eSim Semester long Internship Autumn 2025

On

Integrated Circuit Design using Subcircuit feature of eSim

Submitted by

Anish Ramesh Khapare

Under the guidance of

Prof. Prabhu Ramachandran
Aerospace Engineering Department
IIT Bombay

January 24, 2026

Acknowledgment

I would like to express my deepest gratitude to Prof. Prabhu Ramachandran for providing me the opportunity to be a part of the FOSSEE internship program and for supporting open-source engineering tool development at IIT Bombay. His guidance and vision have encouraged students and researchers to actively contribute to the open-source ecosystem.

I would also like to acknowledge Prof. Kannan M. Moudgalya for his foundational role in establishing and nurturing the FOSSEE initiative. His contributions toward open-source education and the creation of the FOSSEE fellowship framework have directly shaped the platform through which this internship was undertaken.

My sincere appreciation extends to my mentor, **Mr. 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**, **Ms. Shanthi Priya K**, and **Mr. Aditya M**, 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.

I would like to express my sincere gratitude to A. C. Patil College of Engineering, Navi Mumbai, for their academic support and encouragement during this internship and special thanks to our department head **Dr. Dipak Marathe** , **Prof Amol G. Patil** as a FOSSEE institute mentor and all faculty members of the department of Electronics and Telecommunication Engineering for encouraging me to attend this virtual internship.

Contents

Chapter 1

Introduction

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

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

1.1 eSim

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

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

1.2 NgSpice

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

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

1.3 Makerchip

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

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

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

Chapter 2

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 should be capable of performing schematic creation, PCB design and circuit simulation (analog, digital and mixed-signal). It should provide facilities to create new models and components. Thus, eSim offers the following features.-

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

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

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

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

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

Chapter 3

Problem Statement

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

3.1 Approach

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

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

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

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

4. Schematic Testing : Once the test circuits are ready, now it's time to simulate the test circuits so that the output can be obtained in the form of wave-forms and plots Here we take help of KiCad to NgSpice conversion and Simulation feature in eSim. If the output of the test circuit is not as per expectation, this implies that

the test case has failed, and there is some error in the schematic. In such cases we go back to the design phase of the IC or the test circuits, to look for possible errors and then repeat the testing process again after making required changes. Once the

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

Chapter 4

Integrated Circuits

4.1 SN74180 - Odd/Even Parity Generator/Checker

4.1.1 Description

The 74180 is a 9-bit Odd/Even Parity Generator/Checker, used in digital systems for error detection by adding or verifying a parity bit (odd or even) in data streams, featuring inputs for 8 data bits and a parity control, plus outputs for odd/even parity, easily expandable for larger data word lengths, and compatible with other 7400-series logic.

4.1.2 Key Features

- **Purpose:** Generates or checks parity (odd/even) for 9 bits (8 data + 1 parity).
- **Compatibility:** Fully compatible with standard TTL and DTL circuits.
- **Inputs:** Handles 8 data inputs and offers odd/even control inputs to select mode.
- **Outputs:** Provides dedicated outputs for odd parity and even parity.

4.1.3 Pin Diagram

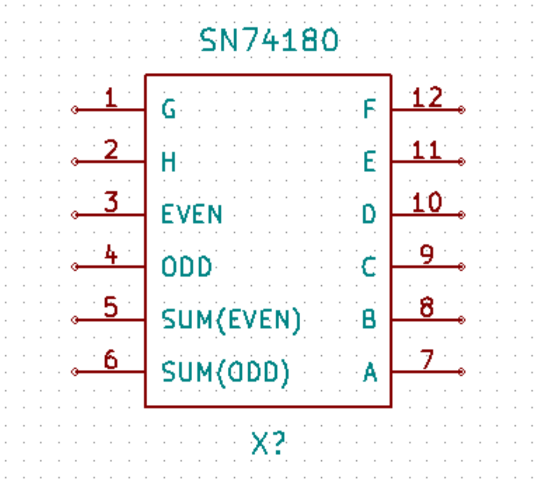

Figure 4.1: Pin diagram of SN74180

4.1.4 Schematic Diagram

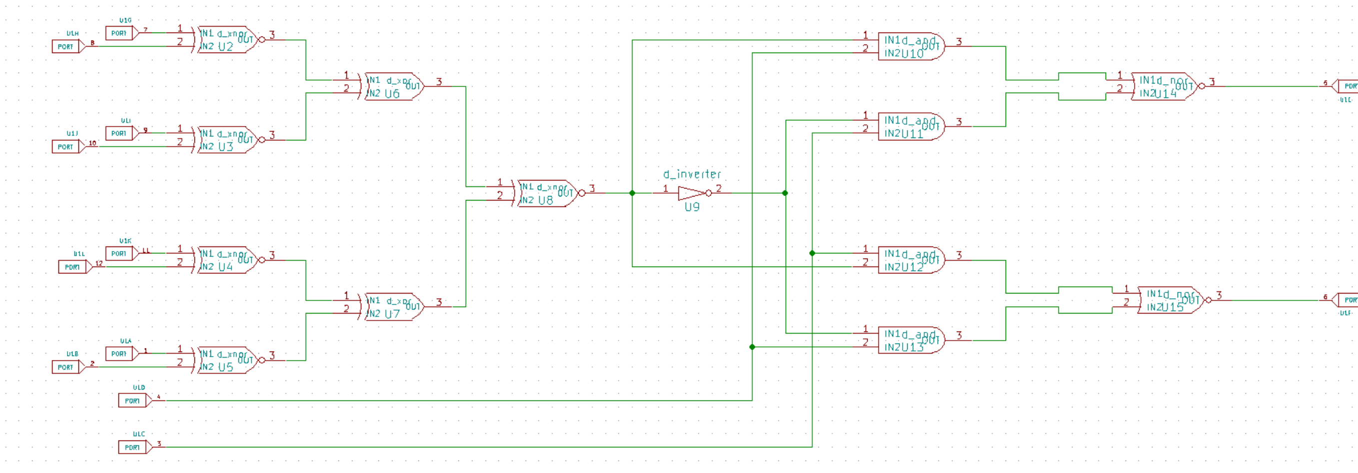

Figure 4.2: Schematic diagram of SN74180

4.1.5 Test Circuit

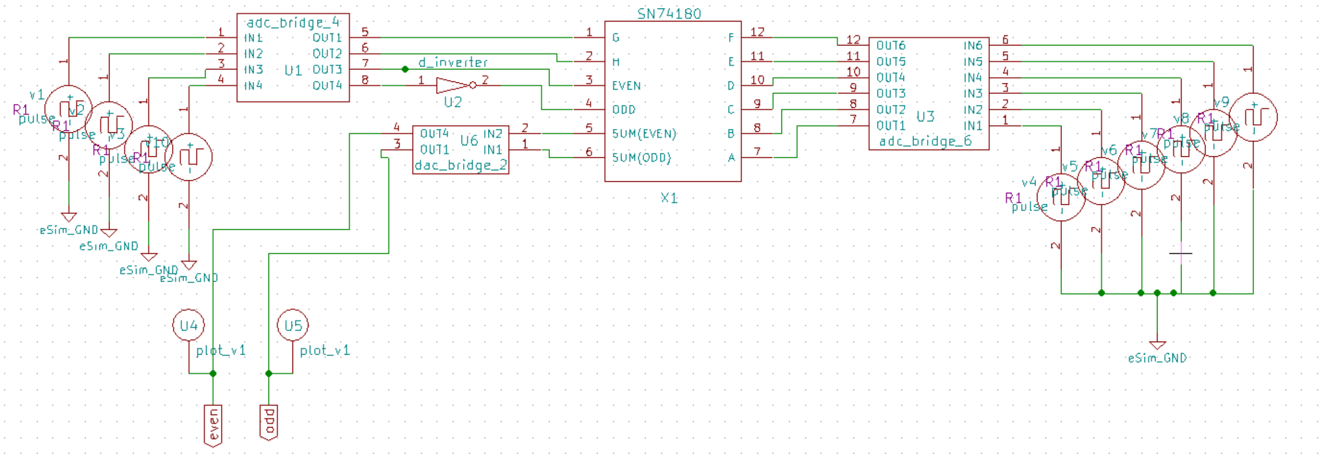

Figure 4.3: Test Circuit of SN74180

4.1.6 Output

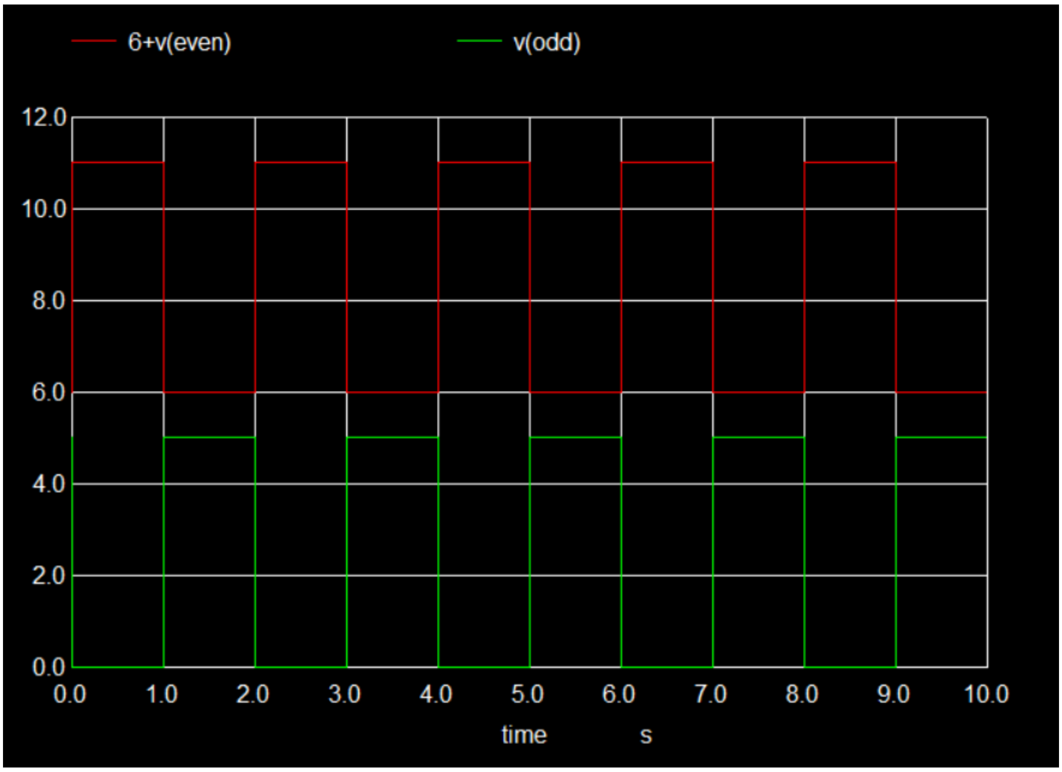

Figure 4.4: Output of SN74180

4.2 SN741LS94 - Bidirectional Universal Shift Register

4.2.1 Description

The SN74LS194 is a 4-bit, high-speed, bidirectional universal shift register that is part of the TTL logic family. It is a versatile integrated circuit capable of shifting data in both left and right directions, as well as performing parallel load operations.

4.2.2 Key Features

- **Data Capacity:** 4 bits.
- **Operating Modes:** Four distinct modes are controlled by two select inputs (S_0 and S_1):
  - **Synchronous Parallel Load:** Loads 4 bits of data simultaneously.
  - **Shift Right:** Shifts data in the direction of Q_A toward Q_D.
  - **Shift Left:** Shifts data in the direction of Q_D toward Q_A.
  - **Holds Data**
- **Clocking:** Operations are synchronous and triggered by the positive edge of the clock input.
- **Inputs/Outputs:** Features both parallel inputs (A to D) and outputs (Q_A to Q_D), as well as dedicated serial inputs for shifting right (DSR) and shifting left (DSL).

4.2.3 Pin Diagram

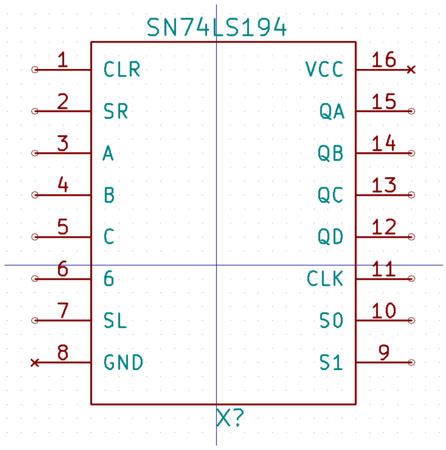

Figure 4.5: Pin diagram of SN741LS94

4.2.4 Schematic Diagram

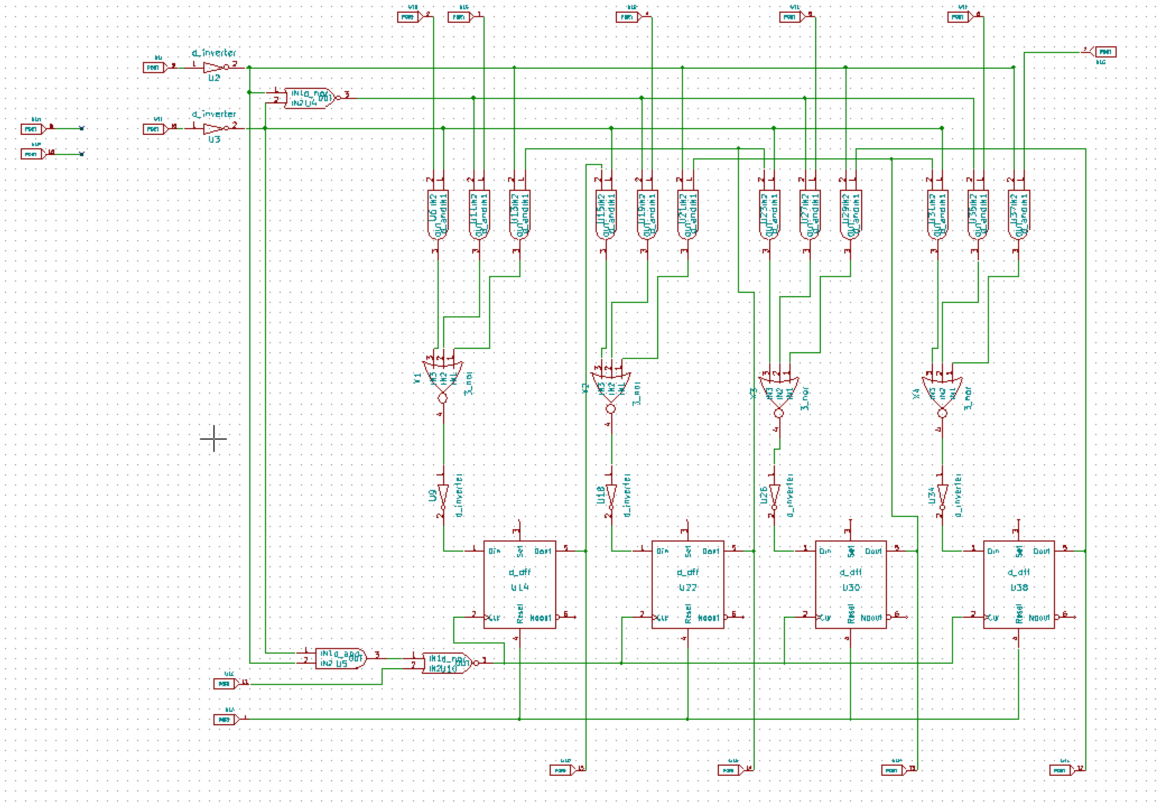

Figure 4.6: Schematic diagram of SN741LS94

4.2.5 Test Circuit

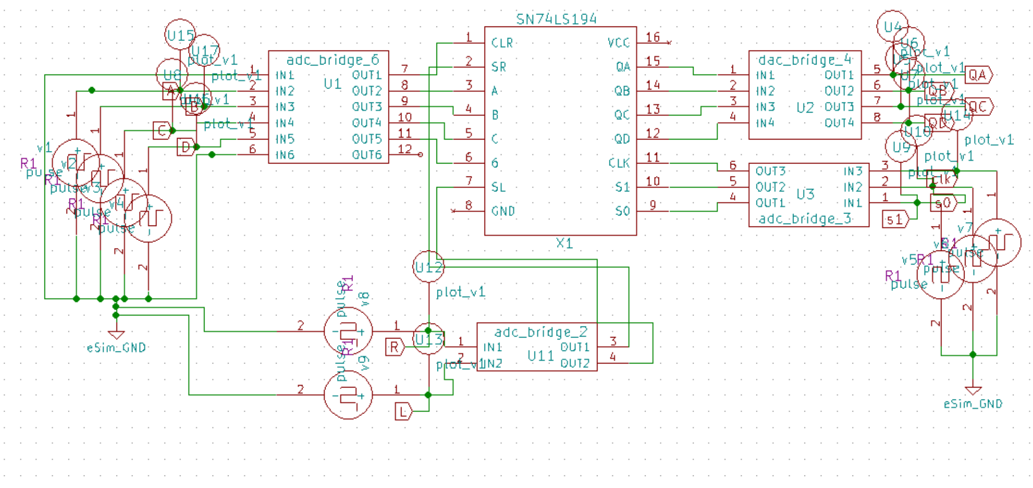

Figure 4.7: Test Circuit of SN741LS94

4.2.6 Output

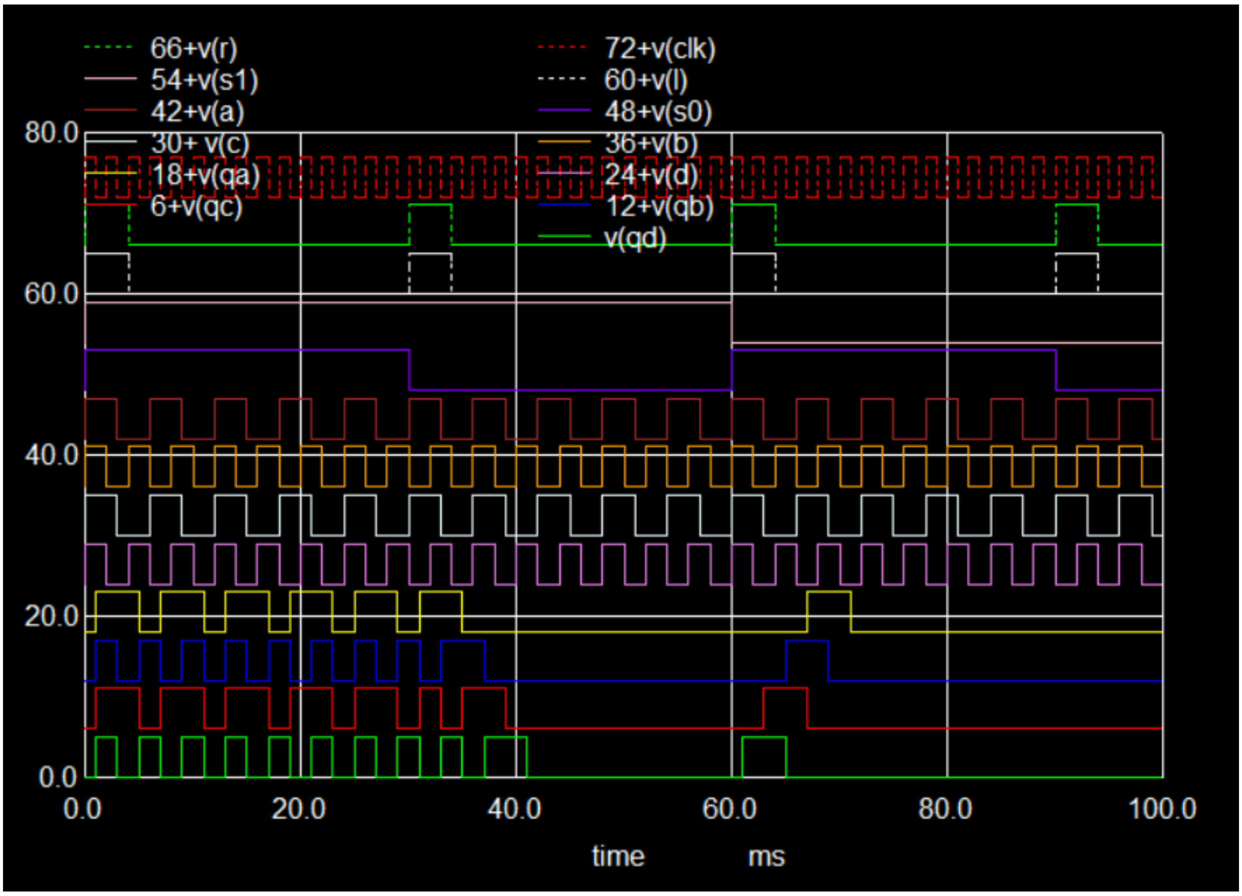

Figure 4.8: Output waveform of SN741LS94

4.3 LM325 - Voltage Regulator

4.3.1 Description

The LM325 is a dual polarity tracking voltage regulator IC , designed to provide balanced positive and negative voltages (often ±15V or ±12V) up to 100mA, featuring adjustable current limiting and internal thermal overload protection, used in power supplies requiring symmetrical dual rails.

4.3.2 Key Features

- **Dual Tracking:** Provides balanced $+V$ and $-V$ outputs from a single input.
- **Output Current:** Up to 100 mA.
- **Input Voltage:** Can handle input voltages up to +30V.
- **Adjustable Current Limit:** Allows setting the maximum output current.

4.3.3 Pin Diagram

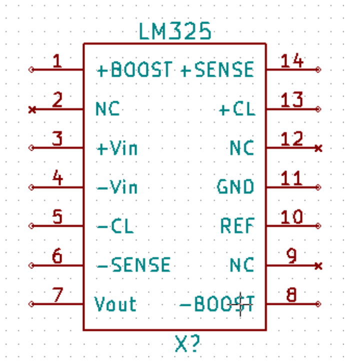

Figure 4.9: Pin diagram of LM325

4.3.4 Schematic Diagram

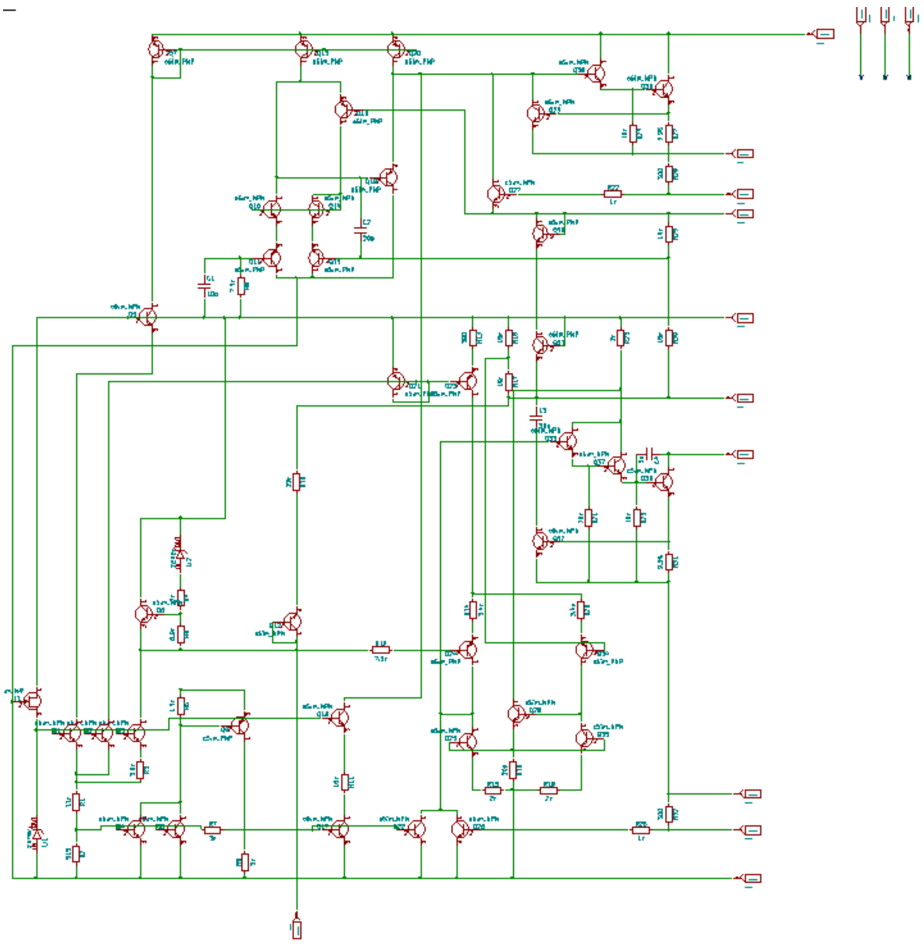

Figure 4.10: Schematic diagram of LM325

4.3.5 Test Circuit

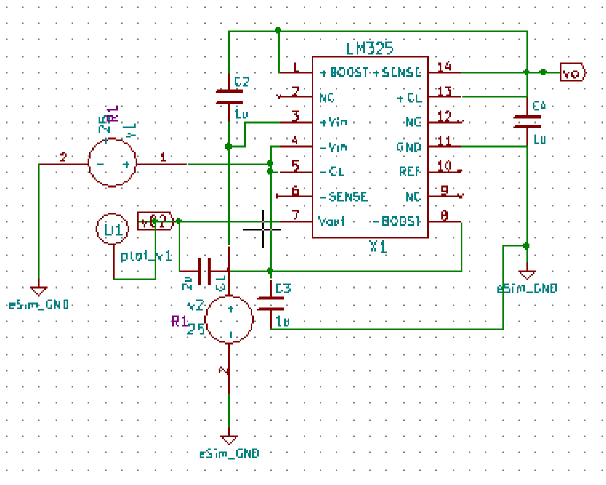

Figure 4.11: Test Circuit of LM325

4.3.6 Output

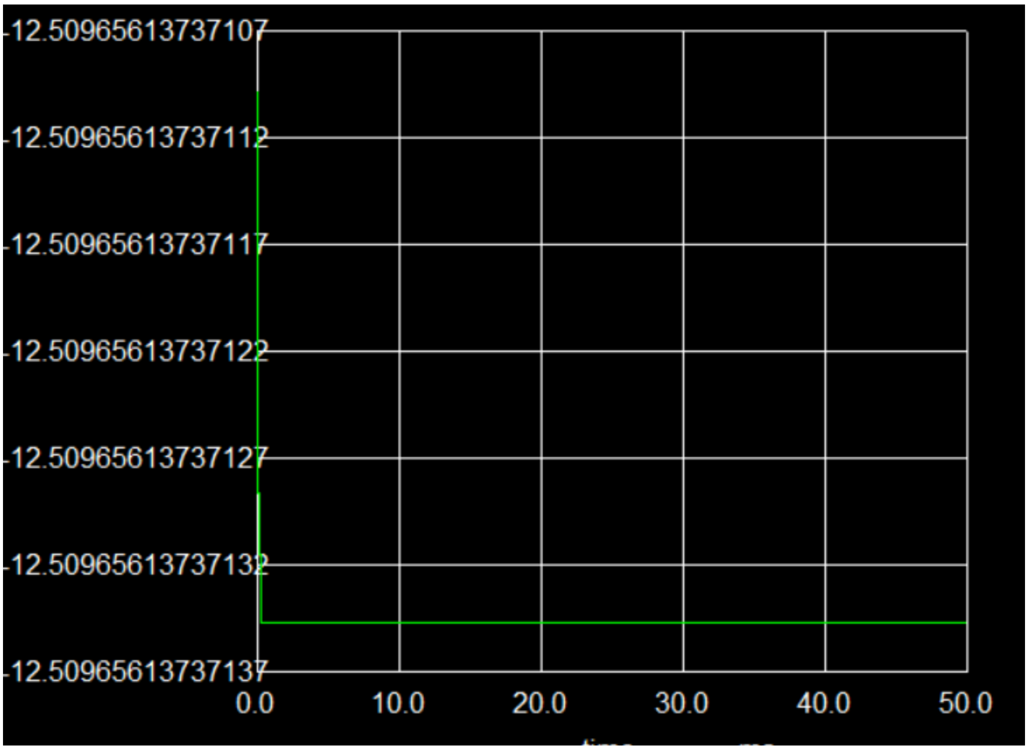

Figure 4.12: Output waveform of LM325

4.4 LM161/361 - High Speed Differential Comparators

4.4.1 Description

The LM161 is a very high-speed differential voltage comparator that features complementary TTL outputs and independent strobes. It is optimized for speed and low input offset voltage, with applications in high-speed analog-to-digital converters and disk file systems.

4.4.2 Key Features

- **Guaranteed High Speed:** The device offers a maximum propagation delay of 20 ns, making it suitable for applications requiring fast response times.
- **Differential Input:** It accepts differential input voltages, which improves noise immunity and versatility in various circuit designs.
- **Complementary TTL Outputs:** It provides two outputs with tight delay matching and maximum skew, directly compatible with TTL logic families.
- **Wide Supply Voltage Range:** It can operate from both single 5V supplies or dual op-amp supplies up to ±15V, offering flexibility in power supply design.
- **Independent Strobes:** The outputs can be enabled or disabled using independent strobe inputs, which allows for control and multiplexing in system applications.

4.4.3 Pin Diagram

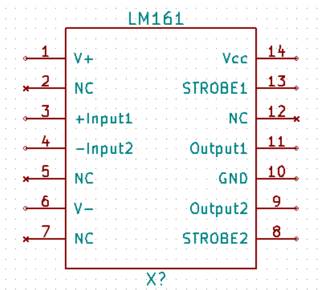

Figure 4.13: Pin diagram of LM161/361

4.4.4 Schematic Diagram

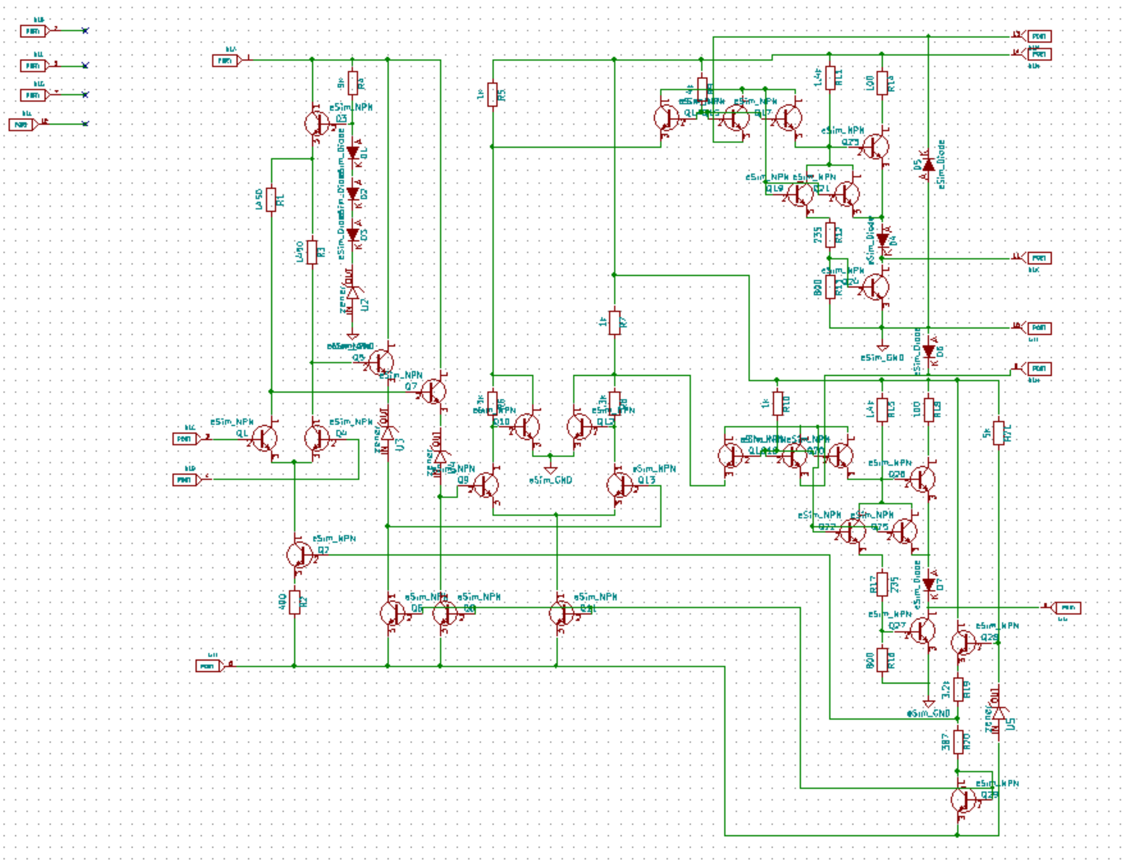

Figure 4.14: Schematic diagram of LM161/361

4.4.5 Test Circuit

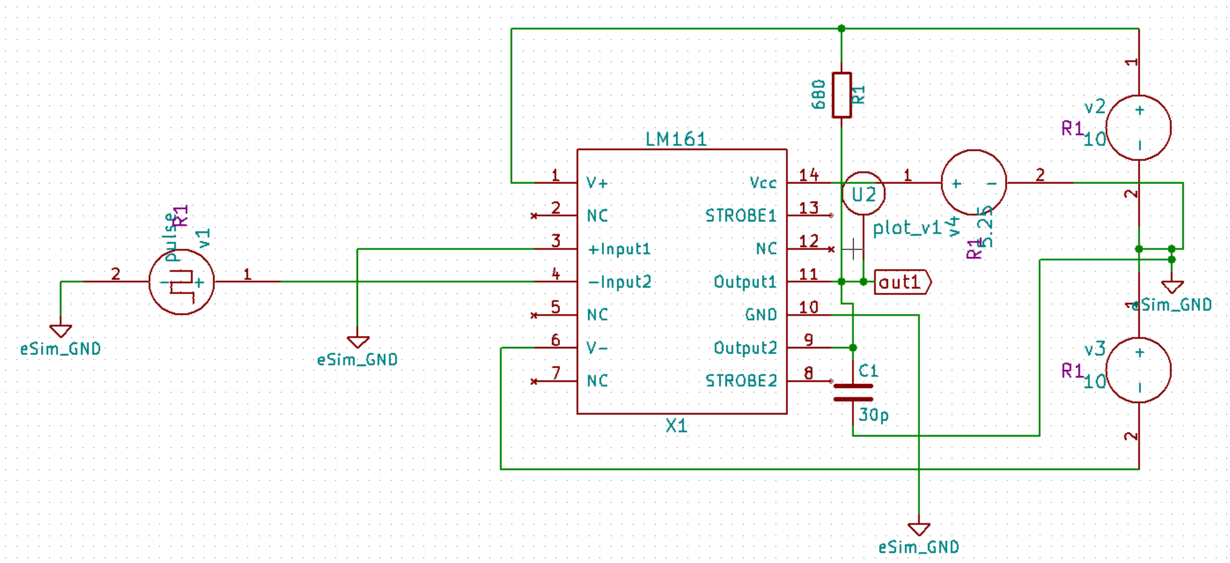

Figure 4.15: Test Circuit of LM161/361

4.4.6 Output

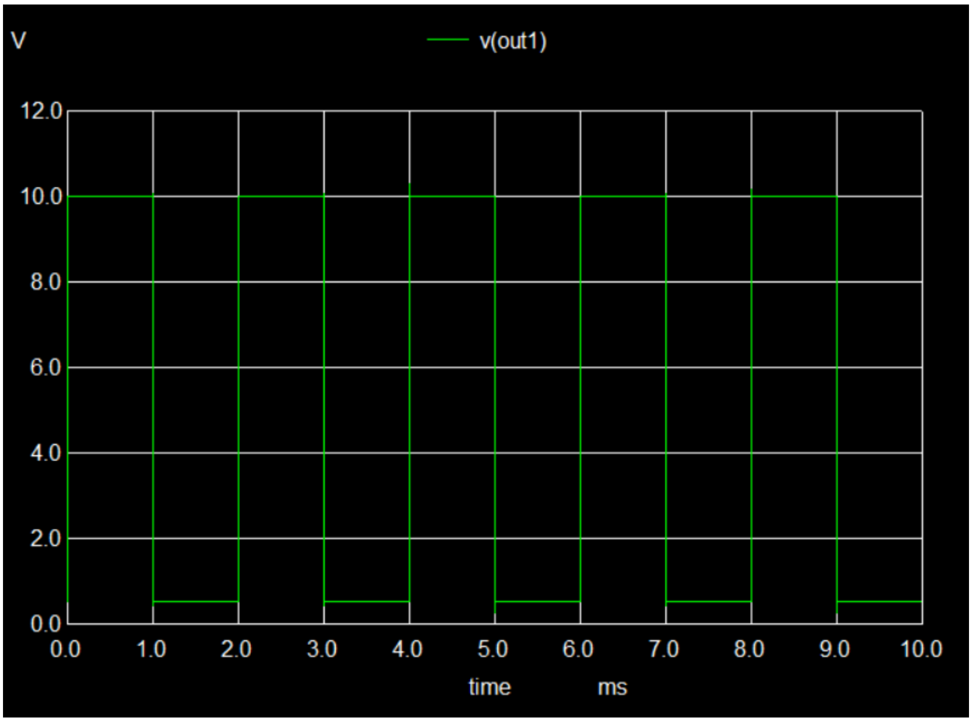

Figure 4.16: Output waveform of LM161/361

4.5 MC100EL1648 - Voltage Controlled Oscillator

4.5.1 Description

The MC100EL1648 (manufactured by onsemi, formerly Fairchild Semiconductor) is a 5V ECL Voltage Controlled Oscillator (VCO) amplifier designed for high-speed applications such as digital telecommunications and electronic test equipment.

4.5.2 Key Features

- **Functionality:** It functions as a voltage-controlled oscillator amplifier but requires an external parallel tank (LC) circuit to determine the oscillation frequency. A varactor diode can be incorporated into the tank circuit to provide the voltage-variable input for the VCO.
- **High Operating Frequency:** The device offers a typical operating frequency of up to 1100 MHz (1.1 GHz).
- **Low Power Consumption:** It typically draws only 19 mA of supply current at 5.0 Vdc.
- **Power Supply Modes:** It supports both Positive ECL (PECL) and Negative ECL (NECL) operating modes.
  - **PECL Mode:** $V_{CC} = 4.2$ V to 5.5 V with $V_{EE} = 0$ V.
  - **NECL Mode:** $V_{CC} = 0$ V with $V_{EE} = -4.2$ V to -5.5 V.
- **Automatic Gain Control (AGC):** The circuit incorporates AGC to optimize frequency response and limit current through the oscillator core.

4.5.3 Pin Diagram

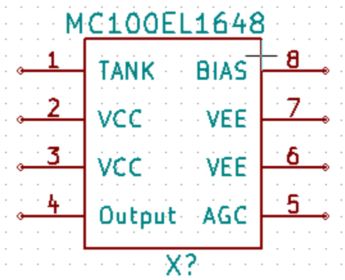

Figure 4.17: Pin diagram of MC100EL1648

4.5.4 Schematic Diagram

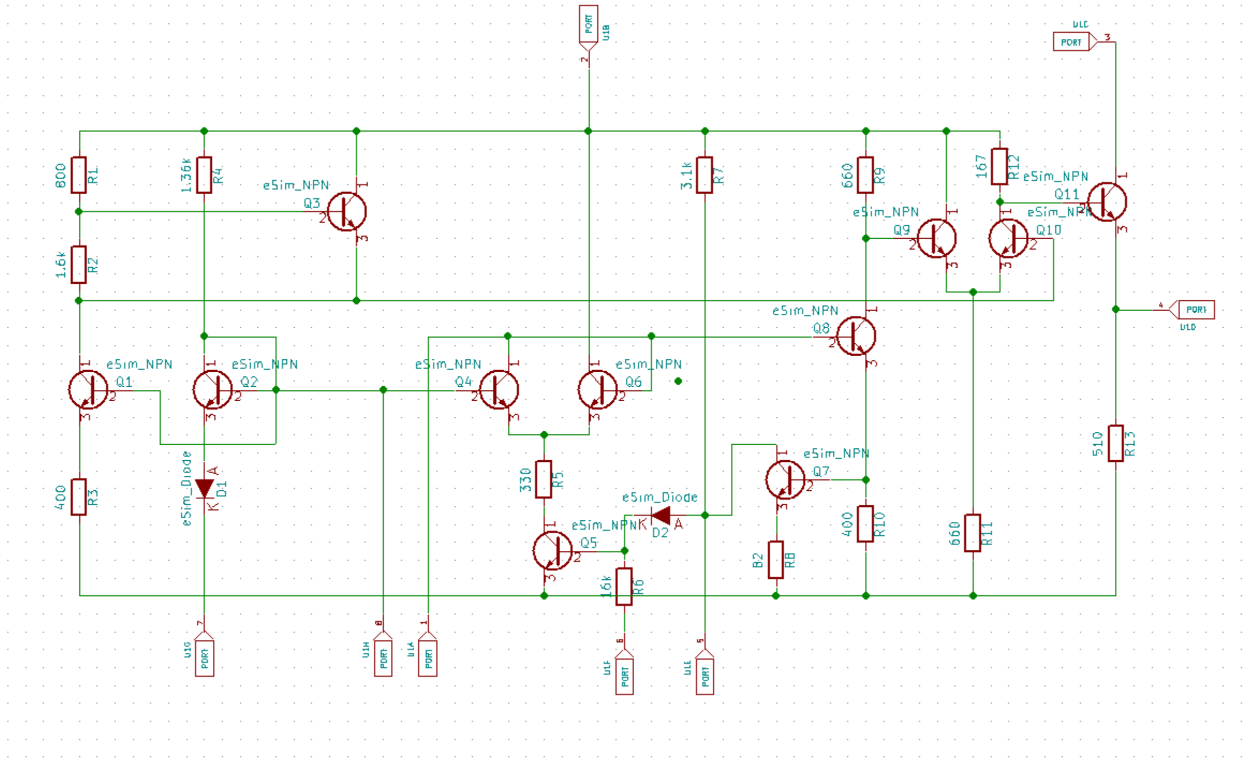

Figure 4.18: Schematic diagram of MC100EL1648

4.5.5 Test Circuit

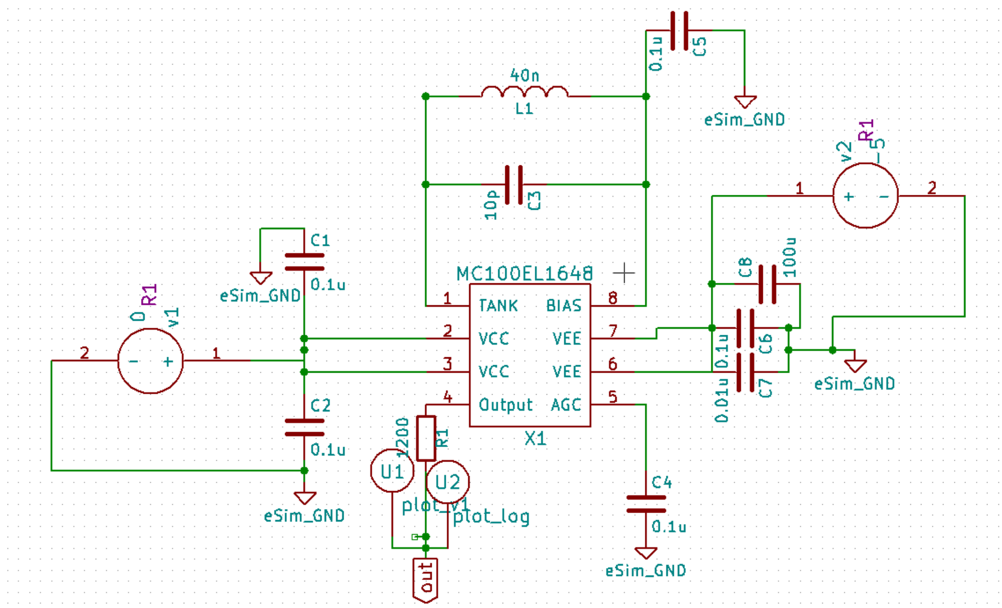

Figure 4.19: Test Circuit of MC100EL1648

4.5.6 Output

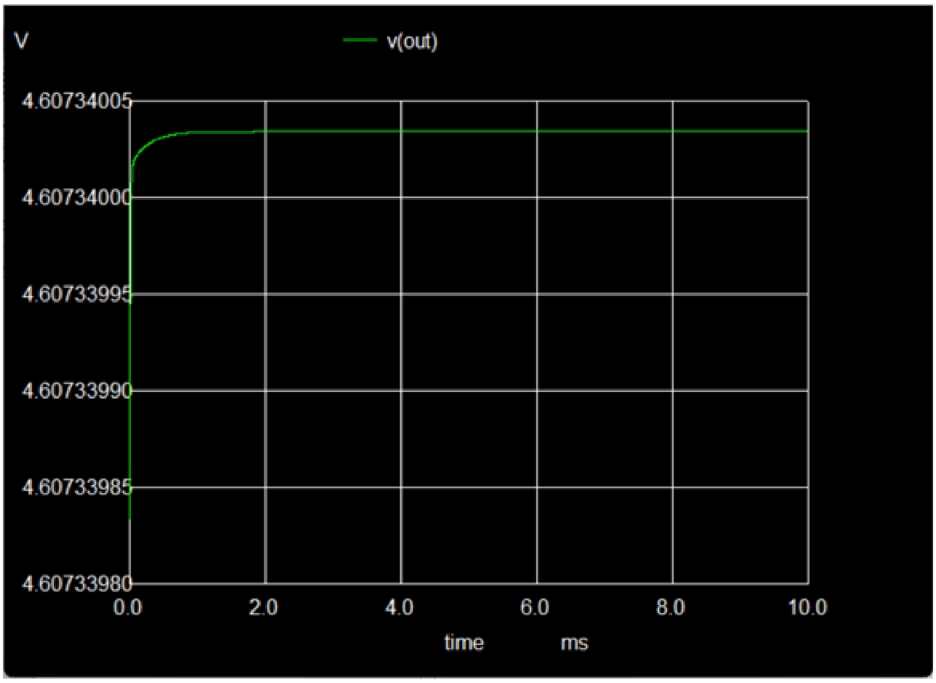

Figure 4.20: PECL mode - MC100EL1648

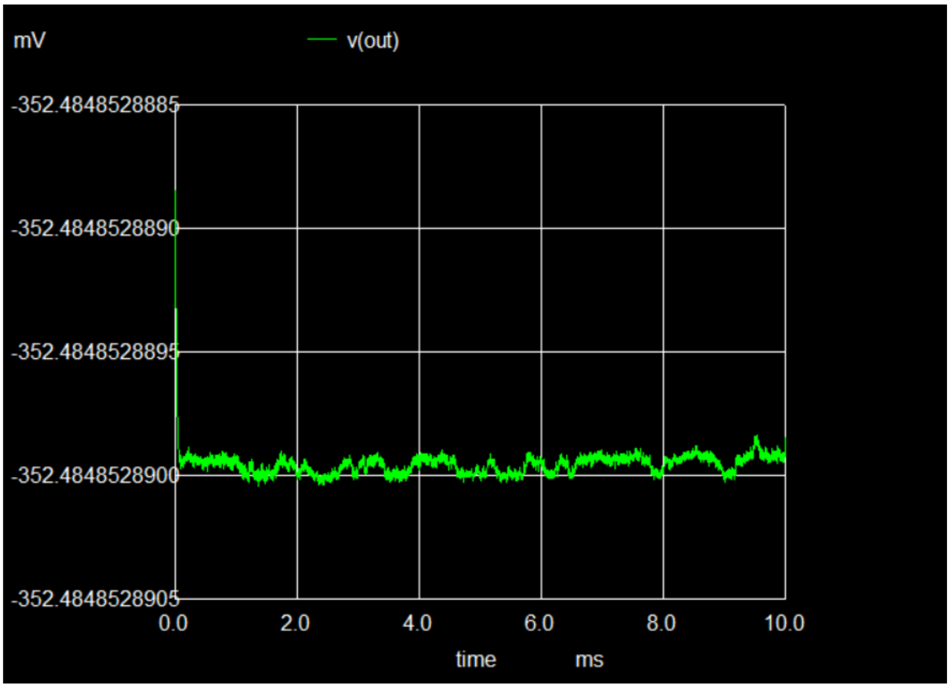

Figure 4.21: NECL mode - MC100EL1648

4.6 CA3085- Adjustable Positive Voltage Regulator

4.6.1 Description

CA3085 is a adjustable positive voltage regulator, the key aspect of this component is its adjustability, allowing users to set the output voltage within a wide range using external components. It incorporates several internal features such as frequency compensation, a temperature-compensated reference voltage, and current-limiting circuitry for protection.

4.6.2 Key Features

- **Adjustable Output Voltage:** The output voltage can be adjusted from as low as 1.7 V up to 45 V (or 36 V depending on the specific variant).
- **Output Current:** It can deliver an output current of up to 100 mA without external pass transistors.
- **Wide Input Voltage Range:** The device can handle a maximum input voltage of up to 40 V or 30 V depending on the variant.

4.6.3 Pin Diagram

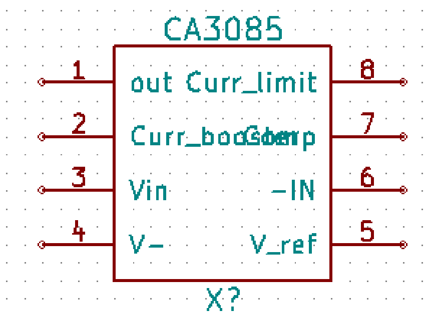

Figure 4.22: Pin diagram of CA3085

4.6.4 Schematic Diagram

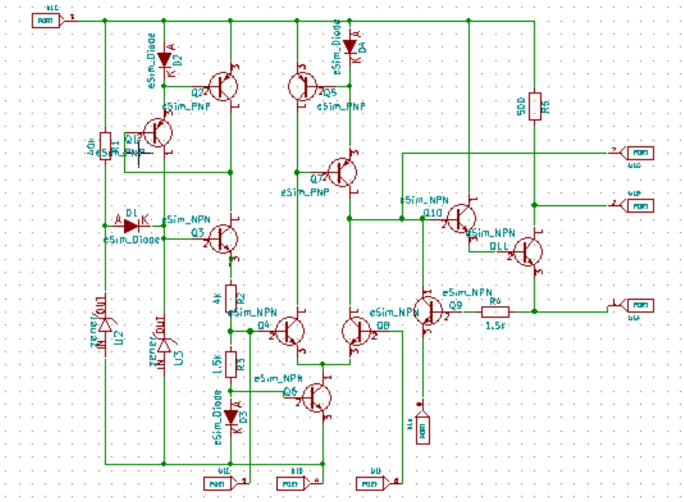

Figure 4.23: Schematic diagram of CA3085

4.6.5 Test Circuit

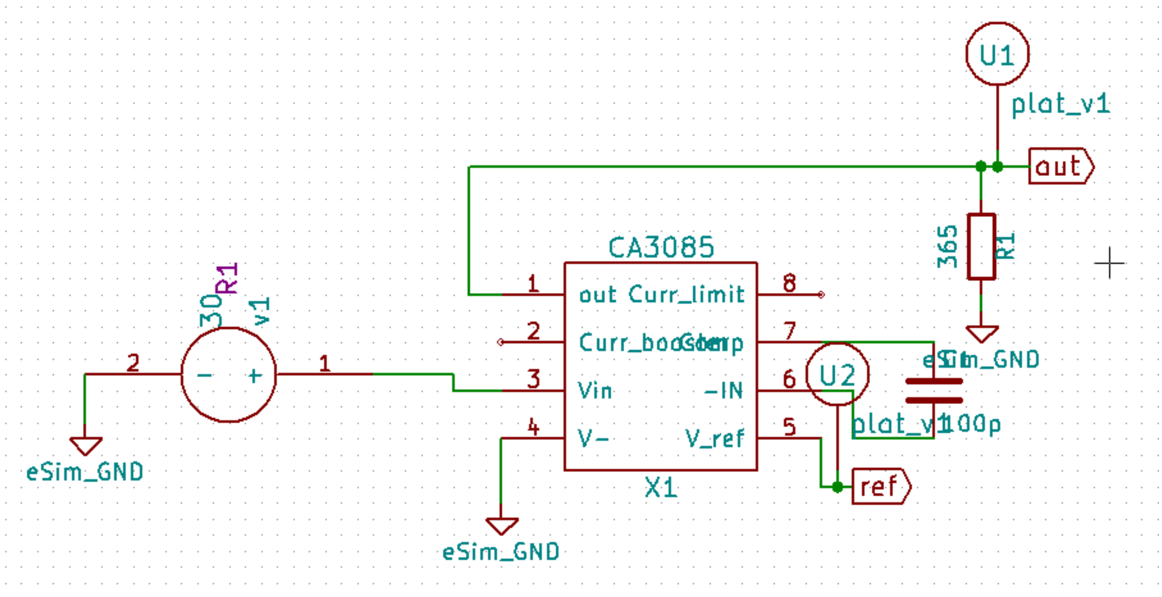

Figure 4.24: Test Circuit of CA3085

4.6.6 Output

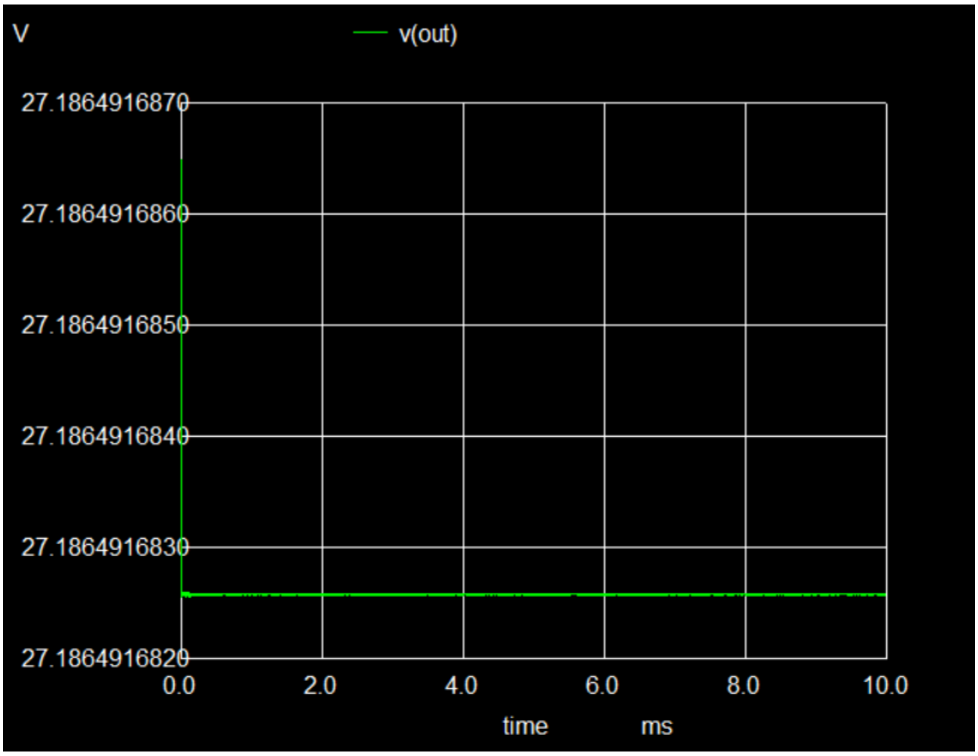

Figure 4.25: Regulated Voltage - CA3085

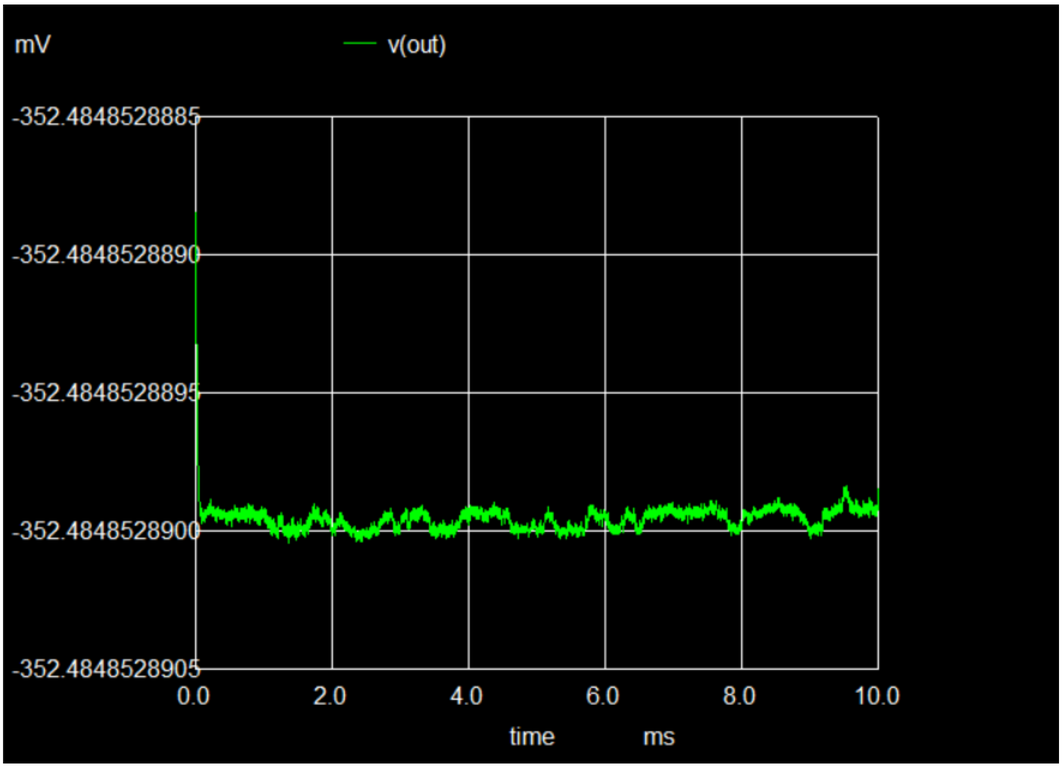

Figure 4.26: Reference Voltage - CA3085

4.7 CA3002 - IF Amplifier

4.7.1 Description

The CA3002 is a classic linear integrated circuit, primarily a differential amplifier or intermediate frequency (IF amplifier), supplied in a 10-lead hermetic TO-5 style package.

4.7.2 Key Features

- **Function:** It is designed for use in various applications including RC-coupled IF amplifiers, video amplifiers, envelope detectors, and trigger circuits.
- **Design:** It utilizes a differential amplifier configuration, a fundamental building block in analog circuit design.
- **Operating Conditions:**
  - Maximum power supply voltage of ±6 V.

4.7.3 Pin Diagram

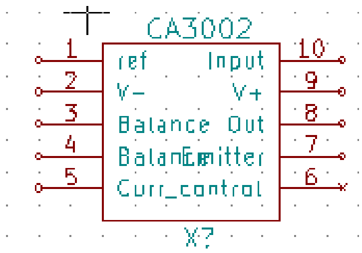

Figure 4.27: Pin diagram of CA3002

4.7.4 Schematic Diagram

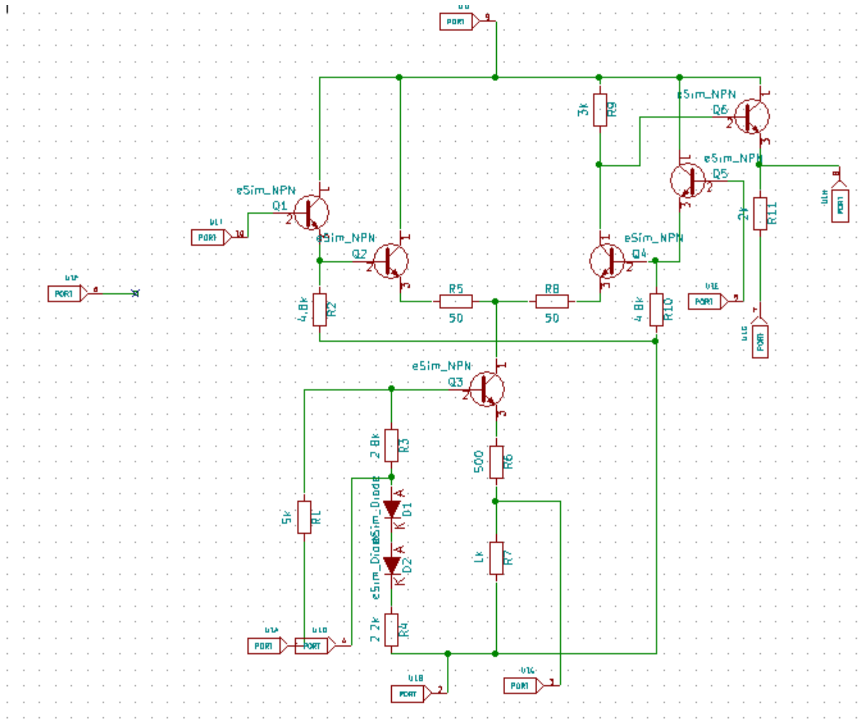

Figure 4.28: Schematic diagram of CA3002

4.7.5 Test Circuit

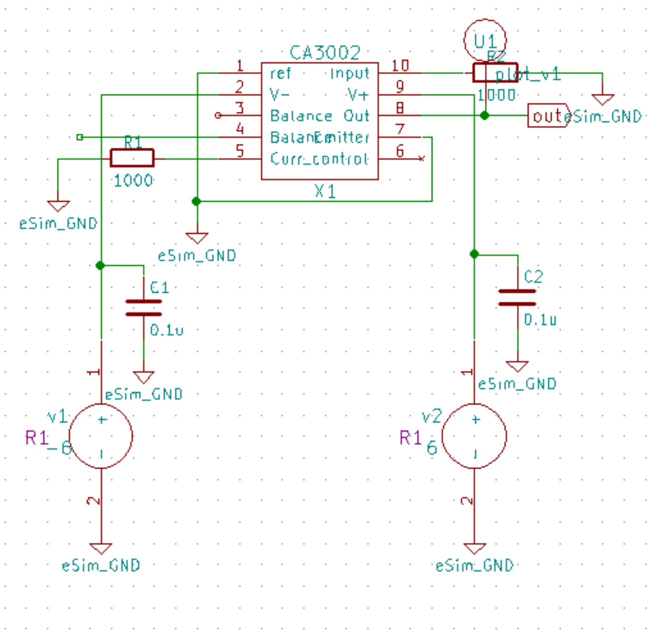

Figure 4.29: Test Circuit of CA3002

4.7.6 Output

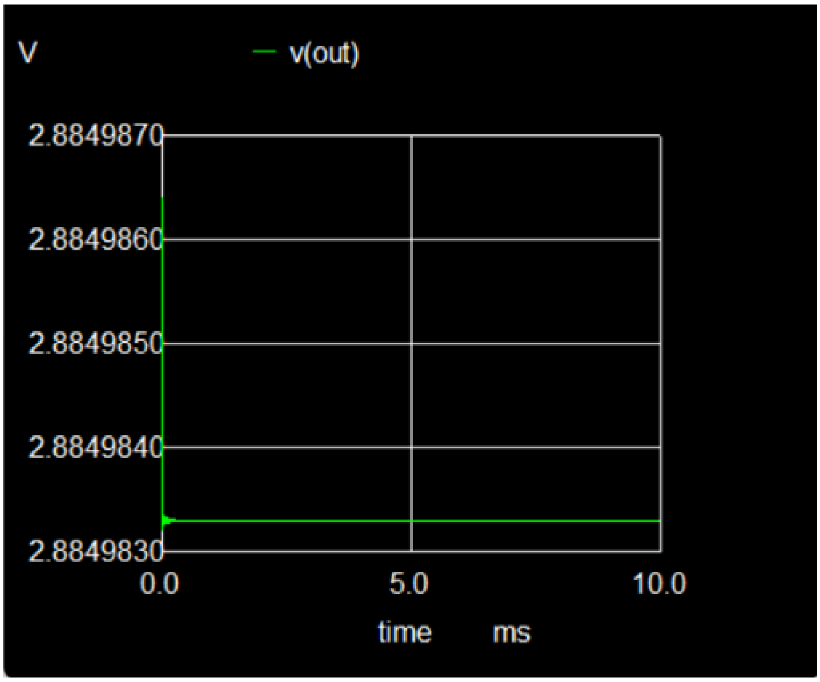

Figure 4.30: mode A- CA3002

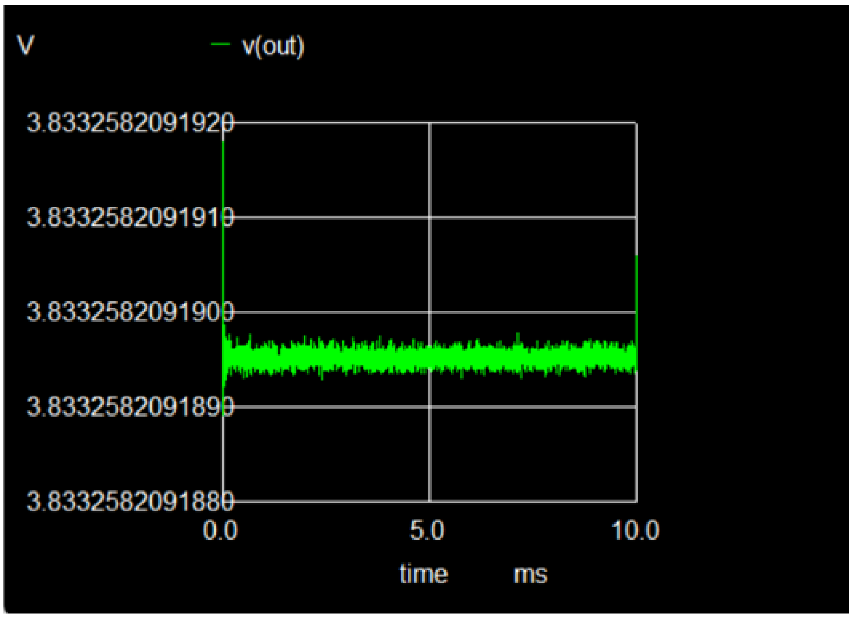

Figure 4.31: mode B - CA3002

4.8 CA3021 - Wideband Amplifier

4.8.1 Description

The IC CA3021 is a high-gain, wideband linear integrated circuit designed specifically for use as a video or broadband amplifier.

4.8.2 Key Features

- **Wide Bandwidth:** Designed for high-frequency video signals.
- **High Gain:** Offers significant signal amplification suitable for video-level signals.
- **Linear Operation:** Optimized for low distortion in amplification.

4.8.3 Pin Diagram

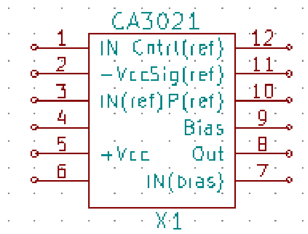

Figure 4.32: Pin diagram of CA3021

4.8.4 Schematic Diagram

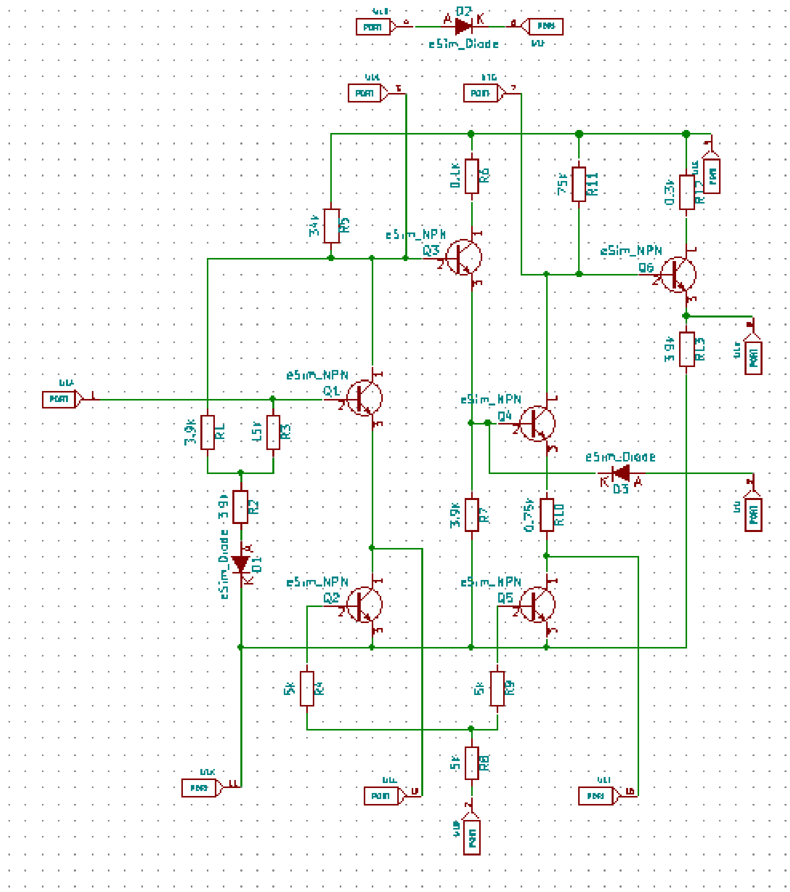

Figure 4.33: Schematic diagram of CA3021

4.8.5 Test Circuit

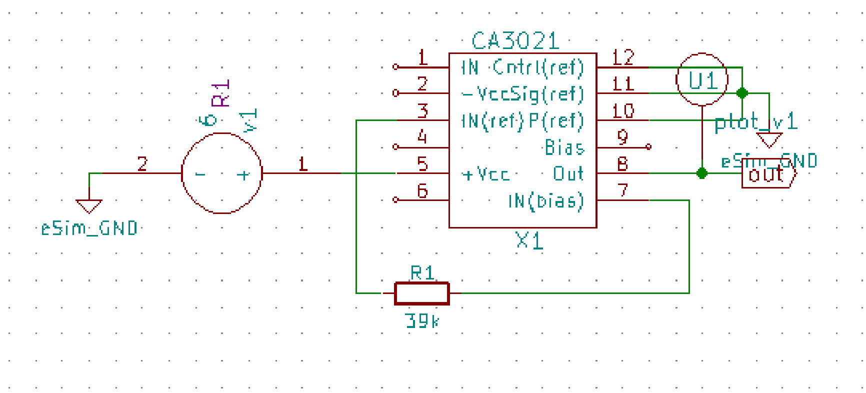

Figure 4.34: Test Circuit of CA3021

4.8.6 Output

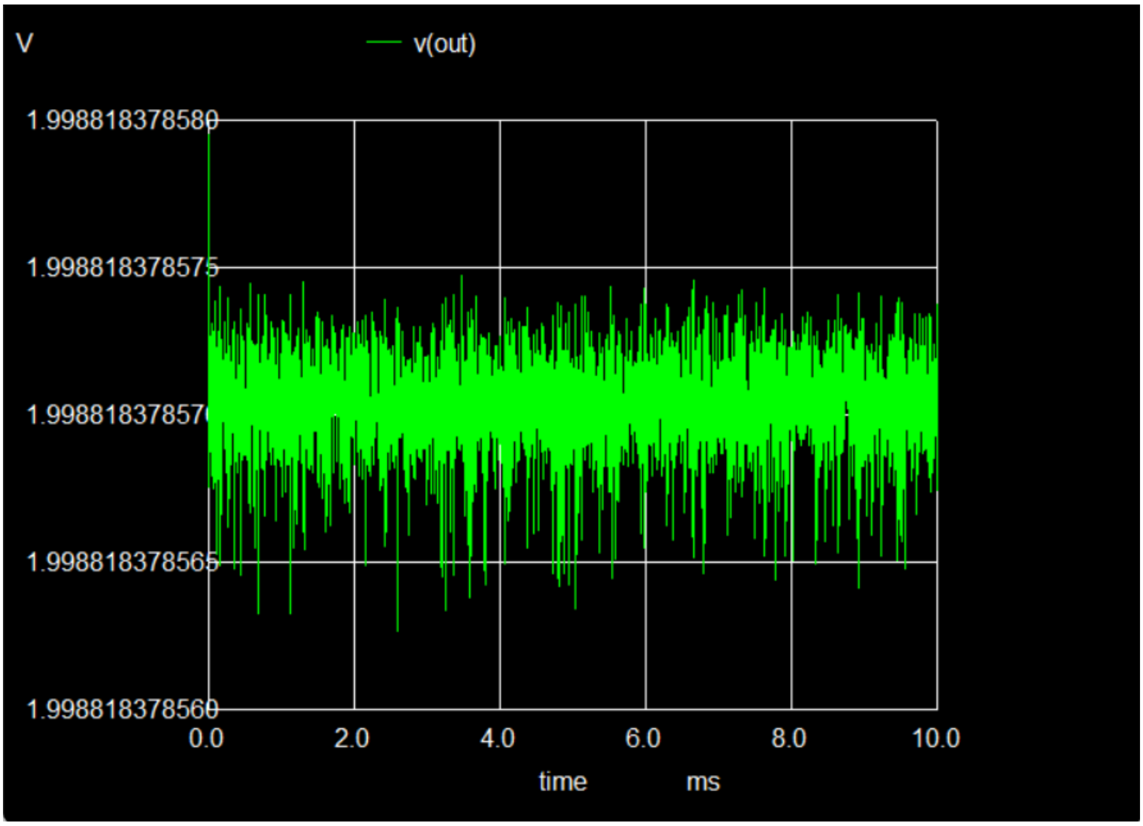

Figure 4.35: Output for testcircuit - CA3021

4.9 Failed ICs

4.9.1 LM359 - Dual Current Mode (Norton) Amplifiers

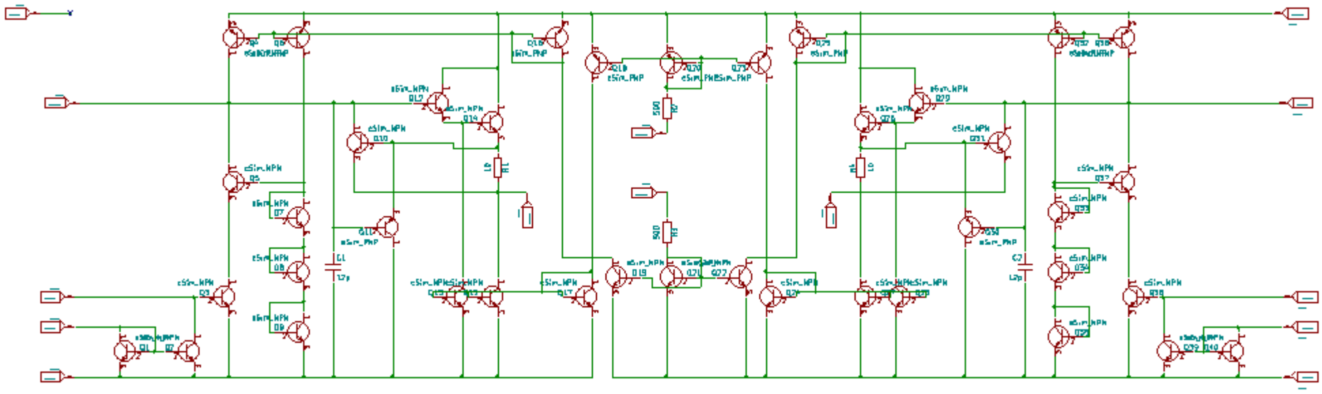

Figure 4.36: Schematic Diagram of LM359

4.9.2 LT1001 - Precision Operational Amplifiers

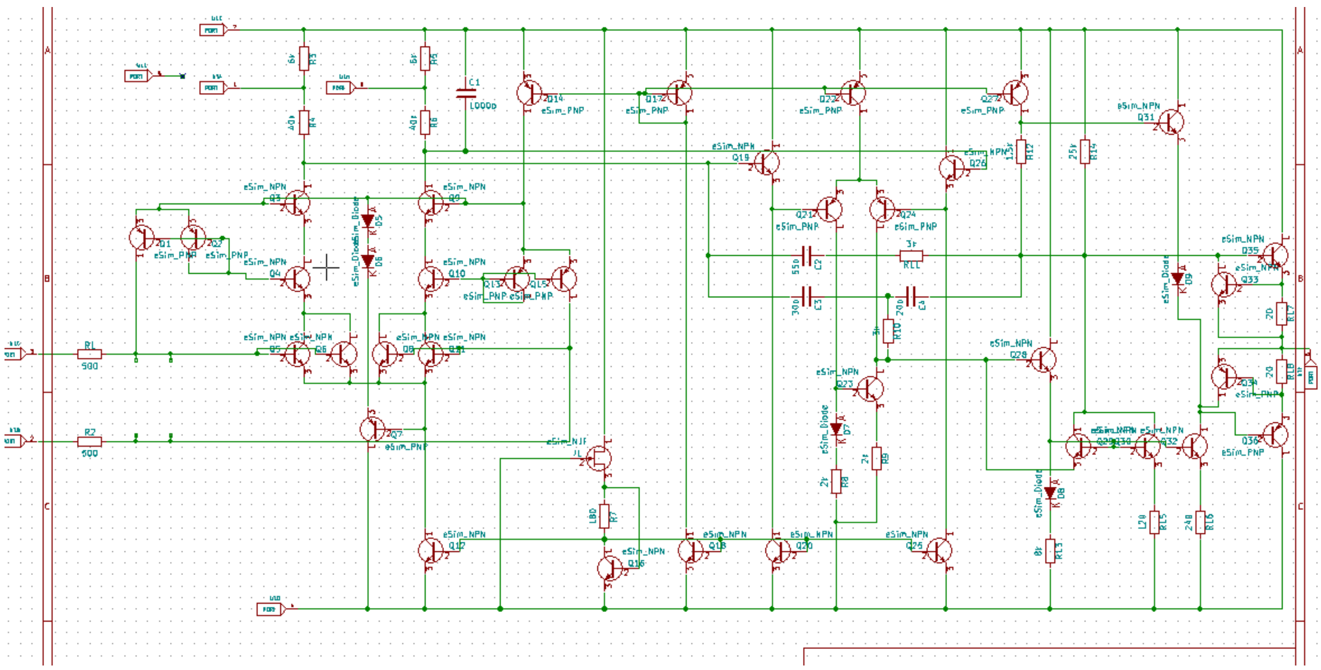

Figure 4.37: Schematic Diagram of LT1001

4.9.3 LT1012 - Precision Amplifiers

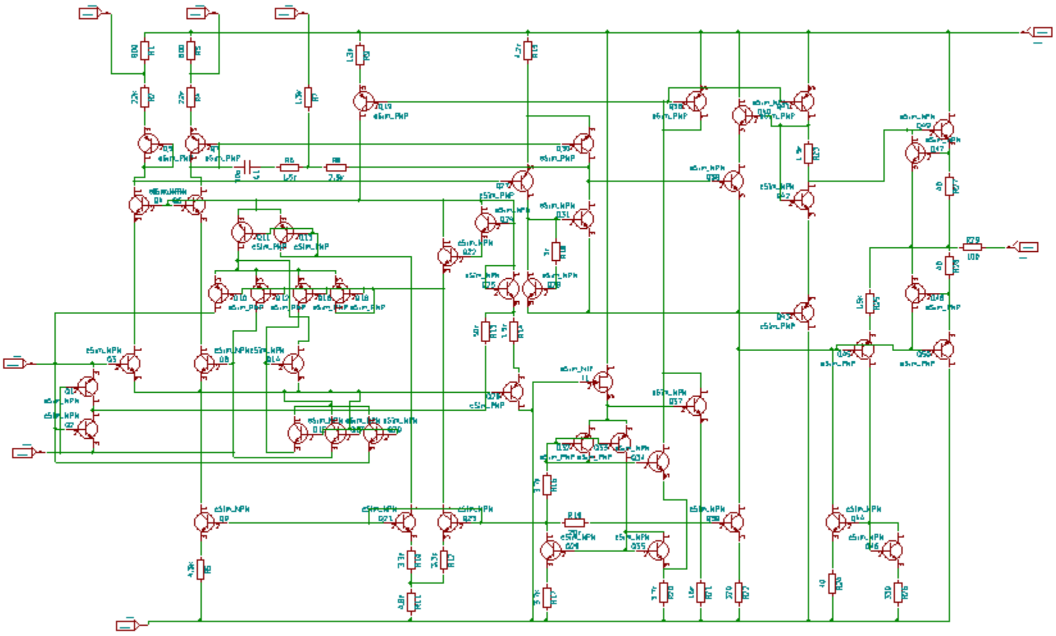

Figure 4.38: Schematic Diagram of LT1021

4.9.4 CA3015 - Operational Amplifiers

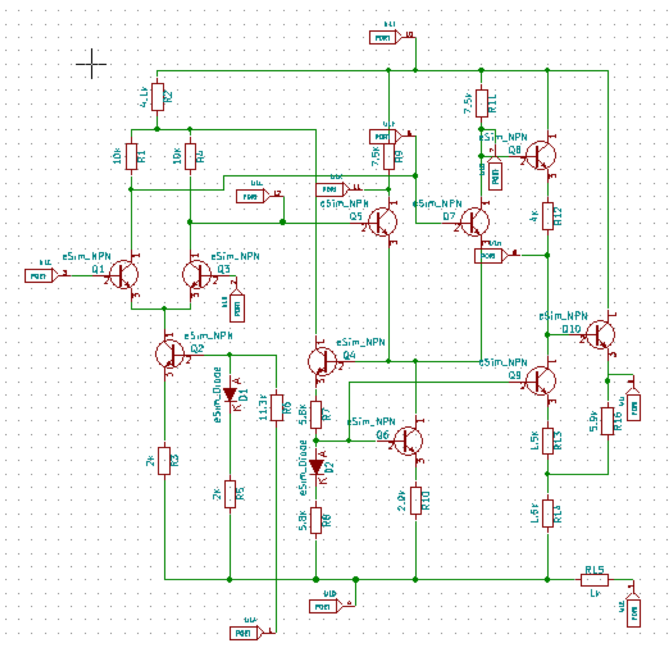

Figure 4.39: Schematic Diagram of CA3015

4.9.5 CA3091D - Four Quadrant Analog Multiplier

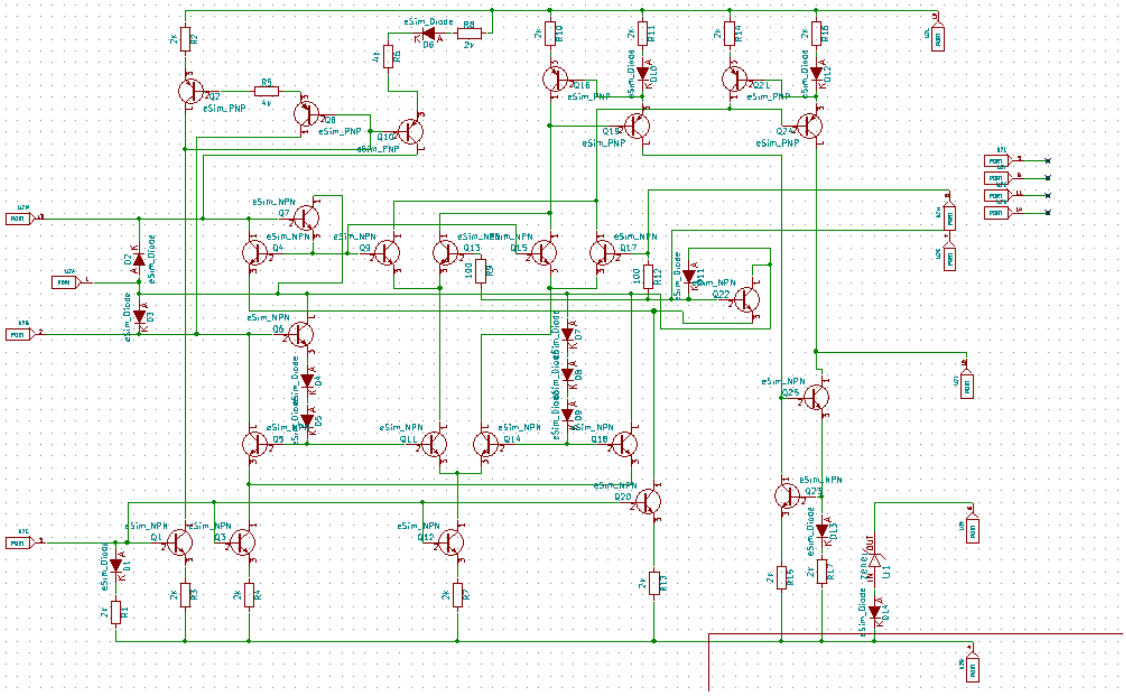

Figure 4.40: Schematic Diagram of CA3015

4.9.6 CA3052 - Four Independant AC Amplifier

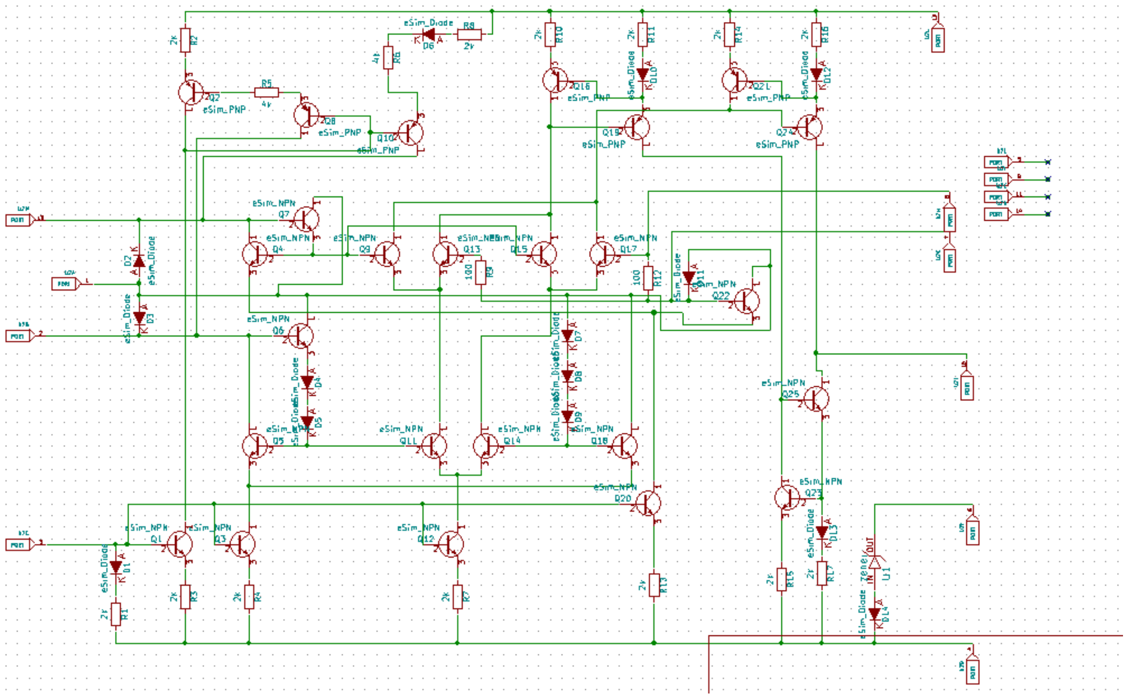

Figure 4.41: Schematic Diagram of CA3052

Chapter 5

Conclusion and Future Scope

We were able to achieve the target of designing and developing various Analog and Digital Integrated Circuit for the eSim library (Subcircuits). IC Subcircuits are designed and also verified based on specification given in official datasheets. 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.

Bibliography

[1] IC74180
URL: https://www.ti.com/lit/ds/symlink/sn54180.pdf?ts=1768720344150&ref_url=https%253A%252F%252Fwww.google.com%252F

[2] SN741LS94
URL:https://www.ti.com/lit/ds/symlink/sn74as194.pdf?ts=1768720399695&ref_url=https%253A%252F%252Fwww.google.com%252F

[3] LM325
URL: https://www.alldatasheet.com/datasheet-pdf/pdf/8842/NSC/LM325.html

[4] LM161/361
URL: https://www.ti.com/lit/ds/symlink/lm361.pdf

[5] MC100EL1648
URL: https://www.onsemi.com/pdf/datasheet/mc100el1648-d.pdf

[6] CA3085 , CA3002 , CA3021 : RCA Corporation, Linear Integrated Circuits. Somerville, NJ, USA: RCA Solid State Division, 1970s.
URL: https://www.worldradiohistory.com/BOOKSHELF-ARH/Technology/RCA-Books/RCA-Linear-Integrated-Circuits-1977.pdf?utm_source=chatgpt.com