# Winter Internship Report

On

# Integrated Circuit Design using Subcircuit Feature of eSim

Submitted by

**Manimaran K**

Under the guidance of

**Prof. Kannan M. Moudgalya**

Chemical Engineering Department

IIT Bombay

February 11, 2025

# Acknowledgment

We sincerely appreciate the **FOSSEE, IIT Bombay** team for giving us this wonderful opportunity to work on the design and integration of multiple sub-circuits in eSim. This experience has been incredibly enriching, allowing us to explore various open-source EDA tools for circuit simulation and understand their practical applications.

Our heartfelt gratitude goes to **Prof. Kannan M. Moudgalya** for his invaluable guidance and encouragement throughout this fellowship program. We are also immensely thankful to our mentors, **Mr. Sumanto Kar, Ms. Usha Vishwanathan**, and **Ms. Vineeta Ghavri**, for their constant support, mentorship, and expertise. Their timely assistance and insights helped us navigate challenges and successfully complete our project.

This internship has been a significant learning experience, equipping us with skills and knowledge that will be valuable in our future endeavors. Contributing to **FOSSEE** has been a rewarding journey, and we truly cherish the knowledge and practical experience gained. As aspiring professionals in the semiconductor industry, we see this internship as an important milestone in shaping our careers.

# Contents

Chapter 1

# Introduction

## 1.1 FOSSEE

FOSSEE, which stands for Free/Libre and Open Source Software for Education, is an organization based at IIT Bombay. It is a remarkable initiative aimed at promoting the use of open-source software in education and research. It was established with the mission to reduce 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 to empower 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

## 1.2 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, and 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.

## 1.3 NgSpice

NgSpice is an open-source SPICE simulator for electric and electronic circuits. It can simulate various circuit elements, including JFETs, bipolar and MOS transistors, passive elements (R, L, C), diodes and other devices, all interconnected in a netlist.

Digital circuits are also simulated, ranging from single gates to complex circuits, including combinations of analog, digital, and mixed-signal circuits. NgSpice offers a wealth of device models for active, passive, analog, and digital elements. Users input their circuits as netlists, and the output is one or more graphs of currents, voltages, and other electrical quantities, or saved in a data file

## 1.4 Makerchip

Makerchip is a platform that offers convenient and accessible 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 and proprietary tools, ensuring a comprehensive range of capabilities.

Users 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 userfriendly 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. While some are capable of PCB design (e.g., KiCad), others focus on simulations (e.g., gEDA). To the best of our knowledge, there is no open-source software that combines circuit design, simulation, and layout design in one platform. eSim addresses this gap by integrating all these capabilities.

**Chapter 2**

# Features of eSim

The objective behind the development of eSim is to provide an open-source EDA solution for electronics and electrical engineers. The software is capable of performing schematic creation, PCB design, and circuit simulation (analog, digital, and mixedsignal). It also provides 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 (Design Rule Check) 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 for future circuit design purposes by developers and users once they are successfully integrated into the eSim sub-circuit library.

## 3.1 Approach

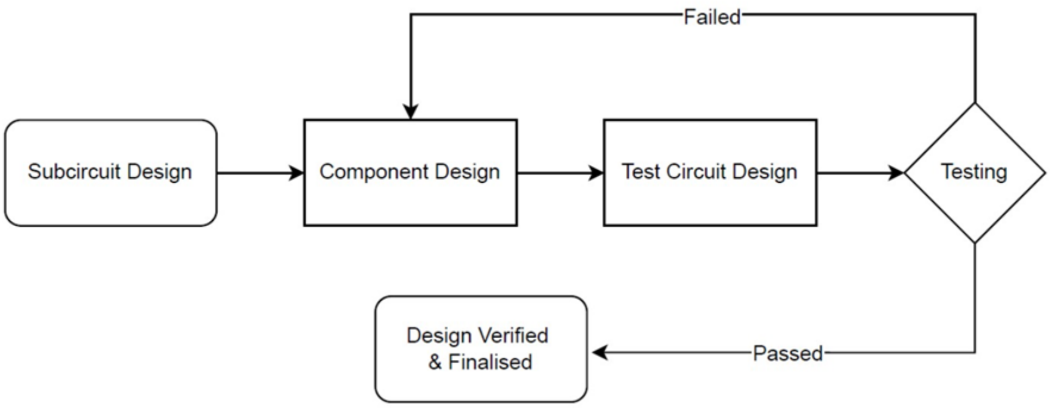


Figure 3.1: Flowchart of IC Design Approach Followed

Our approach to implementing the problem statement involved a systematic process, leveraging datasheets from leading Integrated Circuit (IC) manufacturers such as Texas Instruments, Analog Devices, and NXP Semiconductors. We focused on selecting ICs with diverse functionalities, including precision amplifiers, comparators, encoders, and audio amplifiers. The process is outlined in the following steps:

**1. Analyzing Datasheets:** The first step involved an in-depth review of datasheets for various analog and digital ICs. We aimed to identify circuits suitable for implementation in eSim that were not already present in the eSim library. This process included scrutinizing the detailed schematics of each IC, evaluating component values, and understanding truth tables. The goal was to select ICs that offered unique functionalities or enhancements not yet covered.

**2. Subcircuit Creation:** After selecting appropriate ICs, we proceeded to model these as sub-circuits within eSim. We utilized the model files available in the eSim device model library and ensured that our designs adhered strictly to the specifications outlined in the official datasheets. This phase also involved creating accurate symbol and pin diagrams for each IC, in accordance with the packaging and pin descriptions provided in the datasheets. This step was crucial for ensuring the fidelity of the subcircuit models.

**3. Test Circuit Design:** With the sub-circuits created, we then designed and built test circuits based on the datasheets. This step was essential for verifying the functionality of each sub-circuit. We developed a series of test cases and constructed corresponding test circuits to evaluate the performance and accuracy of the implemented IC models.

**4. Schematic Testing:** Following the construction of test circuits, we conducted simulations to analyze the outputs. This involved generating waveforms and plots to assess the behavior of the circuits. We employed KiCad for converting designs to NgSpice netlists and utilized eSim's simulation features to perform comprehensive testing.

If the simulated outputs deviated from expected results, it signaled potential errors in the schematic. In such instances, we revisited the design phase to identify and correct discrepancies. The iterative process of debugging and re-testing continued until the test cases produced satisfactory results. Once the IC models met the desired performance criteria, they were deemed successful, marking the completion of the design process

Chapter 4

# DAC0800

## 4.1 General Description

The DAC0800 is a high-speed, high-accuracy 8-bit digital-to-analog converter (DAC) known for its exceptional performance and stability. This DAC offers extremely tight regulation across a broad range of operating conditions, coupled with a notably low conversion time and excellent temperature stability.

## 4.2 Key Features

- **Fast Conversion Speed:** Provides a typical settling time of 100 ns, enabling high-speed operations.
- **High Accuracy:** Ensures low non-linearity and precise digital-to-analog conversion.
- **Wide Power Supply Range:**Operates with supply voltages from ±4.5V to ±18V, making it versatile for different applications.
- **Low Power Consumption:**Designed for efficient power usage with a typical current draw of 3mA.

## 4.3 Applications

- **Audio Signal Processing:** Used in digital audio systems for high-fidelity sound conversion.
- **Waveform Generation:** Generates analog waveforms such as sine, square, and triangular waves.
- **Communication Systems:** Used in modulation and signal processing for data transmission.

## 4.4 Pin Configuration

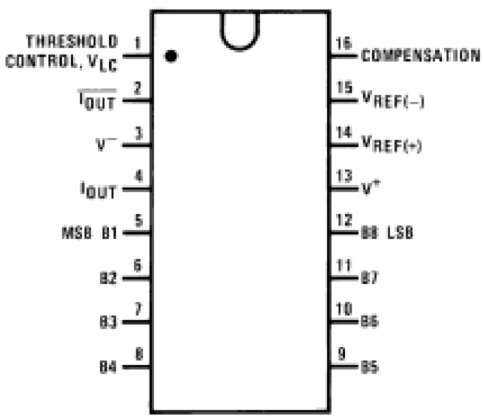


Figure 4.1: Pin Configuration of the DAC0800

### 4.5 IC Layout

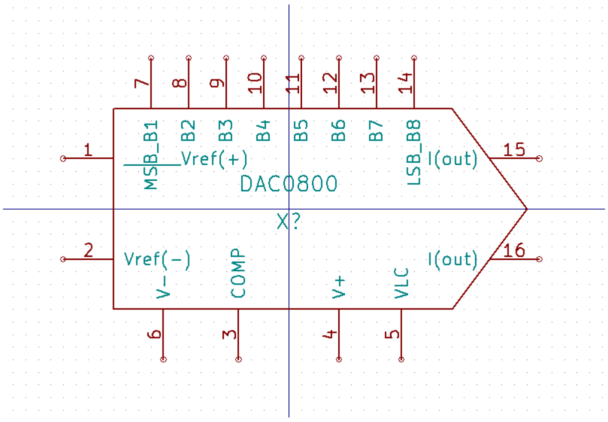


Figure 4.2: IC Layout of DAC0800

### 4.6 Subcircuit Shematic Diagram

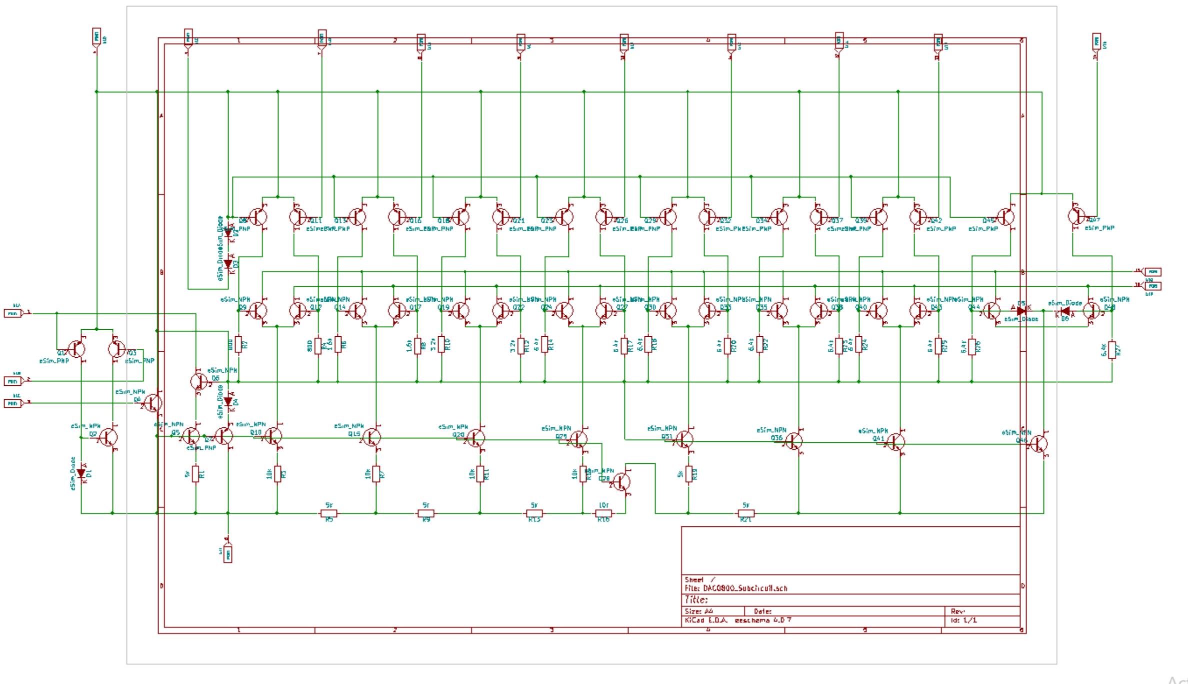

Figure 4.3: Subcircuit Schematic Diagram of the DAC0800

## 4.7 Text Circuit

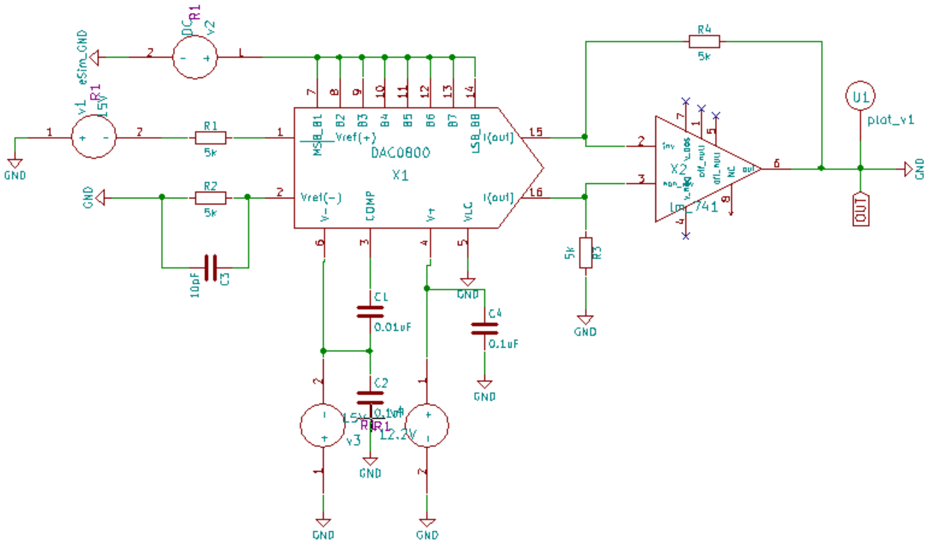


Figure 4.4: Test Circuit of the DAC0800

## 4.8 Output Plot

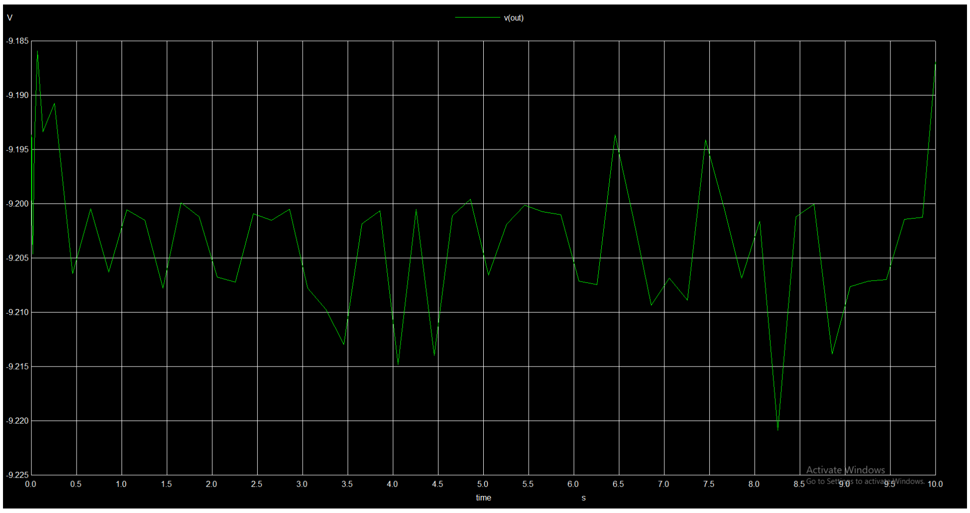


Figure 4.5: Output of the DAC0800

Chapter 5

# SN7408

## 5.1 General Description

The SN7408 is a quad 2-input AND gate known for its exceptional performance and stability. This logic gate offers extremely tight regulation across a broad range of operating conditions, coupled with a notably low power dissipation and excellent temperature stability.

## 5.2 Key Features

- **TTL Compatible:** Designed to interface easily with Transistor-Transistor Logic (TTL) circuits.
- **Wide Operating Voltage Range:** Operates typically at 5V, making it compatible with standard logic circuits.
- **Temperature Stability:** Functions reliably across a wide temperature range, suitable for industrial and commercial applications.

## 5.3 Applications

- **Arithmetic and Logic Units (ALUs):** Plays a key role in performing logical operations in processors.
- **Signal Processing:** Used in filtering and signal validation in digital electronics.
- **Embedded Systems:** Integrated into microcontroller-based applications for logical operations.

## 5.4 Pin Configuration

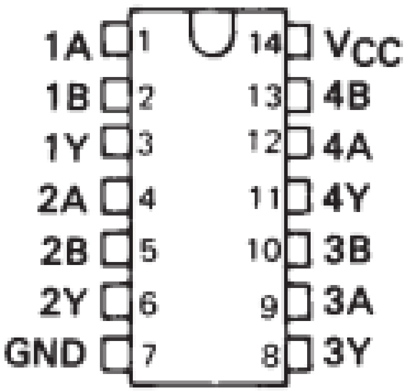


Figure 5.1: Pin Configuration of SN7408

## 5.5 IC Layout

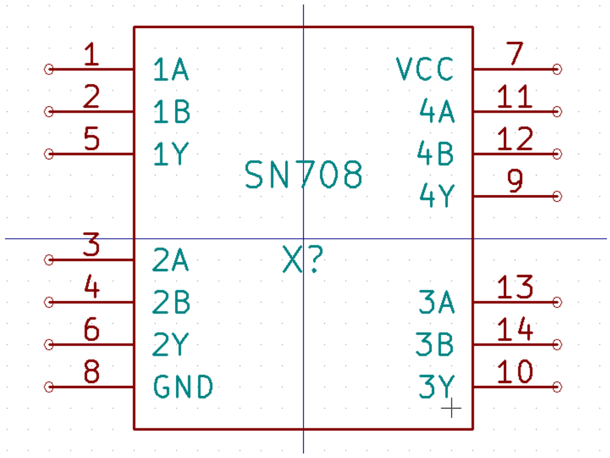


Figure 5.2: IC Layout of the SN7408

## 5.6 Subcircuit Shematic Diagram

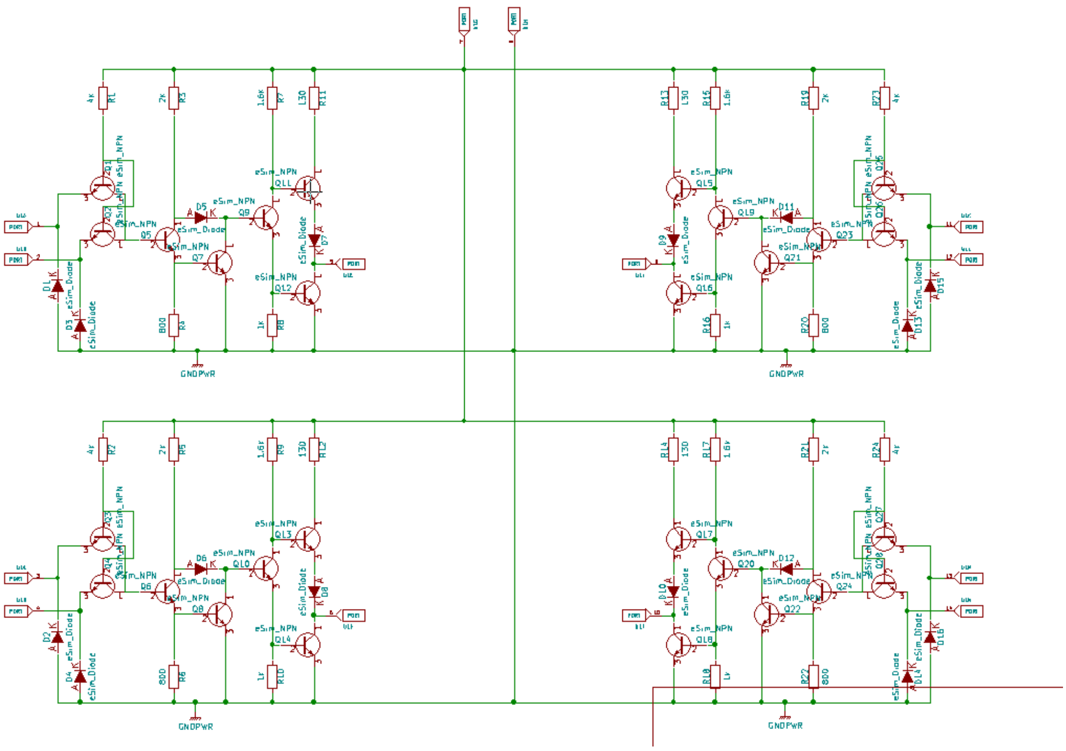

Figure 5.3: Subcircuti Shematic of the SN7408

## 5.7 Text Circuit

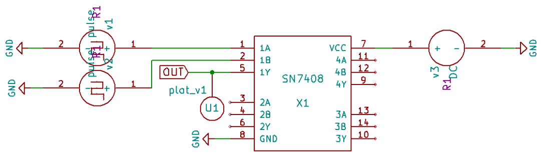


Figure 5.4: Test Circuit of the SN7408

## 5.8 Output Plot HIGH

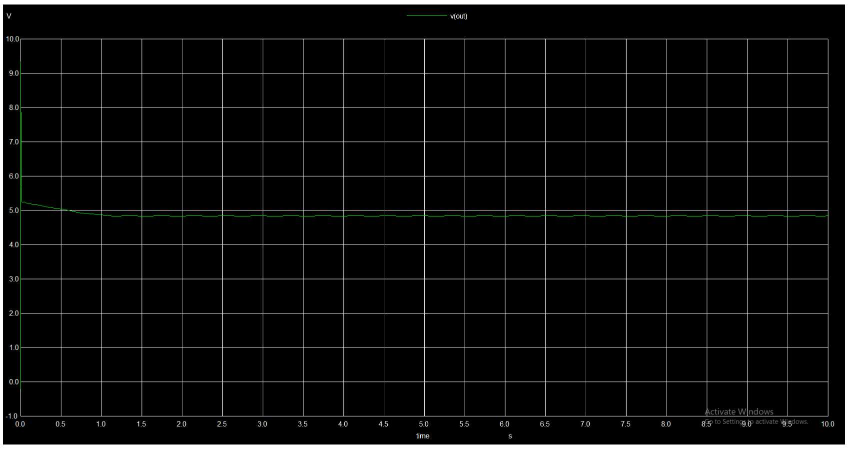


Figure 5.5: HIGH Output of the SN7408

## 5.9 Output Plot LOW

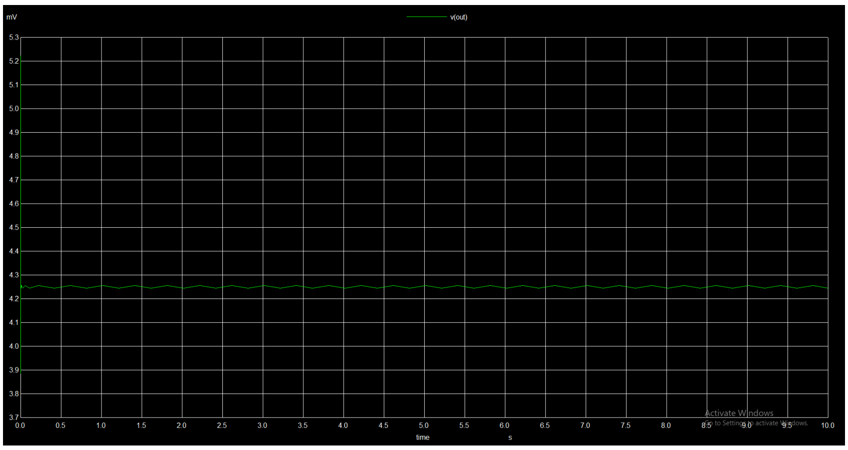


Figure 5.6: LOW Output of the SN7408

**Chapter 6**

# SN7432

## 6.1 General Description

The SN7432 is a quad 2-input OR gate IC from the 74-series TTL logic family. It contains four independent OR gates, providing high-speed logic operations with low power consumption. It is widely used in digital circuits, computing, and control systems for reliable OR logic processing.

## 6.2 Key Features

- **TTL Compatible:** Works with standard Transistor-Transistor Logic (TTL) circuits.
- **Low Power Consumption:** Optimized for efficient digital system performance.
- **Wide Operating Voltage Range:** Typically operates at 5V for standard logic applications.

## 6.3 Applications

- **Digital Logic Circuits:** Used for implementing OR logic in combinational circuits.
- **Arithmetic and Logic Units (ALUs):** Plays a role in performing logical operations in processors.
- **Signal Processing:** Used for filtering and validating digital signals.

## 6.4 Pin Configuration

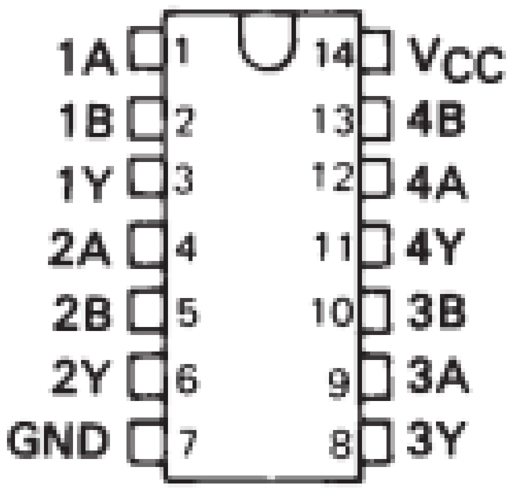


Figure 6.1: Pin Configuration of the SN7432

### 6.5 IC Layout

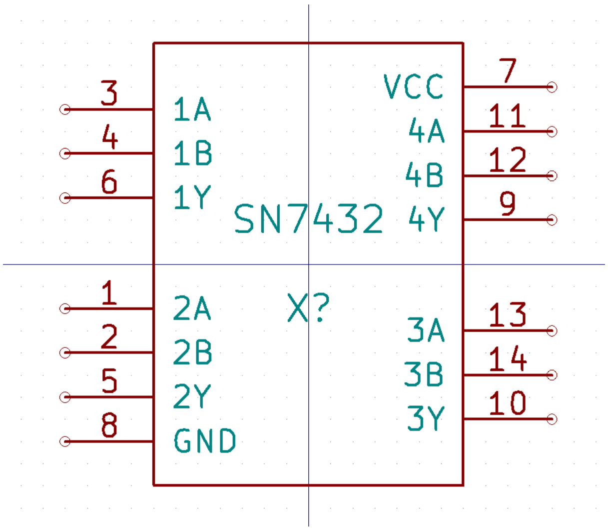


Figure 6.2: IC Layout of the SN7432

### 6.6 Subcircuit Shematic

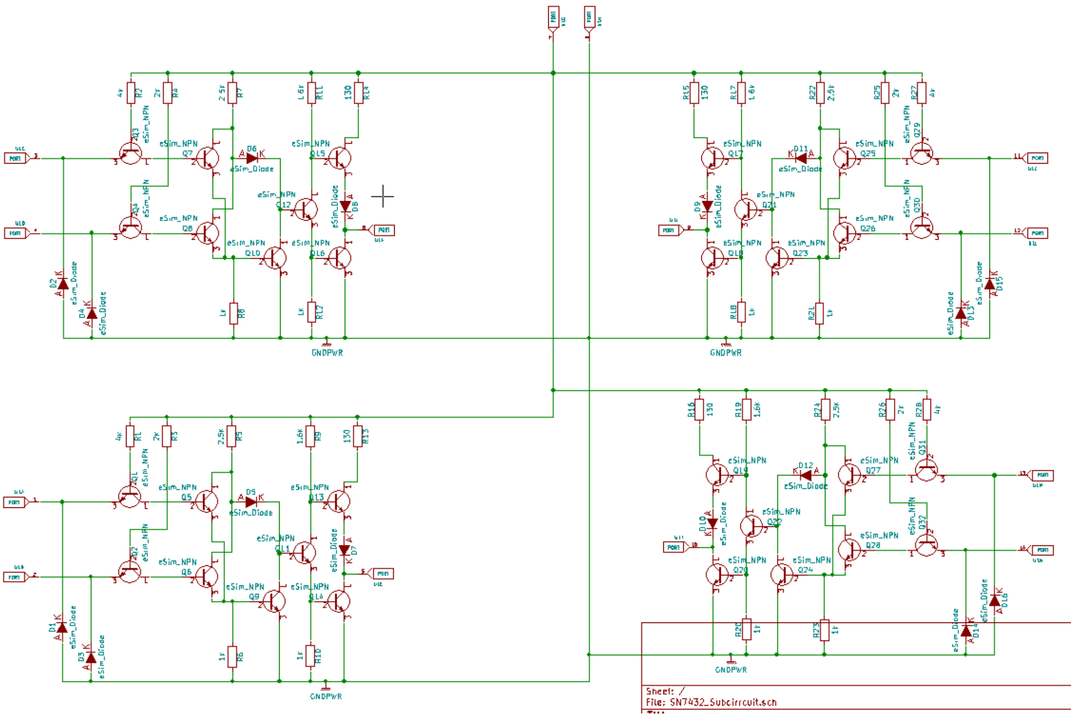

Figure 6.3: Subcircuit Chematic of the SN7432

## 6.7 Test Circuit

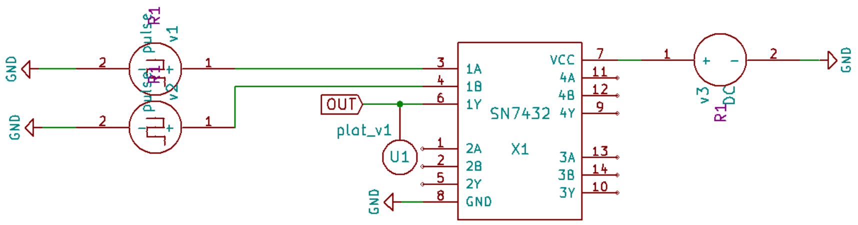


Figure 6.4: Test Circuit of the SN7432

## 6.8 Output Plot HIGH

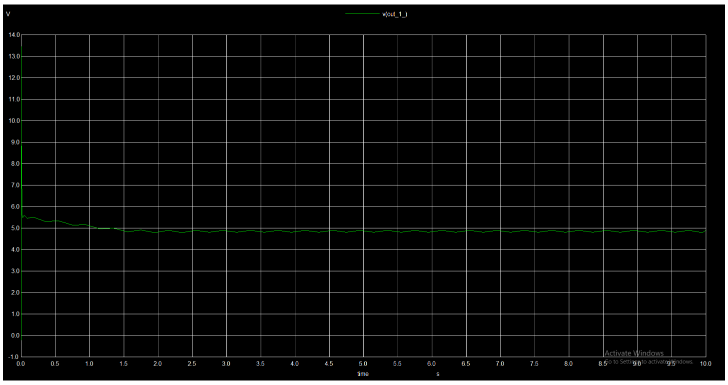

Figure 6.5: HIGH Output Plot of the SN7432

## 6.9 Output Plot LOW

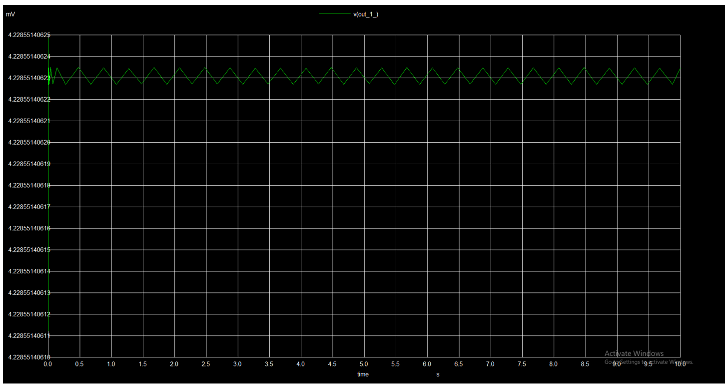

Figure 6.6: LOW Output Plot of the SN7432

Chapter 7

# SN74LS11

## 7.1 General Description

The SN74LS11 is a high-speed triple 3-input AND gate known for its exceptional performance and stability. This logic gate offers extremely tight regulation across a broad range of operating conditions, coupled with a notably low power dissipation and excellent temperature stability.

## 7.2 Key Features

- **TTL Compatible:** Works seamlessly with Transistor-Transistor Logic (TTL) circuits.
- **High-Speed Operation:** Designed for fast logic processing with minimal delay.
- **Schottky-Clamped for Low Propagation Delay:** Provides improved speed compared to standard TTL logic gates.

## 7.3 Applications

- **Digital Logic Circuits:** Used in combinational logic circuits for decision-making operations.
- **Arithmetic and Logic Units (ALUs):** Plays a key role in performing logical operations in processors.
- **Data Processing:** Used in memory address decoding and data routing circuits.

## 7.4 Pin Configuration

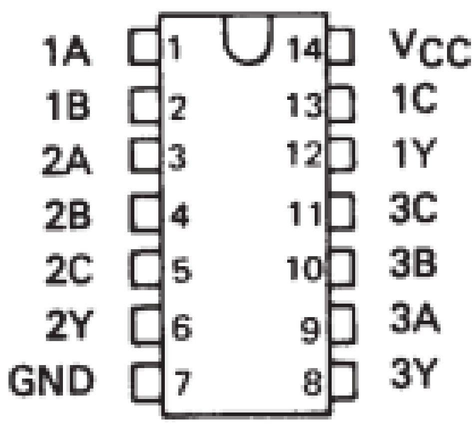


Figure 7.1: Pin Configuration of the SN74LS11

### 7.5 IC Layout

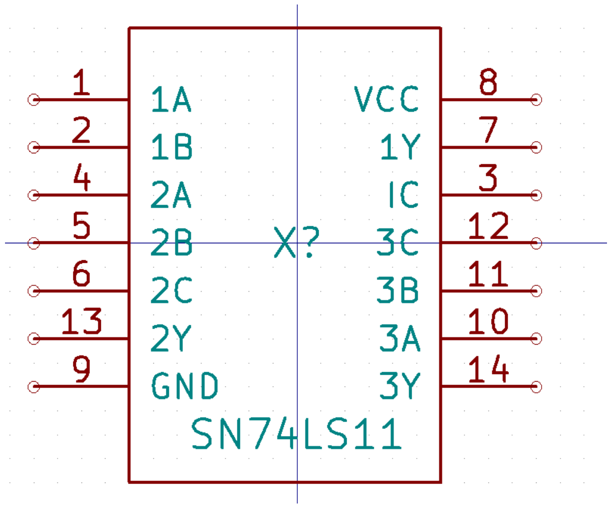


Figure 7.2: IC Layout of the SN74LS11

### 7.6 Subcircuit Shematic

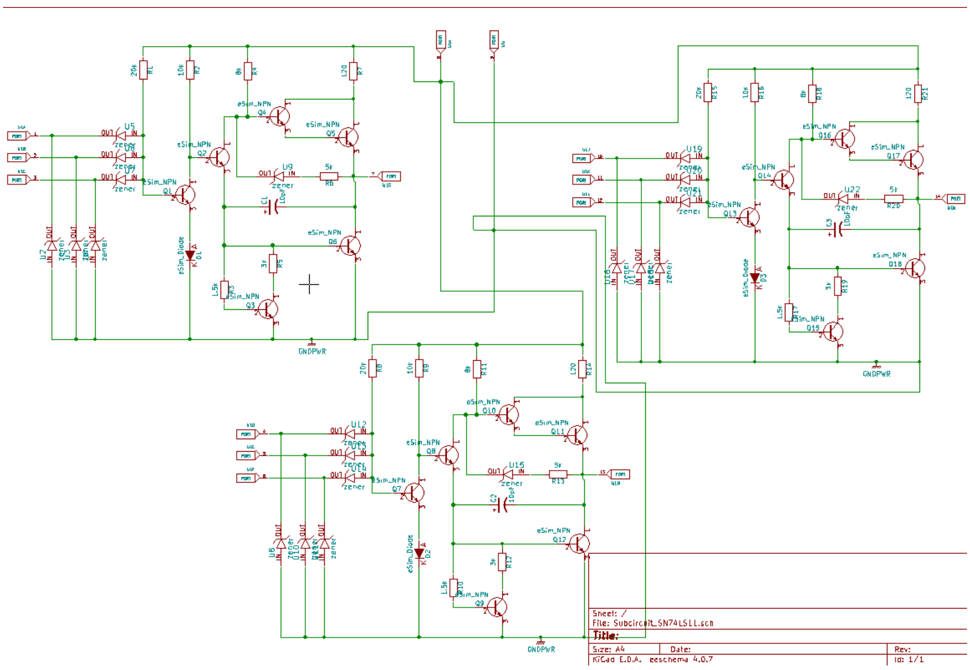

Figure 7.3: Subcircuit Shematic of the SN74LS11

## 7.7 Test Circuit

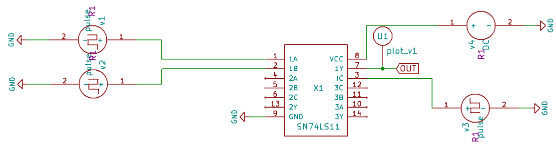


Figure 7.4: Test Circuit of the SN74LS11

## 7.8 Output Plot HIGH

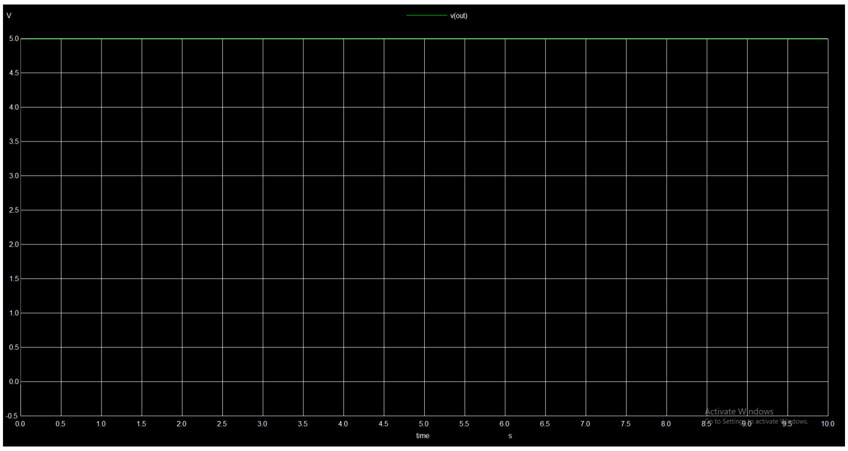


Figure 7.5: HIGH Output Plot of the SN74LS11

## 7.9 Output Plot LOW

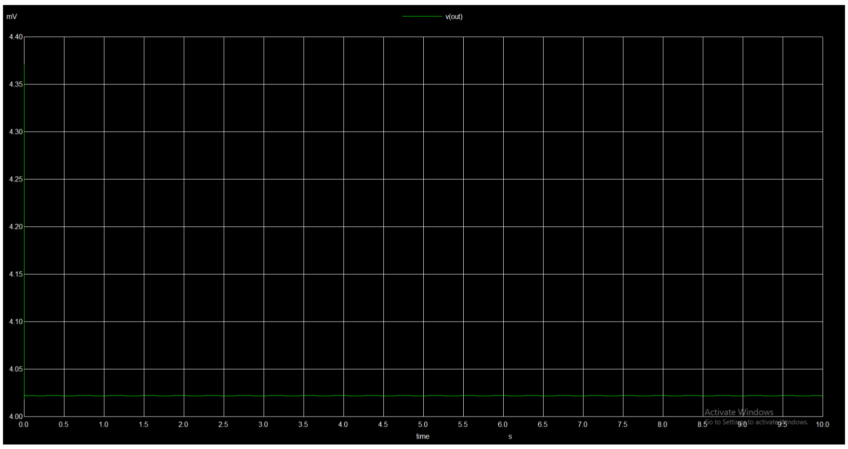


Figure 7.6: LOW Output Plot of the SN74LS11

Chapter 8

# SN7404

## 8.1 General Description

The SN7404 is a hex inverter IC from the 74-series TTL logic family, containing six independent NOT gates. It performs logical inversion, converting HIGH inputs to LOW and vice versa. The SN7404 is widely used in digital logic circuits, signal conditioning, and clock generation for reliable and fast logic operations.

## 8.2 Key Features

- **Hex Inverter Gates:** Contains six independent NOT gates.
- **TTL Compatible**: Works with standard Transistor-Transistor Logic (TTL) circuits.
- **High Noise Immunity:** Ensures stable operation in noisy environments.

## 8.3 Applications

- **Data Communication:** Used in signal buffering and transmission
- **Arithmetic and Logic Units (ALUs):** Plays a key role in performing logical operations in processors.
- **Data Processing:** Used in memory address decoding and data routing circuits.

## 8.4 Pin Configuration

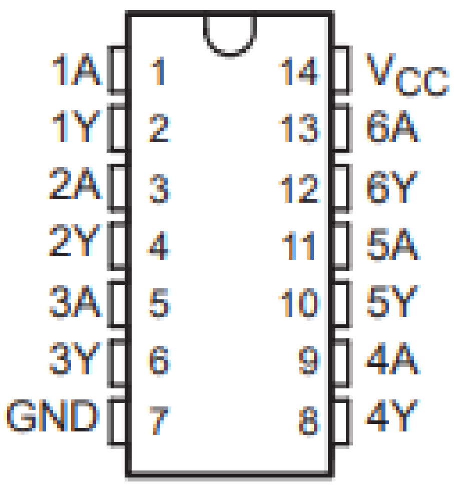


Figure 8.1: Pin Configuration of the SN7404

### 8.5 IC Layout

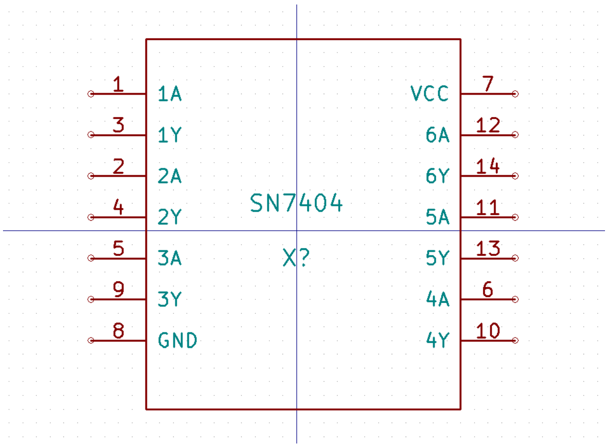


Figure 8.2: IC Layout of the SN7404

### 8.6 Subcircuit Shematic

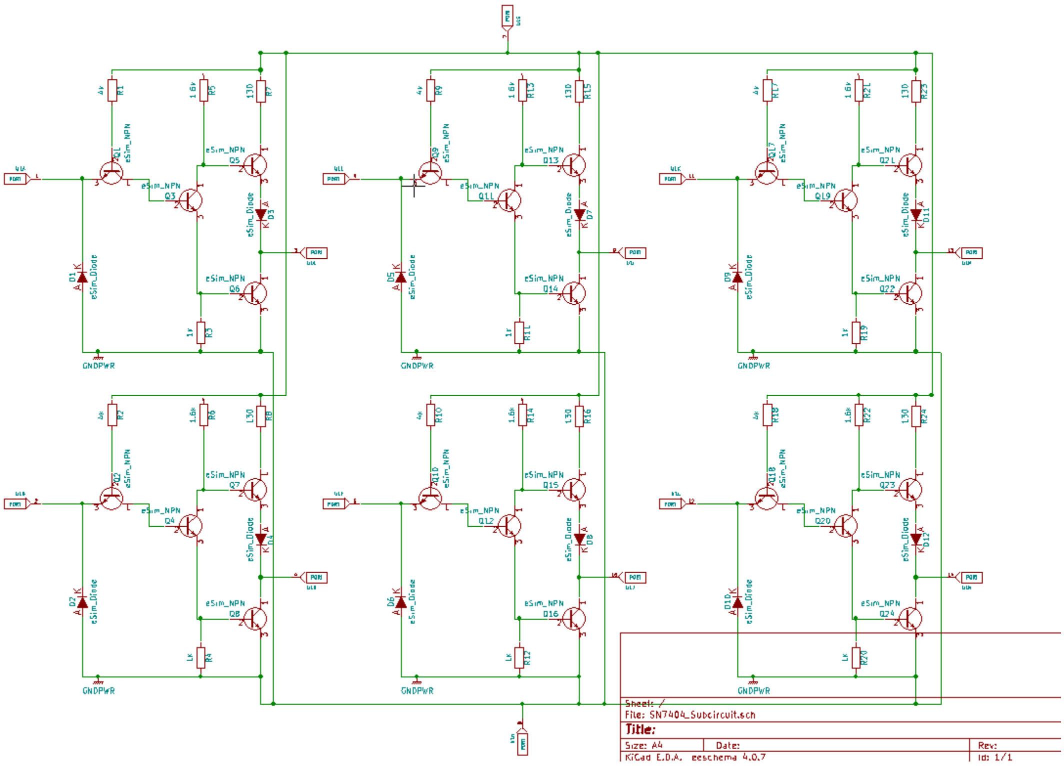


Figure 8.3: Subcircuti Shematic of the SN7404

## 8.7 Test Circuit

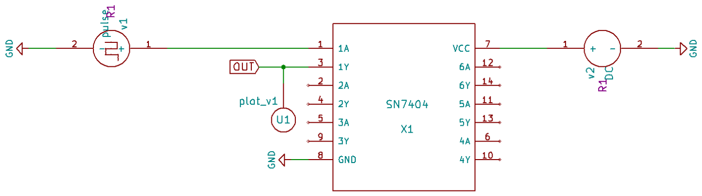


Figure 8.4: Test Circuit of the SN7404

## 8.8 Output Plot HIGH

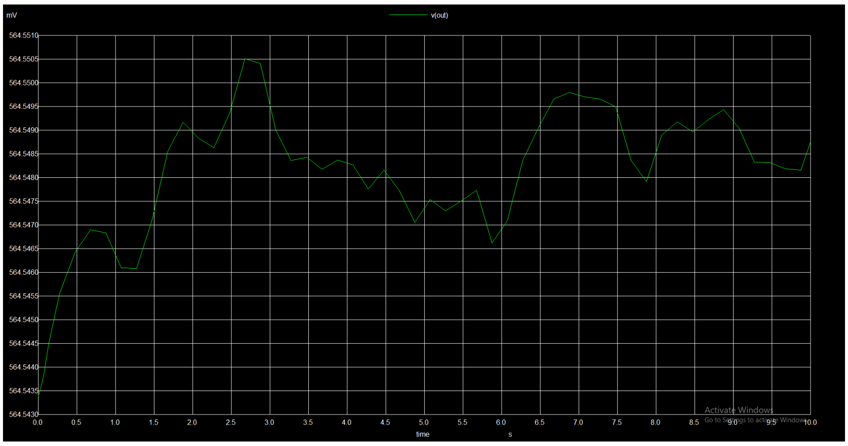


Figure 8.5: HIGH Output Plot of the SN7404

## 8.9 Output Plot LOW

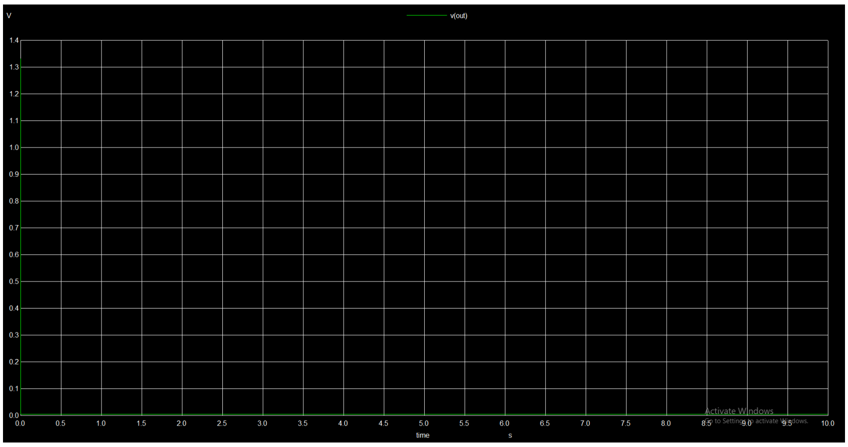


Figure 8.6: LOW Output Plot of the SN7404

Chapter 9

# OP407

## 9.1 General Description

The OP497 is a high-precision, low-noise, quad operational amplifier (op-amp) designed for high-accuracy analog applications. It features low offset voltage, low drift, and low bias current, making it ideal for precision instrumentation and signal processing. The OP497 offers high common-mode rejection and low power consumption, ensuring stable and efficient operation in industrial, medical, and aerospace applications.

## 9.2 Key Features

- **Quad Op-Amp:** Contains four independent precision op-amps.
- **Low Offset Voltage:** Ensures high accuracy with minimal error.
- **High Common-Mode Rejection Ratio (CMRR):** Improves noise immunity.

## 9.3 Applications

- **Precision Instrumentation:** Used in high-accuracy measurement systems.
- **Medical Equipment:** Integrated into ECG, EEG, and other medical devices.
- **Industrial Automation:** Ensures reliable analog signal processing.

## 9.4 Pin Configuration

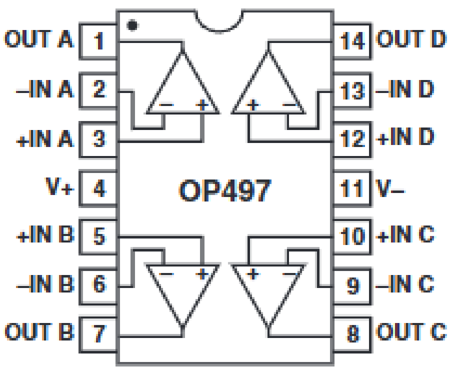


Figure 9.1: Pin Configuration of the OP497

### 9.5 IC Layout

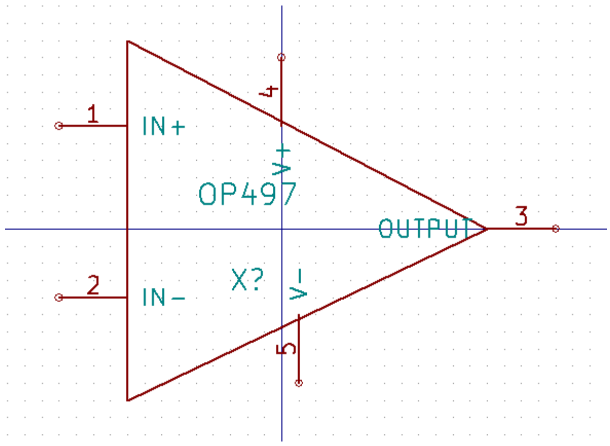


Figure 9.2: IC Layout of the OP497

### 9.6 Subcircuit Shematic

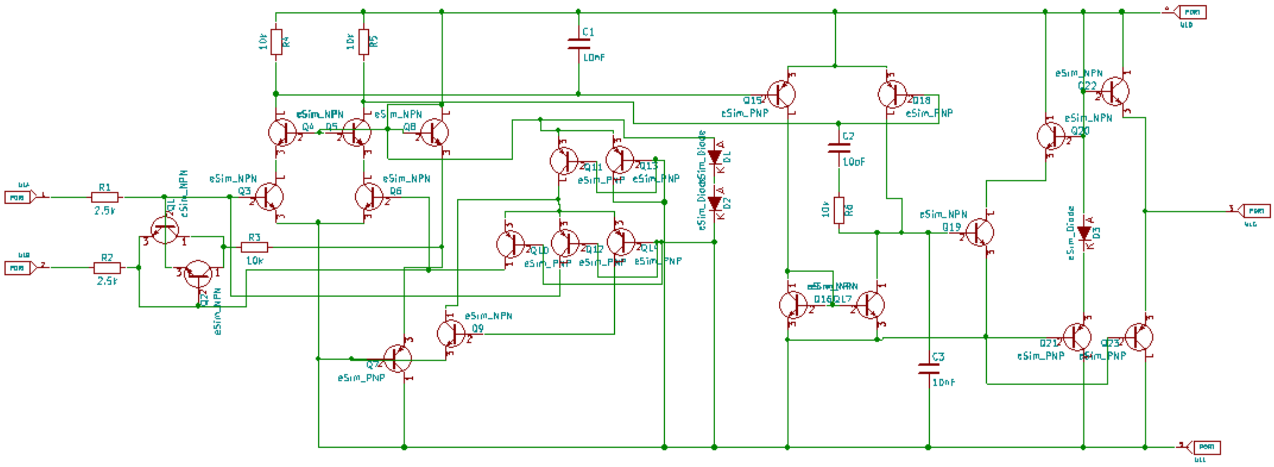

Figure 9.3: Subcircuit Shematic of the OP497

## 9.7 Test Circuit

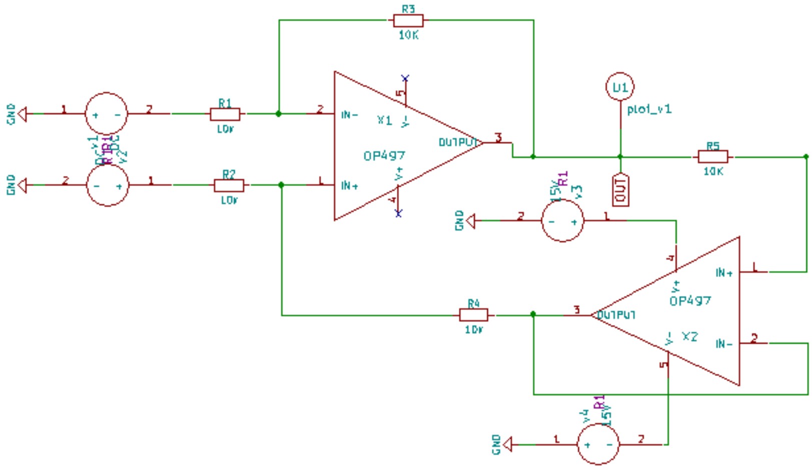

Figure 9.4: Test Circuit of the OP497

## 9.8 Output

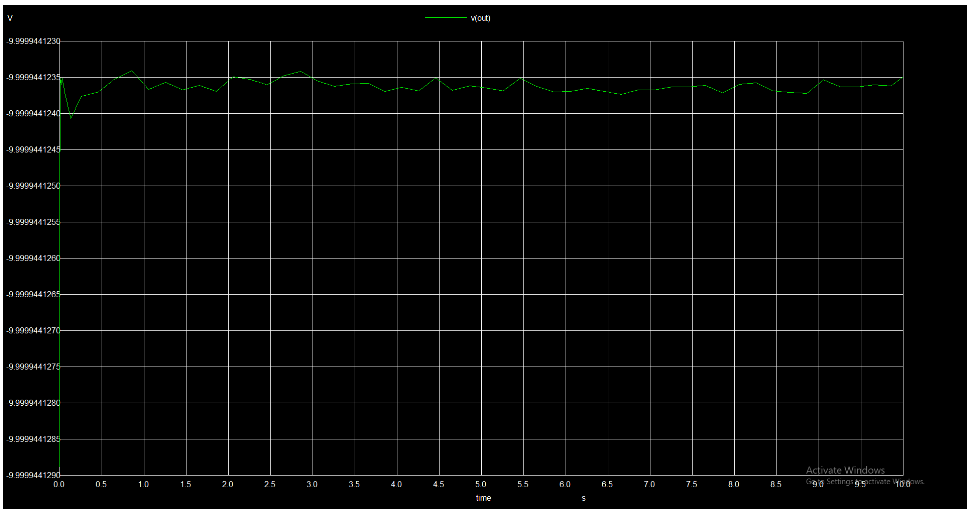

Figure 9.5: Output of the OP497

## Chapter 10

# Failed Circuits

### 10.1 Overview

In this section, we discuss circuits that did not perform as expected during testing.Understanding the reasons for these failures helps in diagnosing issues and improving circuit design. Each failed circuit is analyzed to identify the potential causes of failure and to suggest corrective measures.

### 10.2 NE555

#### Circuit Diagram

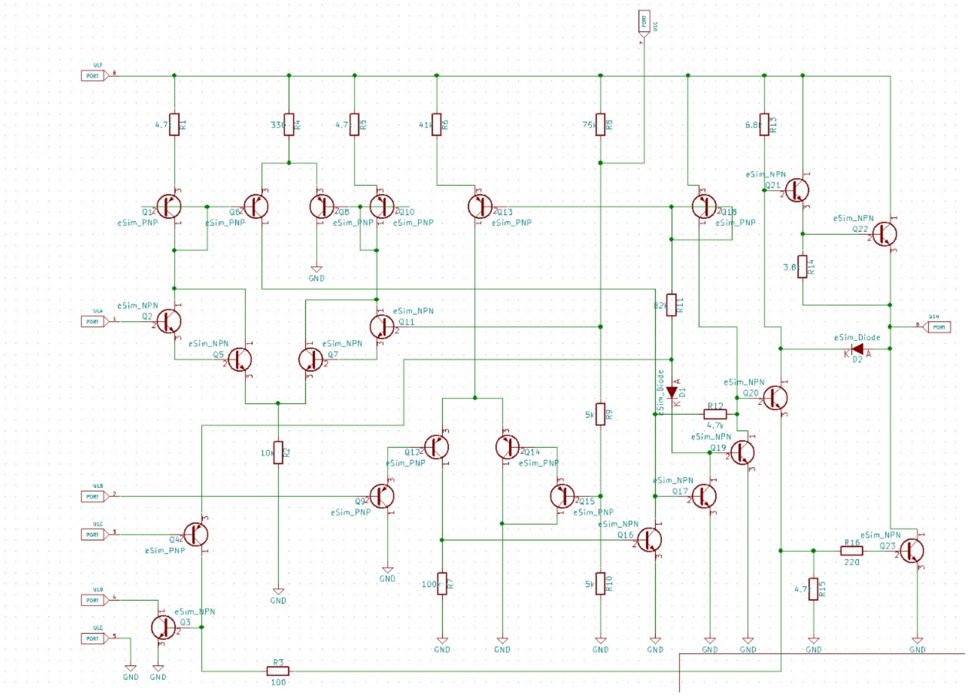

Figure 10.1: Subcircuit Shematic of the NE555

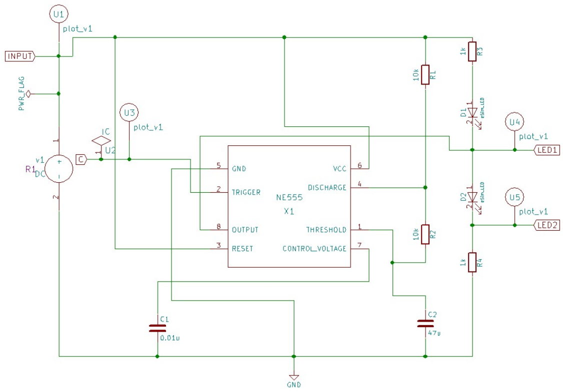


Figure 10.2: Test Circuit of the NE555

### Issue Description

The output waveform of the NE555 timer circuit exhibits an unexpected high-frequency oscillation with a minimal voltage variation in the picovolt (pV) range. This behavior indicates potential instability or incorrect circuit configuration. The issue could stem from incorrect component values, parasitic noise, improper load conditions, or simulation model inaccuracies. Ensuring proper resistor-capacitor selection, minimizing external interference, and verifying circuit connections may help resolve the problem.

## 10.3 AD744

### Circuit Diagram

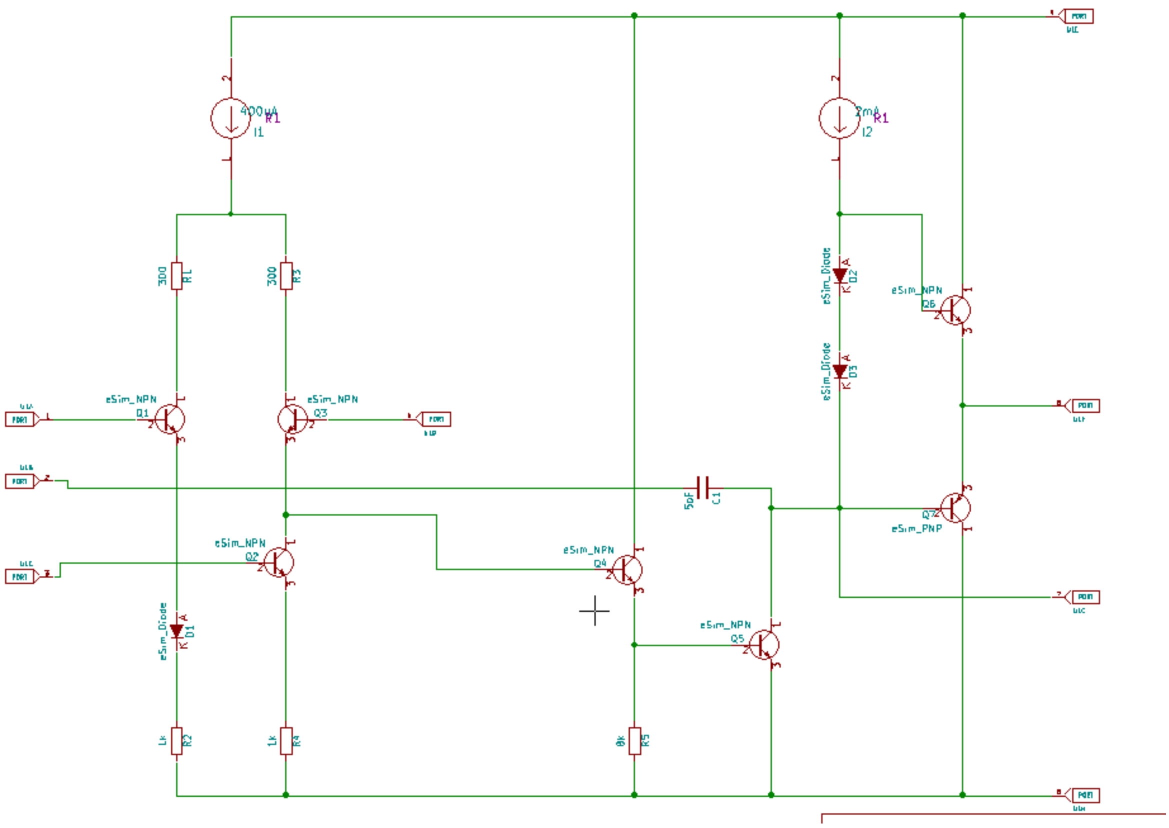


Figure 10.3: Subcircuit Schematic of the AD744

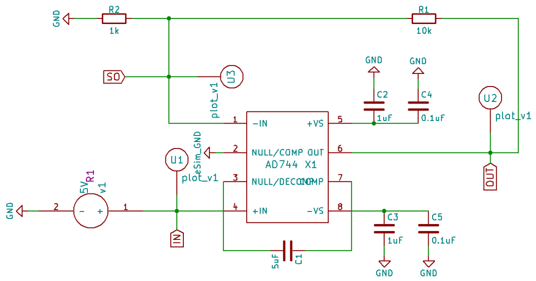


Figure 10.4: Test Circuit of the AD744

### Issue Description

The output waveform of the AD744 operational amplifier (op-amp) exhibits unexpected oscillations in the microvolt (µV) range, indicating potential issues with the circuit configuration. The oscillation may be caused by improper biasing, incorrect feedback network design, inadequate power supply voltage, or parasitic effects within the circuit. Additionally, limitations in the simulation model could contribute to the anomaly.

Chapter 11

# Conclusion and Future Scope

The project achieved its objective of developing a wide range of subcircuits for both Analog and Digital Integrated Circuits, with each IC model meticulously crafted based on the specifications provided in their official datasheets. Through rigorous testing and verification using corresponding test circuits, these IC models were validated for accuracy and functionality. The components developed under this fellowship encompass fundamental circuit elements such as Operational Amplifiers (Op-Amps), Voltage Regulators, Precision Rectifiers, Schmitt Triggers, Differential Amplifiers, Instrumentation Amplifiers, Comparators, Multiplexers, De-Multiplexers, and various Logic Gate ICs. These models are now ready for integration into the eSim subcircuit library, providing a robust resource for developers, students, and researchers. The inclusion of these models in the eSim library will significantly enhance the tool's capabilities, enabling users to easily incorporate these fundamental ICs into their own projects and circuit designs. Looking ahead, this project sets the foundation for the continued expansion of eSim's device model library. We anticipate that more such ready-to-use IC models will be developed, broadening the scope of available components and further empowering the eSim community. This ongoing development will not only aid in academic and research endeavors but also contribute to the growing ecosystem of open-source electronic design automation (EDA) tools.

Chapter 12

# Circuits Contribution

This chapter lists all the Integrated Circuits (ICs) contributed during the fellowship. Each IC has been carefully modeled and tested, and is now part of the eSim library. The contributions include both analog and digital ICs, covering a wide range of functionalities.

### 12.1 Manimaran K List of ICs

1. DAC0800 - A digital-to-analog converter (DAC)
2. SN7408 - A quad 2-input AND gate IC
3. SN7432 - A quad 2-input OR gate IC
4. SN74LS11 - A triple 3-input AND gate IC
5. SN7404 - A hex inverter IC with six NOT gates
6. OP497 - A quad precision operational amplifier
7. NE555 - A widely used timer IC (Failed IC)
8. AD744 - A high-speed, low-noise operational amplifier (Failed IC)

# Bibliography


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

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

[3] Texas Instruments, DAC0800 Datasheet https://www.ti.com/lit/ds/symlink/dac0800.pdf

[4] Texas Instruments, SN7408 Datasheet https://www.ti.com/lit/ds/symlink/sn74ls08.pdf?ts=1739273789599&ref_url=https%253A%252F%252Fwww.mouser.it%252F

[5] Texas Instruments, SN7432 Datasheet https://www.ti.com/lit/ds/symlink/sn7432.pdf?ts=1739286146068&ref_url=https%253A%252F%252Fwww.ti.com%252Fproduct%252FSN7432

[6] Texas Instruments, SN74LS11 Datasheet https://www.ti.com/lit/ds/symlink/sn74ls11.pdf?ts=1739204712496&ref_url=https%253A%252F%252Fwww.google.com%252F

[7] Texas Instruments, SN7404 Datasheet https://www.ti.com/lit/ds/symlink/sn7404.pdf?ts=1739286173725&ref_url=https%253A%252F%252Fwww.ti.com%252Fproduct%252FSN7404

[8] ALL DATASHEET, OP497 Datasheet https://www.alldatasheet.com/datasheet-pdf/view/49060/AD/OP497.html

[9] Texas Instruments, NE555 Datasheet https://www.ti.com/lit/ds/symlink/ne555.pdf?ts=1739263057658&ref_url=https%253A%252F%252Fwww.mouser.cn%252F

[10] Analog Devices, AD744 Datasheet https://www.analog.com/media/en/technical-documentation/data-sheets/ad744.pdf