Kannan M. Moudgalya for his foundational role in establishing and strengthening the FOSSEE initiative. His contributions to open-source education and the development of the FOSSEE fellowship framework have been pivotal in creating the academic and organisational platform through which this internship was undertaken.

My sincere appreciation extends to my mentor, Sumanto Kar, for his continual support, technical guidance, and encouragement throughout the duration of this project. His insights and feedback played a key role in refining ideas, overcoming challenges, and ensuring timely completion of the tasks assigned to me.

I would also like to thank my internal mentors, Mr. Varad Patil and Ms. Shanthi Priya K, for their valuable guidance, coordination, and technical inputs during the internship. Their mentorship contributed significantly to the clarity, progress, and successful execution of the work.

This internship has been an enriching learning experience, allowing me to work closely with open-source EDA tools, develop IC subcircuits in eSim, and gain exposure to real-world circuit modeling and simulation workflows. The knowledge acquired during this period will undoubtedly support my future academic and professional pursuits.

I would also like to thank the entire FOSSEE team for their coordination, assistance, and timely interactions at various stages of this work. Their collective efforts ensured smooth workflow, resource accessibility, and effective project execution.

Contents

Chapter 1

Introduction

1.1 About FOSSEE

FOSSEE (Free/Libre and Open Source Software for Education) is a project promoted by the National Mission on Education through Information and Communication Technology (NMEICT), Ministry of Education, Government of India. The main objective of the FOSSEE project is to encourage the use of Free and Open Source Software (FOSS) tools in academic institutions and improve the quality of education through open-source technologies. The project is coordinated at IIT Bombay and supports various open-source software platforms used in engineering, science, and education.

FOSSEE conducts workshops, internships, training programs, and research activities to help students and educators gain practical exposure to open-source tools. It also provides opportunities for students to contribute to real-world projects involving software development, circuit design, simulation, documentation, and VLSI-related activities.

1.2 About eSim

eSim is an open-source Electronic Design Automation (EDA) tool developed by the FOSSEE project, IIT Bombay. It is mainly used for circuit design, simulation, analysis, and PCB development. eSim integrates various open-source software tools such as KiCad, Ngspice, and Scilab to provide a complete environment for electronic circuit simulation and testing.

Using eSim, users can create schematic diagrams, perform SPICE simulations, analyze waveforms, and design PCB layouts efficiently. The software supports analog, digital, and mixed-signal circuit simulations, making it useful for students, researchers, and engineers.

1.3 Objectives of the Internship

The main objective of this internship was to gain practical knowledge and hands-on experience in electronic circuit simulation and IC subcircuit development using the eSim tool. The internship focused on understanding SPICE modeling techniques, analyzing IC datasheets, and implementing accurate subcircuit models for simulation purposes.

Another important objective was to contribute to the open-source ecosystem by developing and verifying IC subcircuits that can be used by students and researchers in future projects. The internship also aimed to improve technical skills related to circuit analysis, debugging, simulation verification, and documentation.

Throughout the internship, multiple IC subcircuits were successfully designed and tested using eSim and Ngspice simulation tools.

Chapter 2

Literature Survey

eSim is an open-source Electronic Design Automation (EDA) tool developed under the FOSSEE project at IIT Bombay. It is designed for schematic capture, circuit simulation, PCB design, and analysis of electronic circuits. eSim integrates several open-source tools such as KiCad, Ngspice, and Scilab to provide a complete environment for electronic system design.

2.1 Ngspice

Ngspice is an open-source mixed-signal circuit simulator widely used for analog and digital circuit analysis. It is based on the SPICE simulation engine and is integrated with eSim for performing circuit simulations and waveform analysis.

In this internship, Ngspice played a major role in validating the behavior of implemented IC subcircuits. The simulator helped in debugging errors, verifying output waveforms, and checking the functionality of SPICE models developed during the internship.

2.2 KiCad

KiCad is an open-source software suite used for schematic capture and PCB design. It is integrated with eSim to provide a user-friendly interface for designing electronic circuits.

The schematic editor in KiCad was used during the internship to design and verify electronic circuits associated with the IC subcircuits.

2.3 SPICE Subcircuits

SPICE subcircuits are reusable circuit models used to represent integrated circuits and complex electronic components in simulation environments. Subcircuits simplify circuit design by allowing designers to use predefined models instead of creating the entire internal circuitry repeatedly.

Chapter 3

Overview of IC Subcircuits

3.1 Introduction to IC Subcircuits

Integrated Circuits (ICs) are widely used in modern electronic systems for performing various analog and digital operations. SPICE subcircuits are commonly used to model integrated circuits in simulation environments such as eSim and Ngspice.

A subcircuit is a reusable block of circuit components defined using SPICE syntax. It contains internal connections, electronic components, and functional behavior of an IC.

3.2 Need for IC Subcircuits

IC subcircuits are essential for accurate electronic circuit simulations. Subcircuits are useful for:

- Reusing IC models in multiple circuit designs
- Simplifying schematic creation
- Improving simulation accuracy
- Reducing design time
- Testing circuit behavior before hardware implementation

3.3 SPICE Modeling of ICs

SPICE (Simulation Program with Integrated Circuit Emphasis) is widely used for electronic circuit simulation. IC subcircuits are implemented using the `.SUBCKT` command in SPICE syntax.

The SPICE model includes:

- Input and output pins
- Internal circuit connections

- Passive and active components
- Electrical characteristics

3.4 Implementation in eSim

The implementation of IC subcircuits was carried out using the eSim platform. The process involved studying datasheets, writing SPICE subcircuit definitions, integrating the models into eSim libraries, and verifying the output through simulations.

A total of nine IC subcircuits were successfully implemented and verified during the internship period.

Chapter 4

Tools and Technologies Used

4.1 eSim

eSim is an open-source Electronic Design Automation (EDA) tool developed by the FOSSEE project at IIT Bombay. It is used for schematic design, circuit simulation, waveform analysis, and PCB design.

4.2 Ngspice

Ngspice is an open-source SPICE-based circuit simulator used for analog, digital, and mixed-signal simulations. It is one of the core simulation engines integrated with eSim.

4.3 KiCad

KiCad is an open-source software suite used for electronic schematic capture and PCB design. KiCad is integrated with eSim and works together with Ngspice for circuit simulation purposes.

4.4 LaTeX and Overleaf

LaTeX is a document preparation system widely used for technical and scientific documentation. Overleaf is an online LaTeX editor that provides a collaborative environment for creating professional reports and research papers. In this internship, Overleaf was used for preparing and formatting the internship report.

4.5 TLC2264

4.5.1 Schematic Symbol Creation

The schematic symbol of TLC2264 was created in eSim using the symbol editor. Proper pin names, pin numbers, and electrical connections were assigned according to the datasheet specifications.

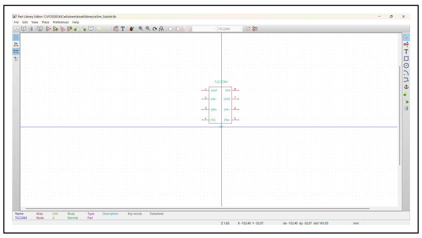

Figure 4.1: Schematic symbol of TLC2264

4.5.2 Subcircuit Schematic Design

The internal schematic representation of the TLC2264 subcircuit was implemented and integrated into the eSim environment. The design included SPICE model connections and internal circuit elements required for simulation.

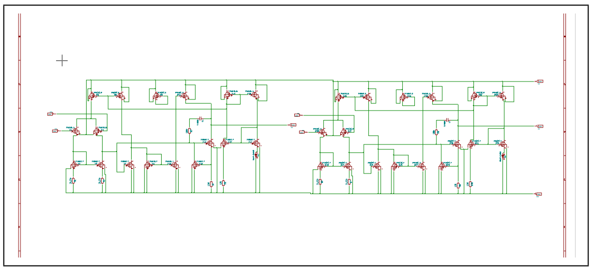

Figure 4.2: Subcircuit schematic of TLC2264

4.5.3 Test Bench Circuit Design

A test bench circuit was designed to verify the behavior of the implemented subcircuit under different operating conditions. The test setup included input sources, passive components, and output measurement points.

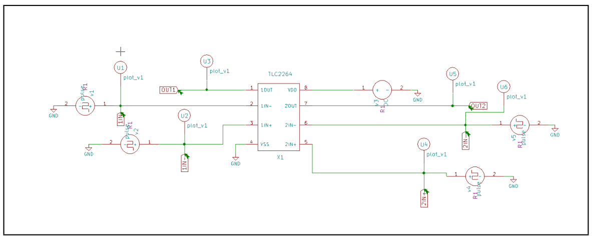

Figure 4.3: Test bench circuit of TLC2264

4.5.4 Simulation Results

The implemented TLC2264 subcircuit was simulated using Ngspice integrated with eSim. The obtained output waveforms matched the expected operational characteristics mentioned in the datasheet.

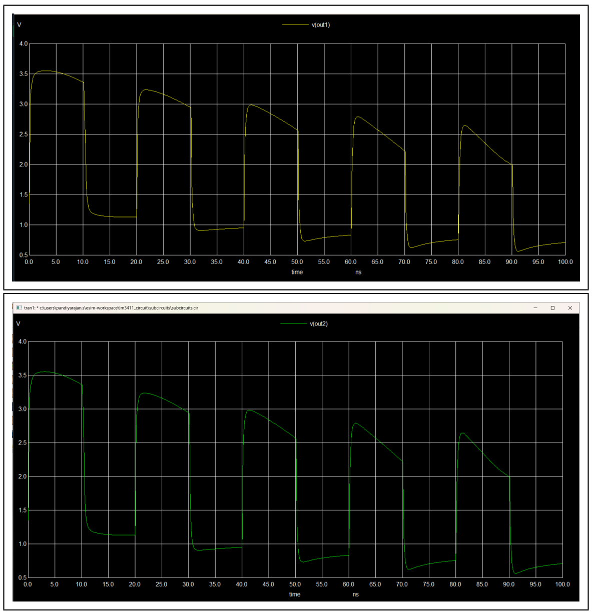

Figure 4.4: Simulation output of TLC2264

4.6 LM386N

4.6.1 Schematic Symbol Creation

The schematic symbol of LM386N was created in eSim according to the datasheet specifications with proper pin assignments and electrical connections.

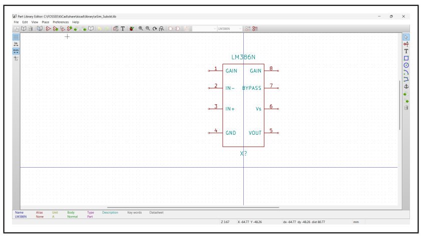

Figure 4.5: Schematic symbol of LM386N

4.6.2 Subcircuit Schematic Design

The internal SPICE subcircuit schematic of LM386N was implemented and integrated into the eSim environment for simulation purposes.

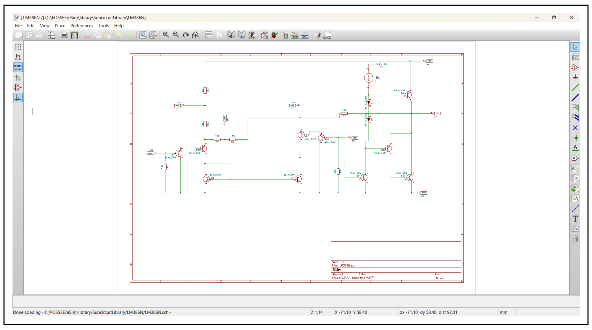

Figure 4.6: Subcircuit schematic of LM386N

4.6.3 Test Bench Circuit Design

A test bench circuit was designed to validate the amplification behavior of the implemented LM386N subcircuit.

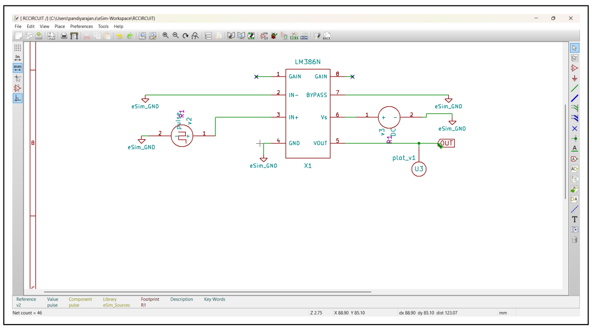

Figure 4.7: Test bench circuit of LM386N

4.6.4 Simulation Results

The implemented LM386N subcircuit was simulated using Ngspice integrated with eSim, and the obtained waveforms verified the expected output behavior.

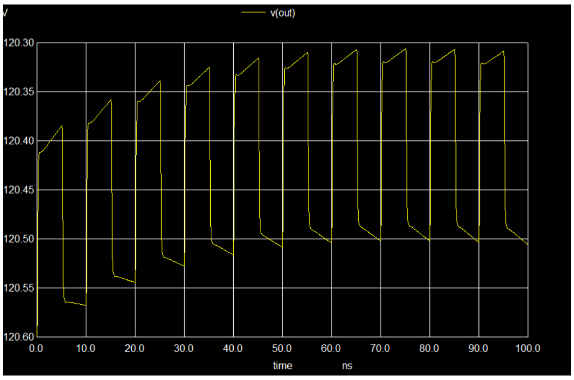

Figure 4.8: Simulation output of LM386N

4.7 LM3411

4.7.1 Schematic Symbol Creation

The schematic symbol of LM3411 was created in eSim with proper pin configuration according to the datasheet specifications.

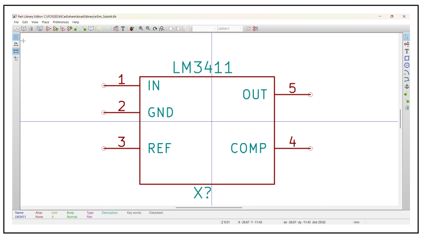

Figure 4.9: Schematic symbol of LM3411

4.7.2 Subcircuit Schematic Design

The SPICE subcircuit schematic of LM3411 was developed and integrated into the eSim simulation environment.

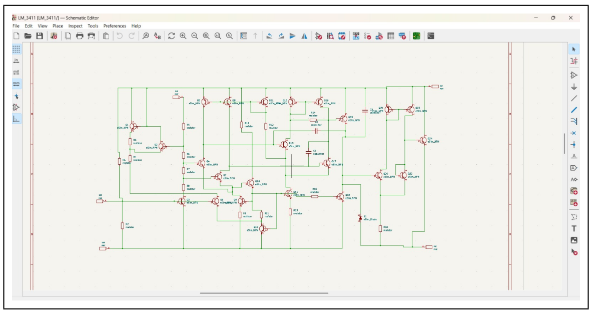

Figure 4.10: Subcircuit schematic of LM3411

4.7.3 Test Bench Circuit Design

A suitable test bench circuit was designed to verify the functionality of the implemented LM3411 subcircuit.

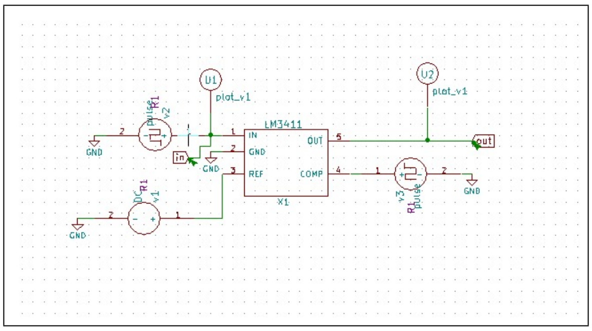

Figure 4.11: Test bench circuit of LM3411

4.7.4 Simulation Results

The simulation outputs verified the operational behavior of the LM3411 SPICE subcircuit model.

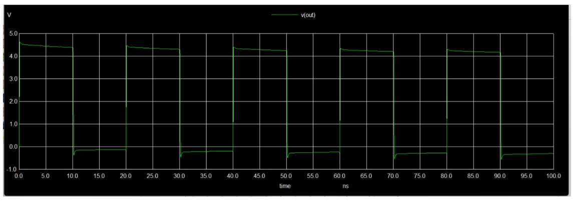

Figure 4.12: Simulation output of LM3411

4.8 MC1556

4.8.1 Schematic Symbol Creation

The schematic symbol of MC1556 was designed in eSim according to the datasheet information.

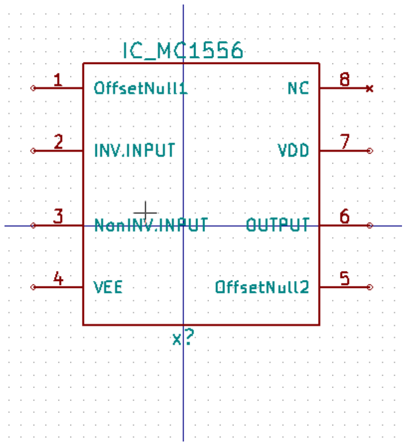

Figure 4.13: Schematic symbol of MC1556

4.8.2 Subcircuit Schematic Design

The SPICE schematic implementation of MC1556 was developed and integrated into eSim for simulation analysis.

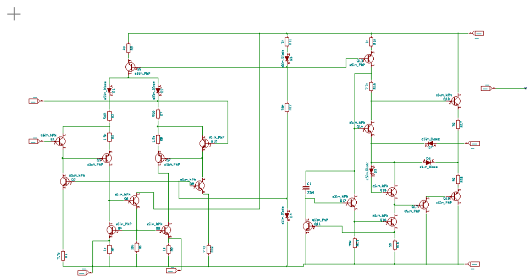

Figure 4.14: Subcircuit schematic of MC1556

4.8.3 Test Bench Circuit Design

A test bench circuit was created to validate the operational characteristics of the MC1556 subcircuit.

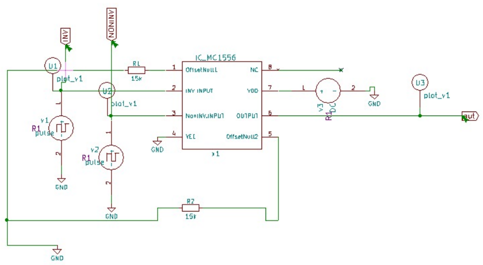

Figure 4.15: Test bench circuit of MC1556

4.8.4 Simulation Results

Simulation results confirmed the correct operation of the implemented MC1556 SPICE model.

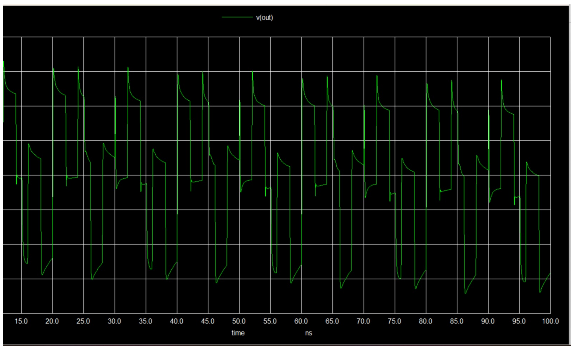

Figure 4.16: Simulation output of MC1556

4.9 SDA12

4.9.1 Schematic Symbol Creation

The schematic symbol of SDA12 was implemented in eSim according to datasheet specifications.

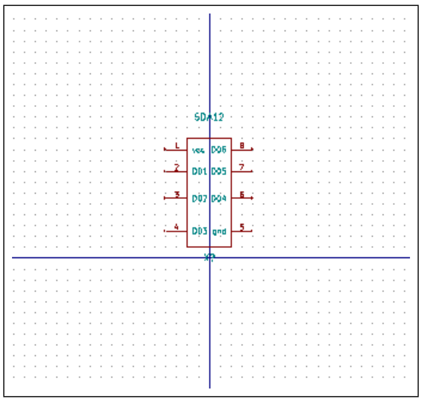

Figure 4.17: Schematic symbol of SDA12

4.9.2 Subcircuit Schematic Design

The SPICE schematic model of SDA12 was created and integrated into the eSim environment.

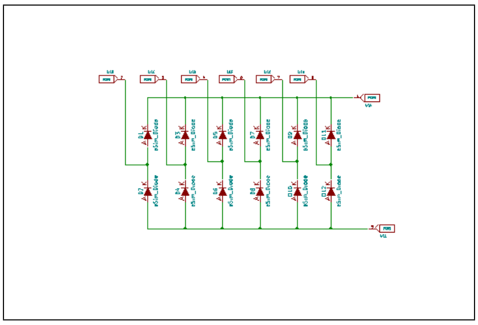

Figure 4.18: Subcircuit schematic of SDA12

4.9.3 Test Bench Circuit Design

A suitable test bench was designed to verify the implemented SDA12 subcircuit model.

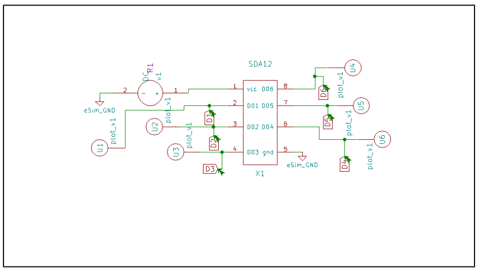

Figure 4.19: Test bench circuit of SDA12

4.9.4 Simulation Results

Simulation outputs verified the functionality and correctness of the SDA12 SPICE implementation.

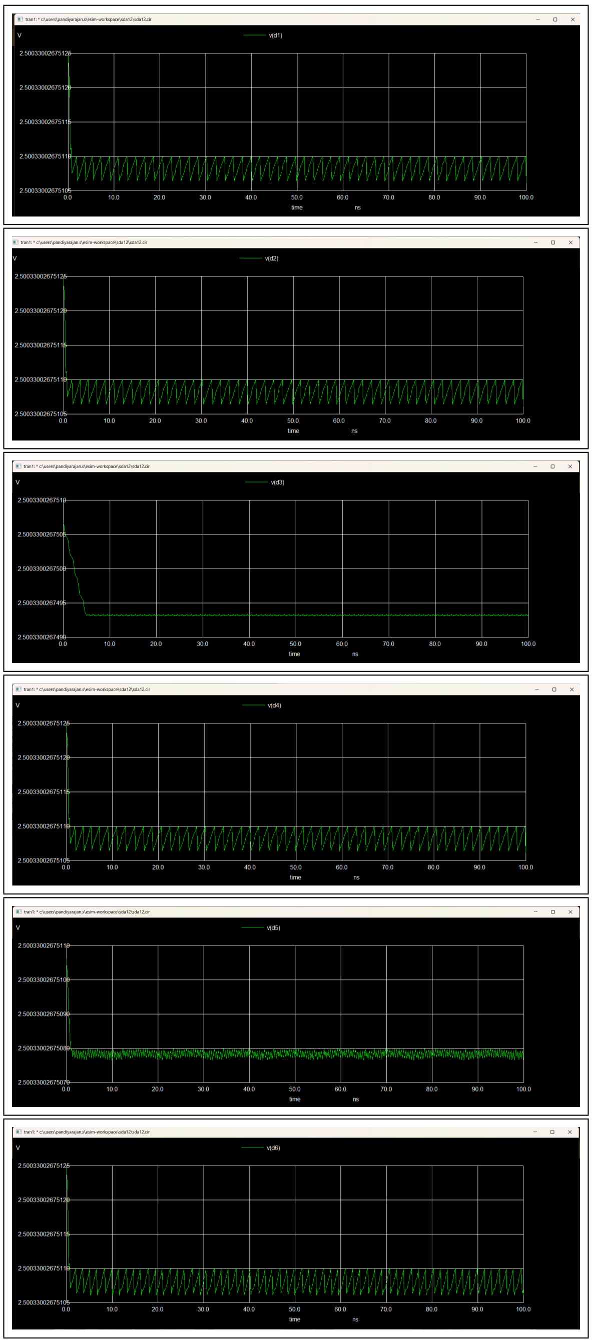

Figure 4.20: Simulation output of SDA12

4.10 TL317

4.10.1 Schematic Symbol Creation

The schematic symbol of TL317 was created in eSim using proper terminal assignments according to the datasheet specifications.

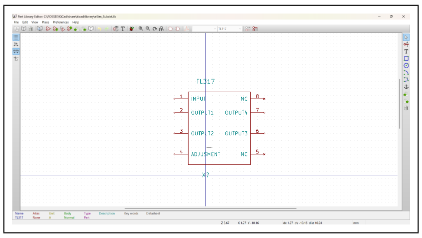

Figure 4.21: Schematic symbol of TL317

4.10.2 Subcircuit Schematic Design

The SPICE subcircuit schematic of TL317 was implemented and integrated into the eSim simulation environment for voltage regulation analysis.

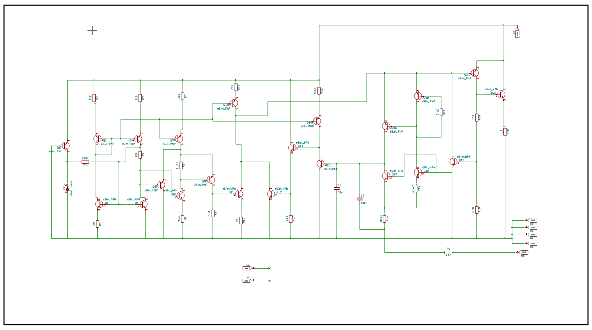

Figure 4.22: Subcircuit schematic of TL317

4.10.3 Test Bench Circuit Design

A test bench circuit was designed to validate the output voltage regulation characteristics of the TL317 IC under different operating conditions.

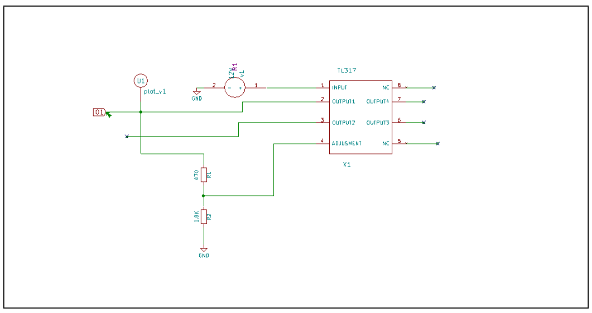

Figure 4.23: Test bench circuit of TL317

4.10.4 Simulation Results

The implemented TL317 subcircuit was simulated using Ngspice integrated with eSim. The obtained output verified the expected voltage regulation characteristics according to the datasheet specifications.

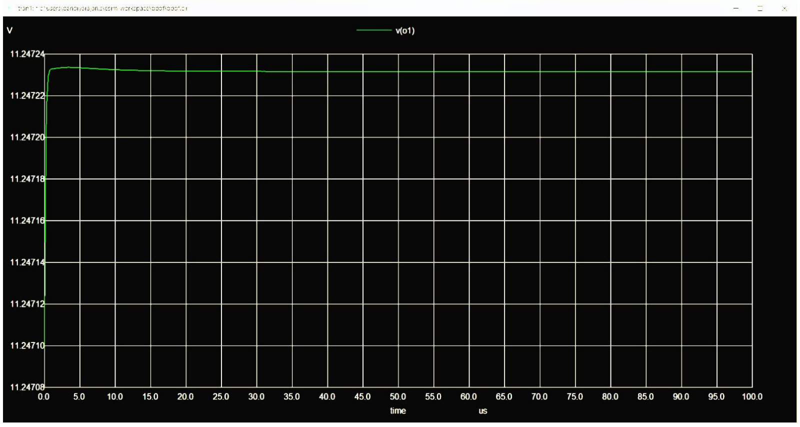

Figure 4.24: Simulation output of TL317

4.11 MC33171

4.11.1 Schematic Symbol Creation

The schematic symbol of MC33171 was created in eSim using proper terminal assignments according to the datasheet specifications.

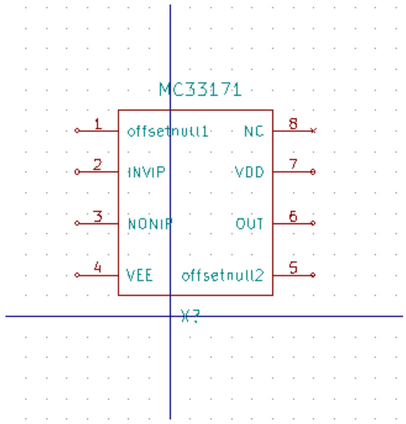

Figure 4.25: Schematic symbol of MC33171

4.11.2 Subcircuit Schematic Design

The SPICE subcircuit schematic of MC33171 was implemented and integrated into the eSim simulation environment for operational amplifier analysis.

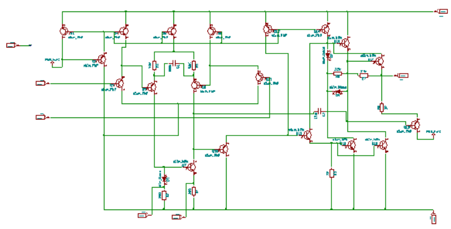

Figure 4.26: Subcircuit schematic of MC33171

4.11.3 Test Bench Circuit Design

A test bench circuit was designed to validate the output characteristics of the MC33171 IC under different operating conditions.

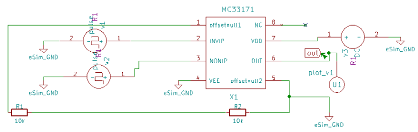

Figure 4.27: Test bench circuit of MC33171

4.11.4 Simulation Results

The implemented MC33171 subcircuit was simulated using Ngspice integrated with eSim. The obtained output verified the expected operational amplifier characteristics according to the datasheet specifications.

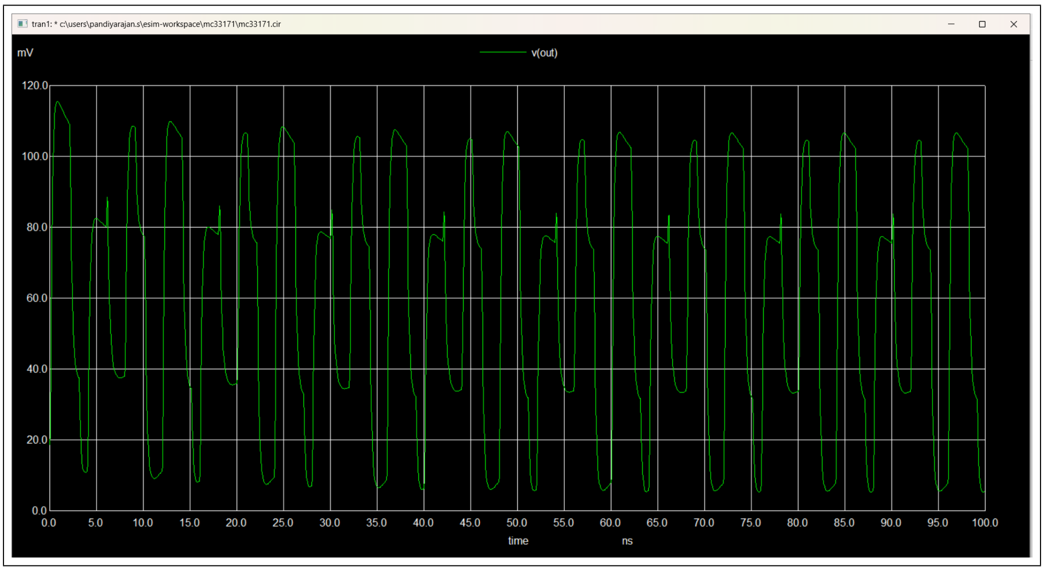

Figure 4.28: Simulation output of MC33171

4.12 MIC4422

4.12.1 Schematic Symbol Creation

The schematic symbol of MIC4422 was created in eSim using proper terminal assignments according to the datasheet specifications.

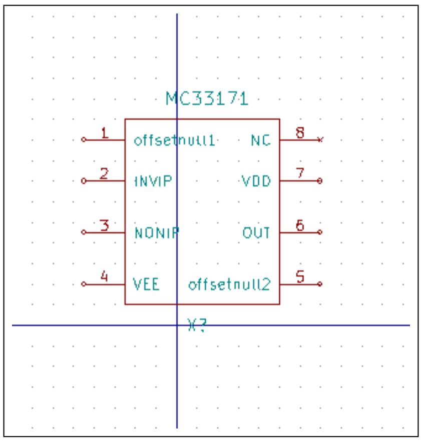

Figure 4.29: Schematic symbol of MIC4422

4.12.2 Subcircuit Schematic Design

The SPICE subcircuit schematic of MIC4422 was implemented and integrated into the eSim simulation environment for MOSFET gate driver analysis.

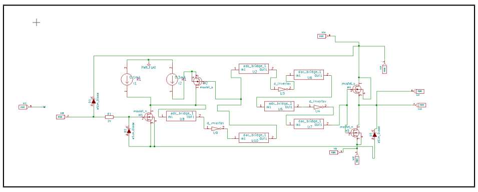

Figure 4.30: Subcircuit schematic of MIC4422

4.12.3 Test Bench Circuit Design

A test bench circuit was designed to validate the output driving characteristics of the MIC4422 IC under different operating conditions.

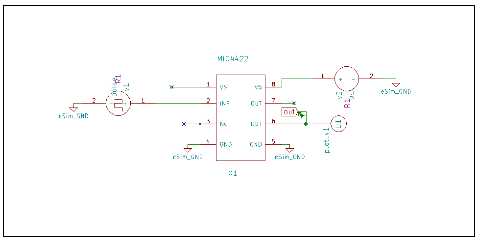

Figure 4.31: Test bench circuit of MIC4422

4.12.4 Simulation Results

The implemented MIC4422 subcircuit was simulated using Ngspice integrated with eSim. The obtained output verified the expected MOSFET gate driving characteristics according to the datasheet specifications.

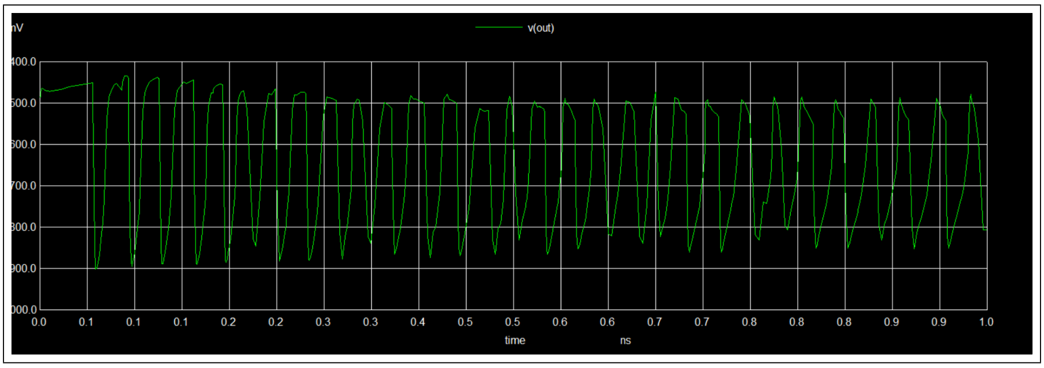

Figure 4.32: Simulation output of MIC4422

4.13 SN75451

4.13.1 Schematic Symbol Creation

The schematic symbol of SN75451 was created in eSim according to the datasheet specifications with proper input, output, and power pin assignments.

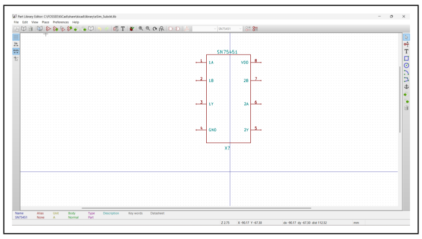

Figure 4.33: Schematic symbol of SN75451

4.13.2 Subcircuit Schematic Design

The internal SPICE subcircuit schematic of SN75451 was implemented and integrated into the eSim simulation environment for functional verification.

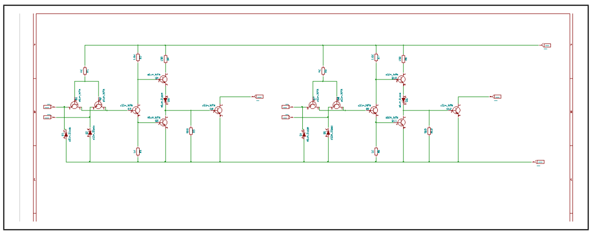

Figure 4.34: Subcircuit schematic of SN75451

4.13.3 Test Bench Circuit Design

A test bench circuit was designed to verify the logic operation and switching behavior of the implemented SN75451 subcircuit model.

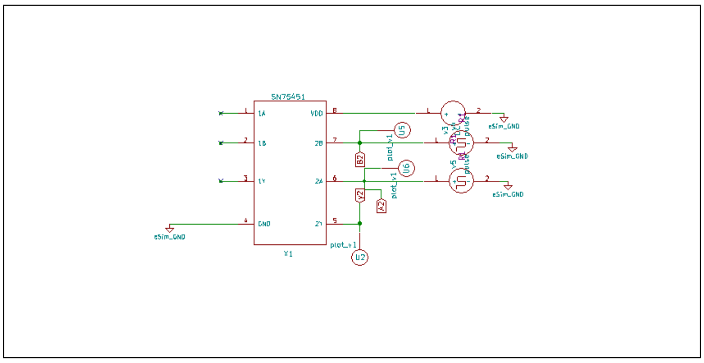

Figure 4.35: Test bench circuit of SN75451

4.13.4 Simulation Results

The implemented SN75451 subcircuit was simulated using Ngspice integrated with eSim. The obtained simulation waveforms verified the correct operational behavior of the IC.

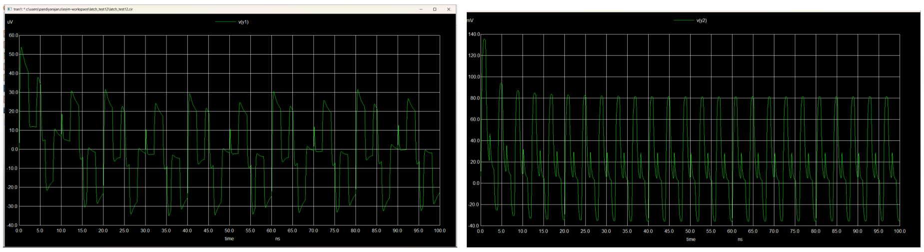

Figure 4.36: Simulation output of SN75451

4.14 MC10136

4.14.1 Schematic Symbol Creation

The schematic symbol of MC10136 was created in eSim according to the datasheet specifications with proper input, output, and power pin assignments.

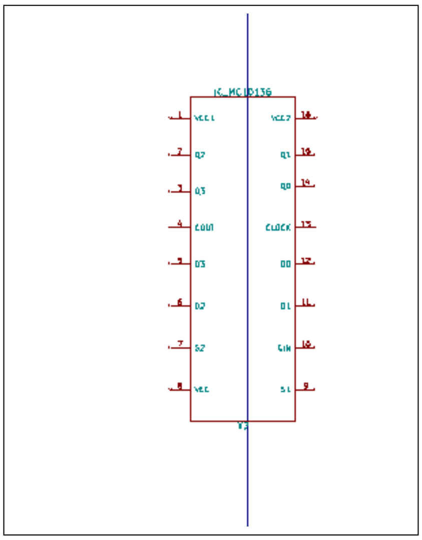

Figure 4.37: Schematic symbol of MC10136

4.14.2 Subcircuit Schematic Design

The internal SPICE subcircuit schematic of MC10136 was implemented and integrated into the eSim simulation environment for functional verification.

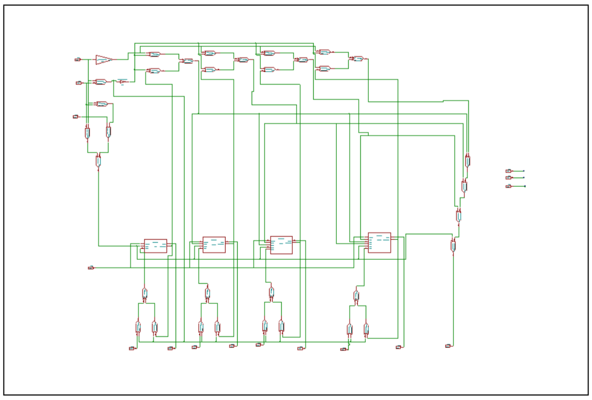

Figure 4.38: Subcircuit schematic of MC10136

4.14.3 Test Bench Circuit Design

A test bench circuit was designed to verify the functional operation and output characteristics of the implemented MC10136 subcircuit model.

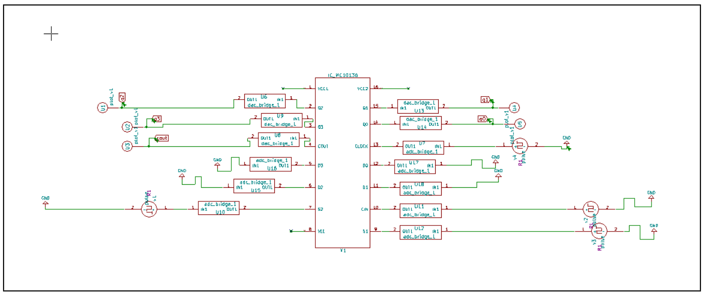

Figure 4.39: Test bench circuit of MC10136

4.14.4 Simulation Results

The implemented MC10136 subcircuit was simulated using Ngspice integrated with eSim. The obtained simulation waveforms verified the correct operational behavior of the IC according to the datasheet specifications.

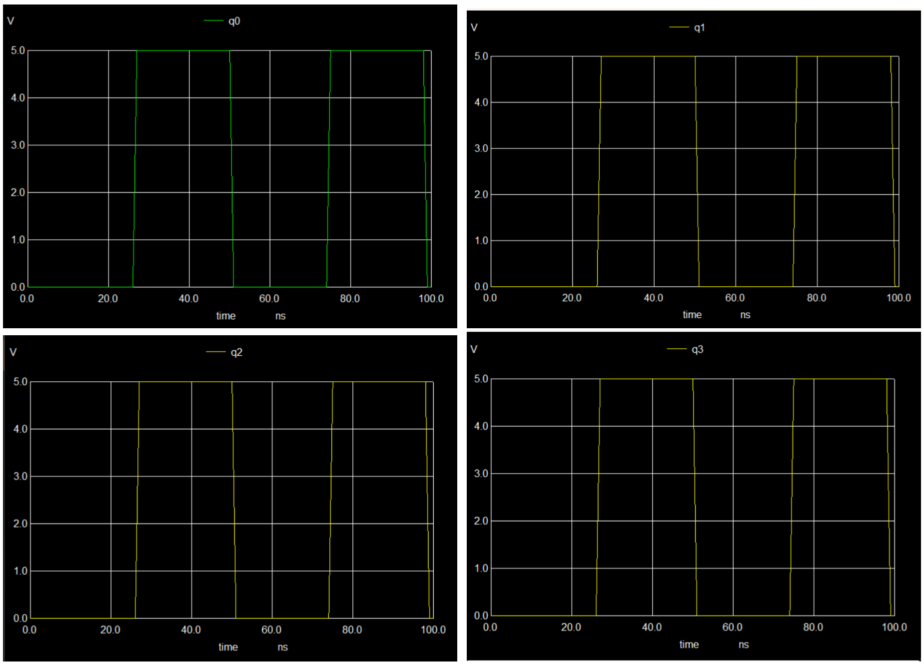

Figure 4.40: Simulation output of MC10136

Chapter 5

Conclusion

During the internship, various electronic devices and integrated circuits were studied, modeled, and simulated using the eSim Electronic Design Automation (EDA) tool. The workflow involved datasheet analysis, schematic symbol creation, subcircuit development, test bench design, and functional verification through Ngspice simulations.

A total of multiple devices, including TLC2264, LM386N, LM3411, MC1556, SDA12, TL317, MC33171, MIC4422, SN75451, and MC10136, were successfully implemented and tested. The simulation results obtained from eSim were consistent with the expected behavior described in the respective datasheets, validating the correctness of the developed models.

This internship provided practical exposure to SPICE-based circuit modeling, analog and digital circuit simulation, datasheet interpretation, and open-source EDA tools such as eSim, Ngspice, and KiCad. The experience enhanced both theoretical understanding and hands-on skills in electronic circuit design and verification. The developed device models can further contribute to the eSim library and support future educational and research activities in electronic system design.

Bibliography

[1] FOSSEE Project, IIT Bombay, *eSim - Open Source EDA Tool for Circuit Design, Simulation and PCB Design.*
https://esim.fossee.in

[2] The ngspice Team, *Ngspice User's Manual and Simulation Documentation.*
https://ngspice.sourceforge.io

[3] KiCad EDA Project, *KiCad Electronic Design Automation Suite.*
https://www.kicad.org

[4] FOSSEE Project, IIT Bombay, *Free/Libre and Open Source Software for Education.*
https://fossee.in

[5] L. W. Nagel and D. O. Pederson, *SPICE (Simulation Program with Integrated Circuit Emphasis).*
University of California, Berkeley.

[6] Texas Instruments, *TLC2264 Quad Operational Amplifier Datasheet.*
https://www.alldatasheet.com/datasheet-pdf/view/82991/TI/TLC2264.html

[7] Texas Instruments, *LM386 Low Voltage Audio Power Amplifier Datasheet.*
https://www.ti.com/lit/ds/symlink/lm386.pdf

[8] Texas Instruments, *LM3411 LED Driver Datasheet.*
https://www.ti.com

[9] Motorola Semiconductor, *MC1556 Timer Datasheet.*
https://www.alldatasheet.com

[10] SDA12 Device Datasheet.
https://www.alldatasheet.com

[11] Texas Instruments, *TL317 Adjustable Voltage Regulator Datasheet.*
https://www.ti.com/lit/ds/symlink/lm317.pdf

[12] ON Semiconductor, *MC33171 Operational Amplifier Datasheet.*
https://www.onsemi.com

[13] Microchip Technology, *MIC4422 MOSFET Driver Datasheet*.
https://ww1.microchip.com/downloads/en/DeviceDoc/mic4421-22.pdf

[14] Texas Instruments, *SN75451 Dual Peripheral Driver Datasheet*.
https://www.ti.com/lit/ds/symlink/sn75451b.pdf

[15] Motorola Semiconductor, *MC10136 MECL Logic Device Datasheet*.
https://www.alldatasheet.com/datasheet-pdf/pdf/8990/MOTOROLA/MC10136.html