# Semester long internship Report

On

**IC Design using Subcircuit in eSim**

Submitted by

**Sadichha Patil**

Under the guidance of

**Prof.Kannan M. Moudgalya**
Chemical Engineering Department
IIT Bombay

June 3, 2025

# Acknowledgment

I take this opportunity to express my sincere gratitude to the entire FOSSEE team at IIT Bombay for providing me with the invaluable opportunity to work on the eSim internship project. It has been an enriching and transformative journey that significantly contributed to my technical and professional growth.

I am deeply thankful to Prof. Kannan M. Moudgalya, whose vision and leadership of the FOSSEE initiative have enabled students like me to actively engage with open-source tools in the field of electronic design. His commitment to accessible and high-quality education through open-source platforms is truly inspiring.

I would like to extend my heartfelt thanks to Mr. Sumanto Kar, who served as my mentor throughout this internship. His patient guidance, insightful feedback, and continuous encouragement made this complex learning process much more structured and effective. His support was instrumental in helping me understand subcircuit modeling and simulation using eSim.

I also appreciate the collaborative environment fostered by the eSim team, where sharing of ideas and peer learning were encouraged. The support from fellow interns and technical coordinators played a vital role in helping me overcome challenges and stay motivated throughout the internship.

Finally, I would like to thank my college faculty and peers for their moral support and flexibility, which allowed me to manage this internship alongside my academic responsibilities. This experience has significantly improved my understanding of circuit design and simulation, and I am proud to have contributed to the growing eSim ecosystem.

# Contents

# Chapter 1

# Introduction

The electronics industry fundamentally depends on the design, simulation, and verification of circuits prior to their physical realization. Electronic Design Automation (EDA) tools are indispensable in ensuring design accuracy, efficiency, and reliability. Among these, eSim stands out as a robust open-source EDA tool developed by the FOSSEE (Free/Libre and Open Source Software for Education) project at IIT Bombay. eSim integrates KiCad for schematic capture and PCB design with Ngspice for simulation, offering a comprehensive platform for circuit design and analysis.

The FOSSEE initiative aims to enhance the quality of education by promoting the adoption of open-source software tools, thereby reducing dependence on proprietary software in academic institutions. By developing and disseminating tools like eSim, FOSSEE empowers students and educators to engage with advanced technological resources without the barriers of licensing costs.

eSim is constructed by integrating several open-source tools:

- **KiCad:** Facilitates schematic drawing, layout design, and component management.
- **Ngspice:** A circuit simulator capable of performing DC, transient, and AC analyses.
- **KiCad to Ngspice Converter:** Bridges the design from KiCad to simulation by setting up analysis parameters and managing device models.
- **Model Builder:** Supports the creation of new device models for components like diodes, BJTs, MOSFETs, JFETs, IGBTs, and magnetic cores.
- **Subcircuit Builder:** Enables the development of reusable subcircuits applicable across various projects.
- **NGHDL:** Facilitates mixed-signal simulation using VHDL.
- **NgVeri:** Enables simulation using Verilog/System Verilog.

This internship was undertaken with the objective of gaining hands-on experience in circuit simulation using eSim. Throughout the internship, I explored various features of the software, including schematic creation, SPICE simulation, netlist generation, and waveform analysis. The focus was on understanding the workflow of circuit design using open-source tools and applying this knowledge to practical projects and exercises.

A significant aspect of the internship involved addressing practical challenges to enhance eSim's functionality. One such challenge was the suppression of default Ngspice pop-up plots to maintain a seamless user interface within eSim. Additionally, I implemented multithreading capabilities to prevent the application from becoming unresponsive during simulations. Furthermore, I restructured the execution of Ngspice by enabling it to run in the background, complemented by a user interface designed to monitor its status directly within eSim.

This report outlines the activities performed during the internship, key learnings, challenges encountered, and the skills acquired. It highlights the importance of adopting open-source alternatives in the field of electronics and reflects on how eSim contributes to self-reliant, cost-effective design practices in both academic and professional domains.

# Chapter 2

# Problem Statement

the growing adoption of open-source tools in electronics design, many of these tools still face challenges in matching the functionality, performance, and user experience offered by commercial Electronic Design Automation (EDA) software. eSim, developed under the FOSSEE project at IIT Bombay, serves as a promising alternative by integrating schematic design and circuit simulation capabilities into a single open-source platform. However, to be a truly viable and scalable educational and design tool, eSim must continuously evolve in terms of performance, usability, and integration.

One of the key limitations observed in eSim is the lack of seamless interaction between its integrated modules, especially during the simulation phase involving Ngspice. This results in redundant user interfaces, application unresponsiveness, and non-intuitive simulation workflows, which can be challenging for students and first-time users. Furthermore, certain aspects of eSim's execution, such as external terminal dependency and UI blocking during simulations, hinder its efficiency and practical usability in academic environments.

This internship was undertaken to identify and address such functional gaps in eSim, with the broader aim of enhancing its reliability, interactivity, and ease of use. The core problem addressed is not just the technical glitches, but the larger need for improving user experience, system responsiveness, and simulation handling—all while preserving the philosophy of open-source education and accessibility. The project aims to contribute to the long-term goal of making eSim a truly competitive and sustainable tool for electronics education and design.

# Chapter 3

# Workflow

## 3.1 Approach

- The approach followed during this internship was systematic and aligned with the goal of contributing meaningful improvements to the eSim platform while gaining practical exposure to circuit simulation and model development.
- The first step involved **selecting analog ICs** that were not available in the default eSim library. After finalizing the ICs, their **datasheets were thoroughly studied** to understand internal schematics, pin configurations, electrical characteristics, and expected behaviors under different conditions.
- Using this reference information, the **subcircuit models were developed** in eSim. This required replicating the internal schematic of the IC in eSim's Subcircuit Builder module using Eeschema. Care was taken to ensure that port placements, connections, and component values matched the datasheet specifications. Once the schematic was ready, **annotation and netlist generation** were carried out to prepare the circuit for simulation.
- Following this, the subcircuit design was converted using the **KiCad to Ngspice converter**. Device models were added as needed to reflect the correct behavior during simulation. To facilitate reuse and testing, custom **symbols were created** using the Symbol Editor in eSim.
- Each subcircuit was then integrated into a **test circuit**, based on typical applications or test configurations provided in the IC datasheet. These circuits were simulated, and the outputs were verified by comparing them with the expected results mentioned in the datasheet. In cases where the output deviated significantly, circuit parameters were adjusted, or debugging was performed to refine the model.
- In addition to circuit modeling, specific improvements were made to the **eSim simulation workflow**—such as suppressing Ngspice's default plot pop-ups, introducing multithreading to avoid UI freezing, and embedding a custom simulation status viewer into the GUI. These steps were essential to enhance

the overall user experience while ensuring that simulations remained accurate and efficient.

- Through this structured approach—starting from component selection to validation and tool enhancement—the internship not only contributed to the expansion of the eSim library but also addressed practical limitations in its simulation interface.

## 3.2 Execution

- The execution of the internship tasks involved a step-by-step workflow using the various modules and features offered by eSim. Initially, the software was installed, and a dedicated workspace was set up to manage all design files and subcircuits.

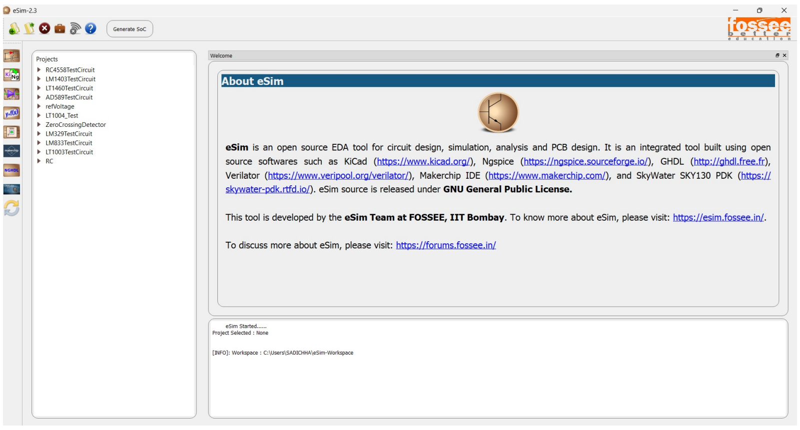


Figure 3.1: eSim workspace setup

- For each selected IC, a new subcircuit schematic was created using **Eeschema**. The internal schematic was carefully constructed based on the IC's datasheet, and ports were positioned to match input/output pins.

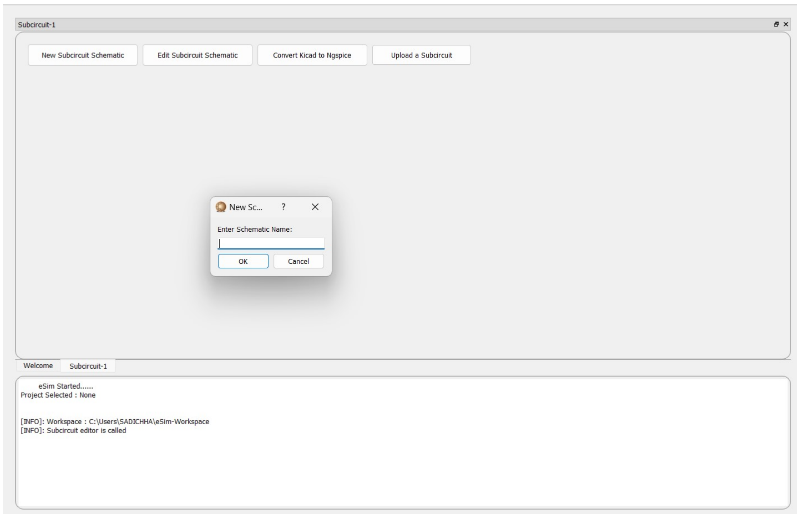


Figure 3.2: Creating Subcircuit for IC

- Once the schematic was finalized, annotation and netlist generation were performed to prepare it for simulation.

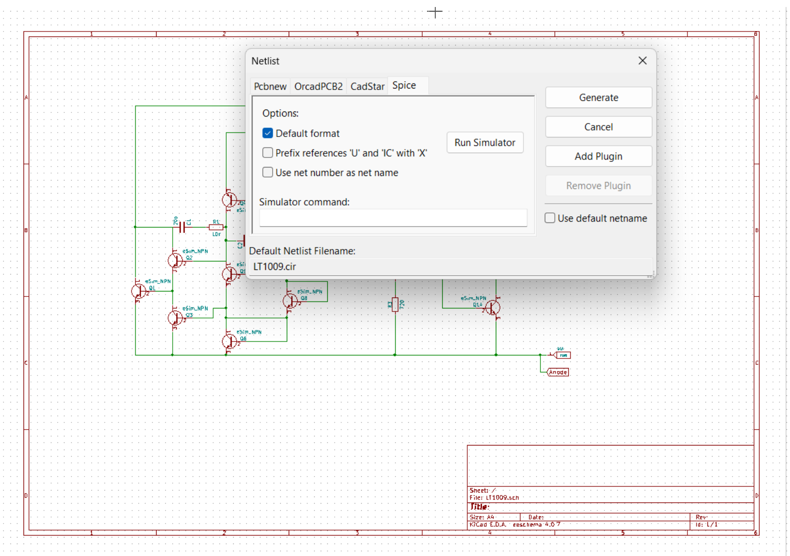


Figure 3.3: Netlist generation process in eSim

- The schematic was then passed through the **KiCad to Ngspice converter**, where appropriate device models were linked from the built-in model library. Simulation parameters were also defined during this step.

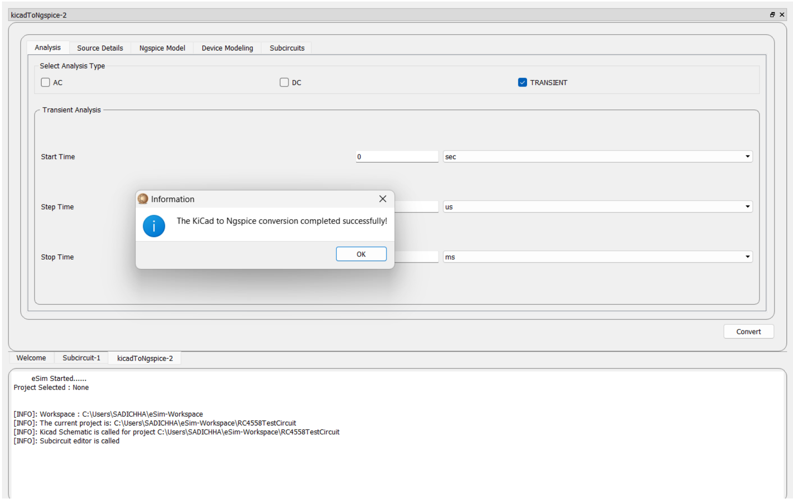


Figure 3.4: KiCad to Ngspice converter interface

- A custom symbol was created using the **Symbol Editor** in eSim, enabling the subcircuit to be reused in multiple test circuits and designs.

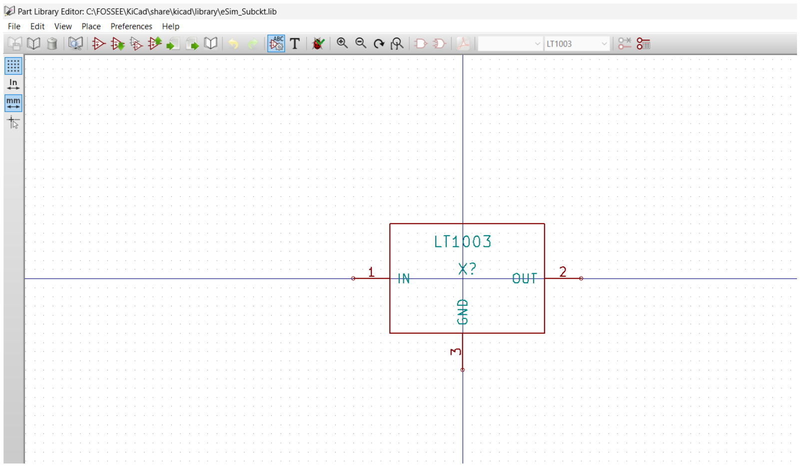


Figure 3.5: Custom symbol creation in Symbol Editor

- A complete test circuit was built by placing the newly created symbol, and simulations were carried out. The output waveforms were observed and compared with expected results from the datasheet. Necessary tweaks were made to improve accuracy.

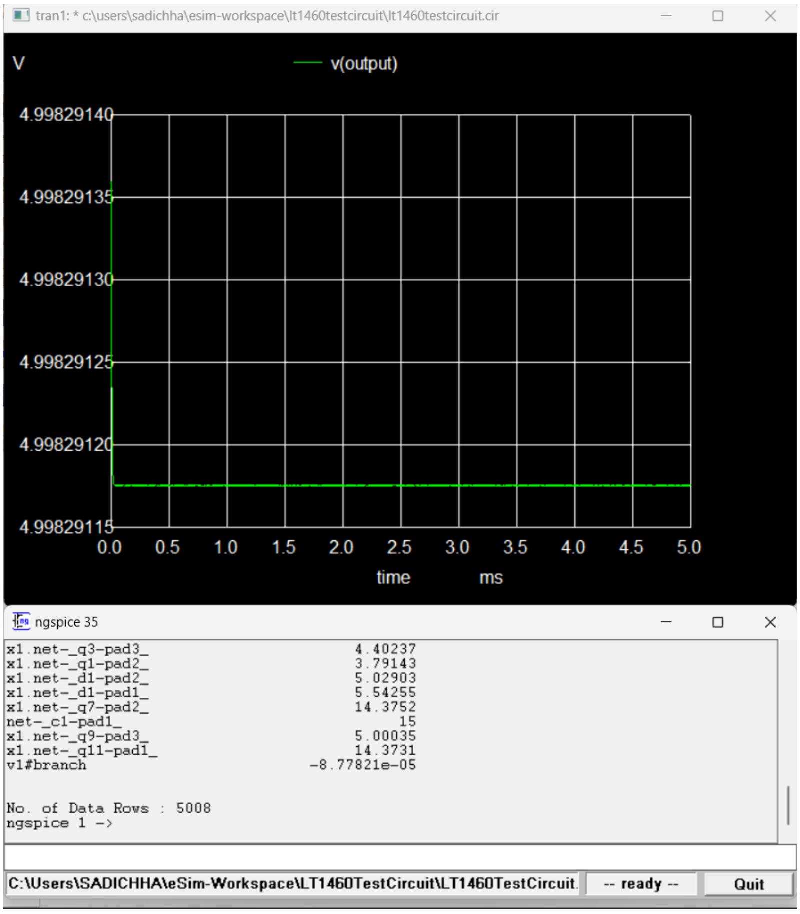


Figure 3.6: output from simulation in eSim

Overall, the execution phase involved both IC-level subcircuit development and higher-level system enhancements to improve the usability and efficiency of eSim.

# Chapter 4

# Designed Circuits

This section presents all the integrated circuits (ICs) designed and simulated during the internship using eSim. Each IC includes its type, pin configuration, key features, subcircuit design, test setup, and output waveform.

## 4.1 LT1009

- **Type of IC:** Precision 2.5V Shunt Voltage Reference
- **Pin Count:** 3 pins

### 4.1.1 Description:

The LT1009 is a precision-trimmed 2.5V shunt voltage reference designed for applications requiring stable and accurate voltage regulation. It offers high initial accuracy (±5mV) through on-chip trimming and maintains excellent temperature stability. An adjustment terminal allows fine-tuning of the output voltage by ±5%, making it highly adaptable for precision analog systems. It is commonly used in ADC/DAC references, digital voltmeters, power monitors, and control systems.

### 4.1.2 Key Features:

- 5V initial accuracy via on-chip trimming
- Wide operating current Range
- Low dynamic impedance (0.6Ω)
- Initial Tolerance:(0.2%)
- Directly Interchangeable with LM136

### 4.1.3 Subcircuit:

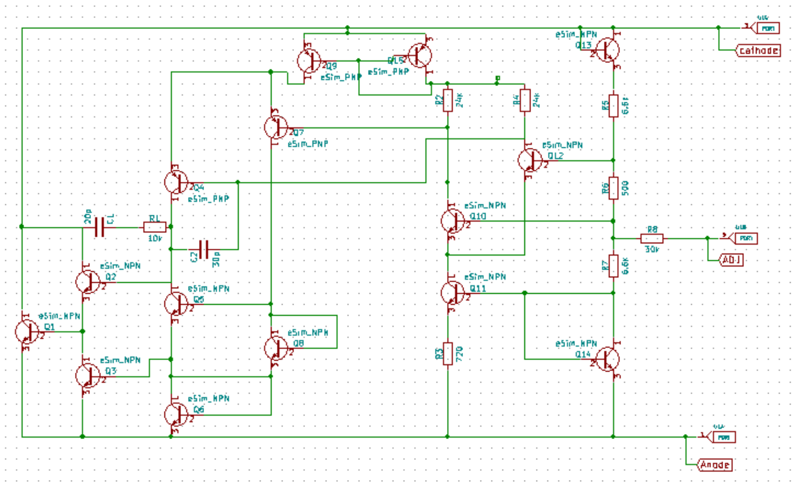

Figure 4.1: LT1009 Subcircuit in Eeschema

### 4.1.4 Test Circuit:

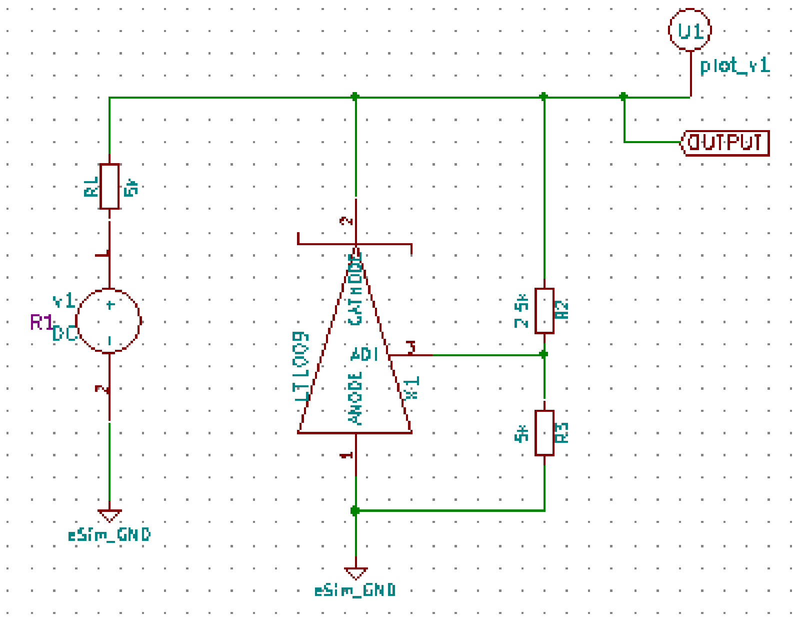


Figure 4.2: LT1009 Test Circuit

### 4.1.5 Output:

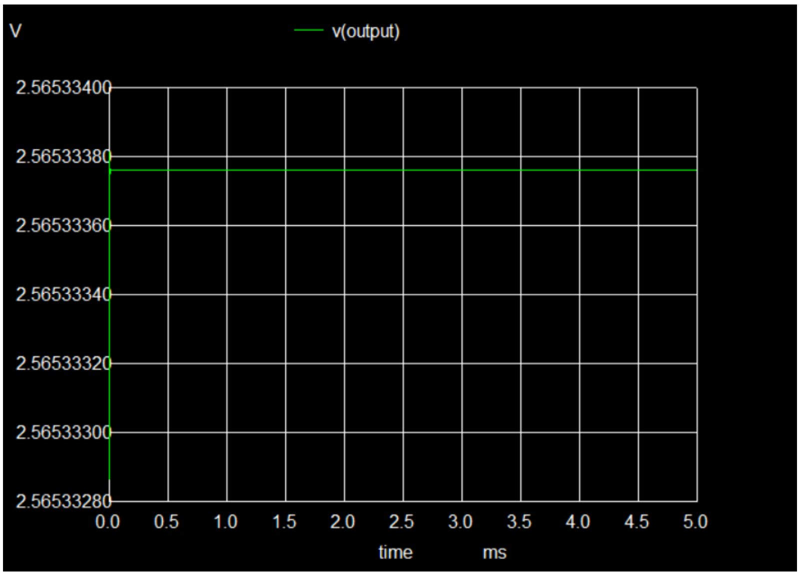


Figure 4.3: LT1009 Simulation Output

## 4.2 LT1004

- **Type of IC:** Micropower Voltage Reference
- **Pin Count:** 2 pins

### 4.2.1 Description:

The LT1004 micropower voltage reference is a two-terminal band-gap reference diode designed to provide high accuracy and excellent temperature characteristics at very low operating currents. Optimizing the key parameters in the design, processing, and testing of the device results in specifications previously attainable only with selected units.The LT1004 is a pin-for-pin replacement for the LM285 and LM385 series of references, with improved specifications. It is an excellent device for use in systems in which accuracy previously was attained at the expense of power consumption and trimming.

### 4.2.2 Key Features:

- Low operating current
- Excellent temperature stability
- Available in 1.2V and 2.5V versions
- Pin-compatible with LM285/LM385

### 4.2.3 Subcircuit:

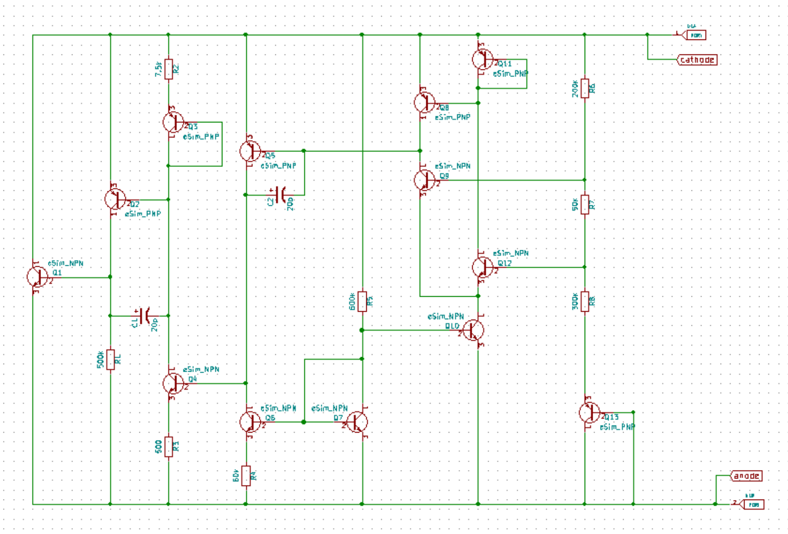

Figure 4.4: LT1004 Subcircuit in Eeschema

### 4.2.4 Test Circuit:

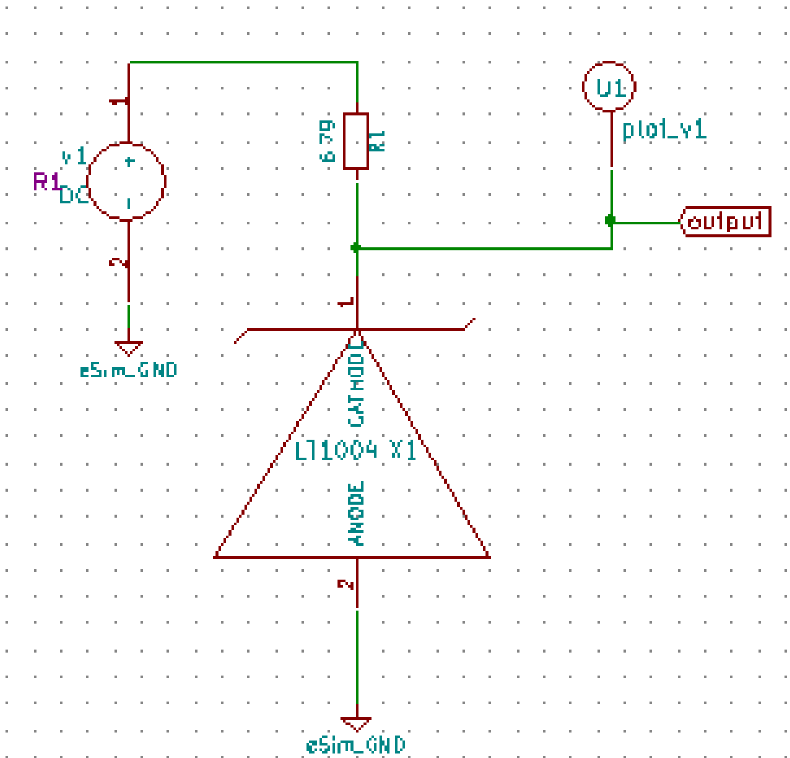


Figure 4.5: LT1004 Test Circuit

### 4.2.5 Output:

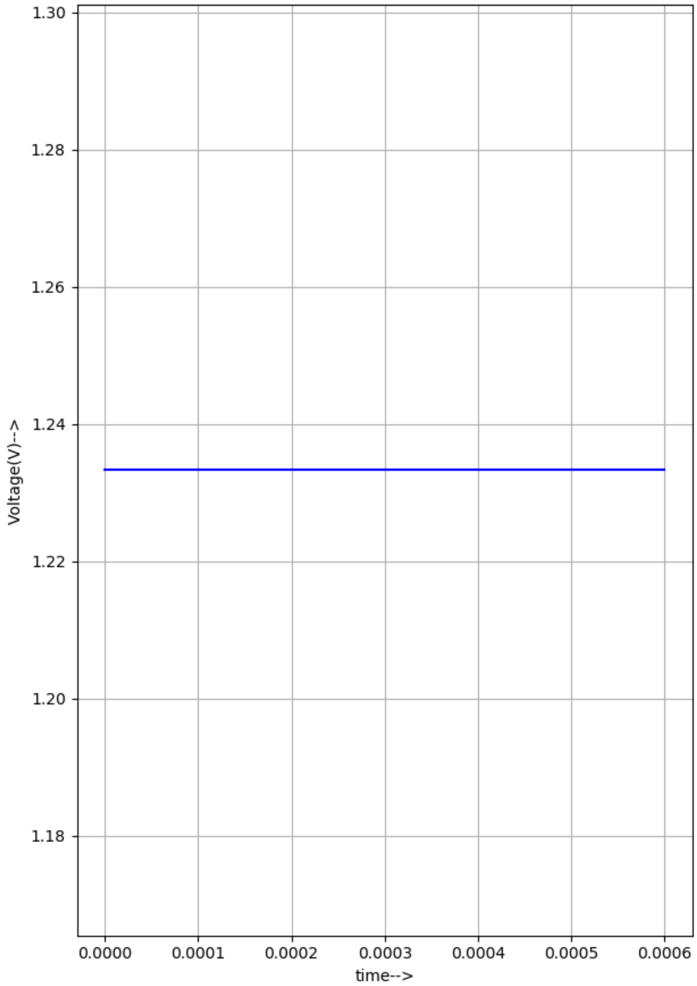


Figure 4.6: LT1004 Simulation Output

## 4.3 LM311-N

- **Type of IC:** Differential Comparator with Strobe
- **Pin Count:** 8 pins

### 4.3.1 Description

The LM311 is a high-speed voltage comparator featuring a wide operating supply range and open-collector output. Its fast switching time and low offset make it suitable for zero crossing detectors, oscillators, and other comparator-based circuits. Their output is compatible with RTL, DTL and TTL as well as MOS circuits. Further, they can drive lamps or relays, switching voltages up to 50V at currents as high as 50 mA.

### 4.3.2 Key Features:

- Operates From Single 5V Supply
- Fast response time (165 ns)
- Low input bias current
- Offset Current: 20 nA Max. Over Temperature
- Power Consumption: 135 mW at ±15V

### 4.3.3 Subcircuit:

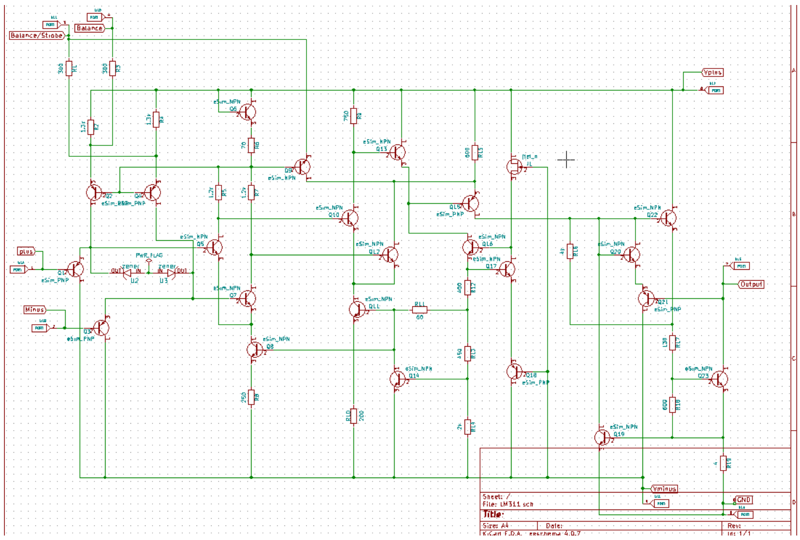

Figure 4.7: LM311 Subcircuit in Eeschema

### 4.3.4 Test Circuit:

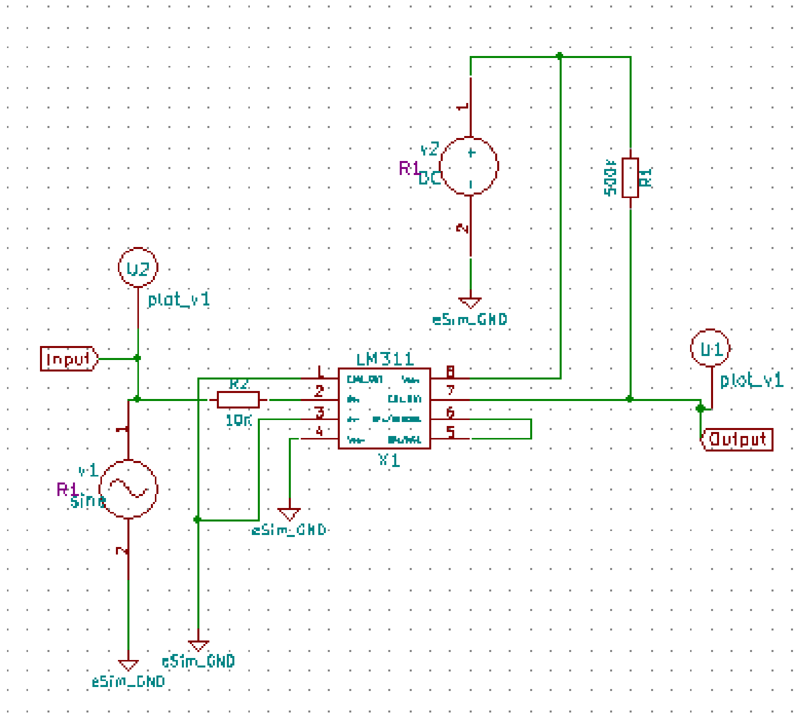


Figure 4.8: LM311 Test Circuit

### 4.3.5 Output:

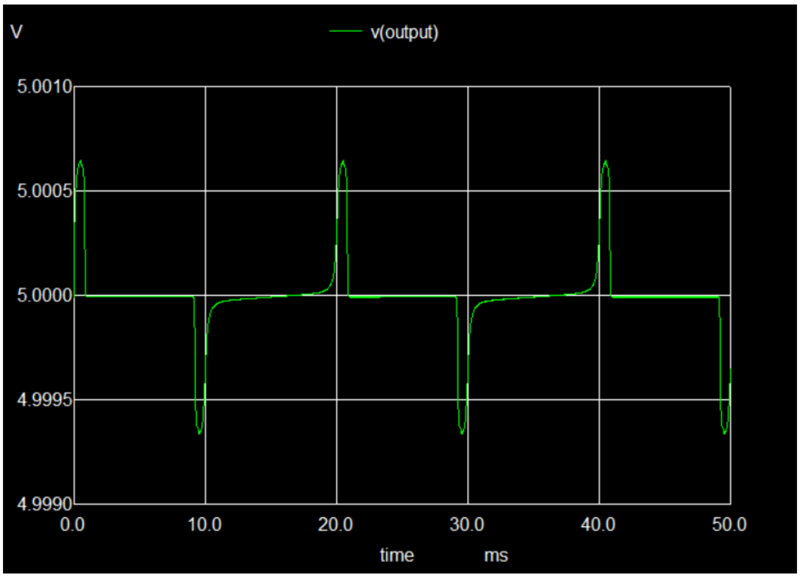


Figure 4.9: LM311 Simulation Output

## 4.4 LM329

- **Type of IC:** 6.9V Precision Voltage Reference

- **Pin Count:** 2 pins

### 4.4.1 Description:

The LM329 is precision multi-current temperature-compensated 6.9V zener refrence with dynamic impedence a factor of 10 to 100 less than discrete diodes. Constructed in a singele silicon chip, the LM329 uses active circuitry to buffer the internal zener alloing the device to operate over a 0.5mA to 15mA range with virtually no chnage in performance.

### 4.4.2 Features:

- 0.6 mA to 15 mA Operating Current
- 0.8 Dynamic Impedance at Any Current
- Wide operating current range
- 7V Wideband Noise
- 0.002% Long Term Stability
- Subsurface Zener

### 4.4.3 Subcircuit:

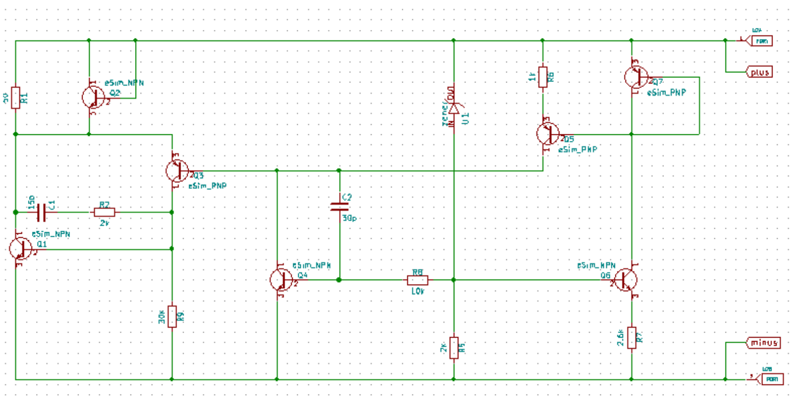

Figure 4.10: LM329 Subcircuit in Eeschema

### 4.4.4 Test Circuit:

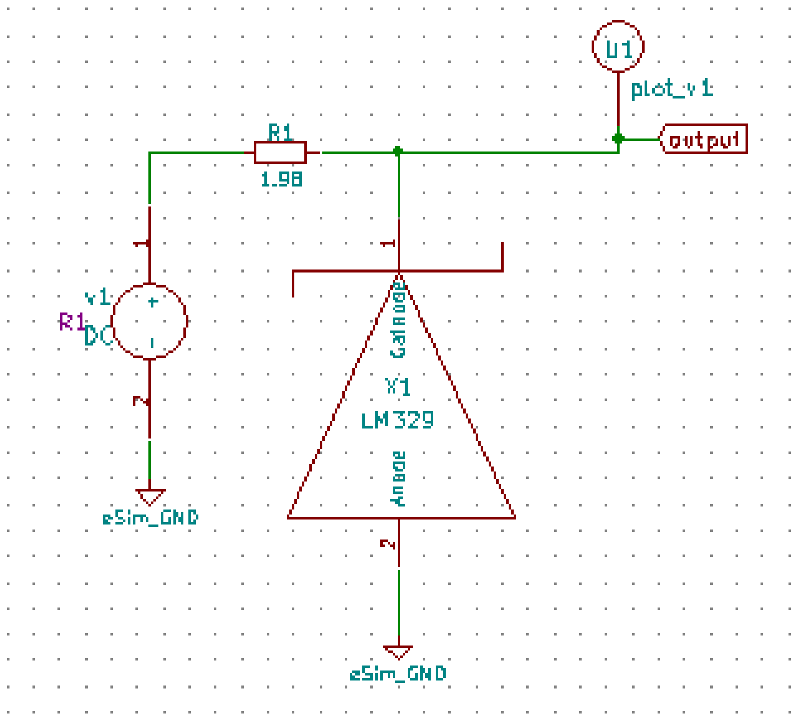


Figure 4.11: LM329 Test Circuit

### 4.4.5 Output:

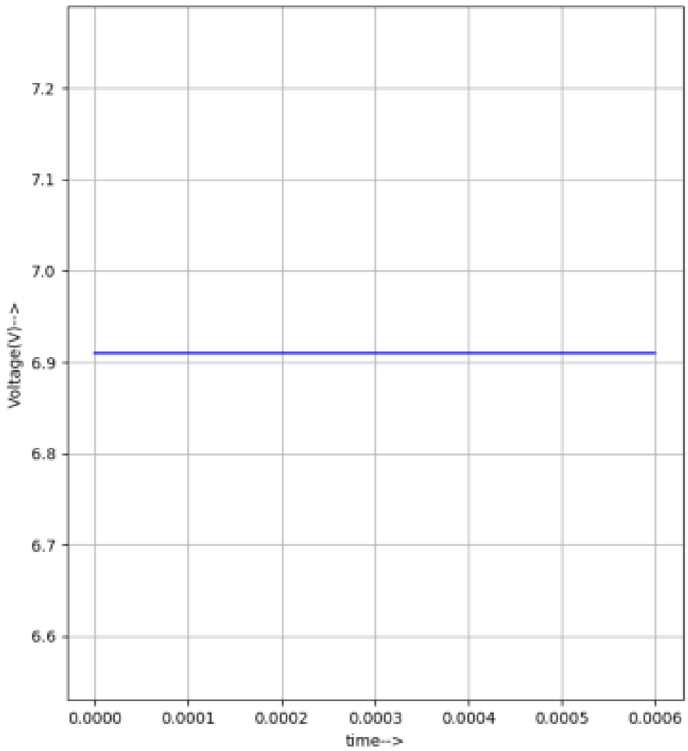


Figure 4.12: LM329 Simulation Output

## 4.5 LM833-N

- **Type of IC:** Dual High-Speed Audio Operational Amplifier
- **Pin Count:** 8 pins

### 4.5.1 Description

The LM833-N is a dual low-noise operational amplifier tailored for high-fidelity audio applications. With low distortion, high slew rate, and a wide bandwidth, it performs well in active filters, preamplifiers, and tone controls.The LM833-N is pin-for-pin compatible with industry standar dual operational amplifier.

### 4.5.2 Key Features:

- Low input noise voltage
- Low total harmonic distortion

- High slew rate (7 V/$\mu$s)
- Wide Power Bandwidth: 120KHz
- Low Offset Voltage: 0.3mV
- Large Phase Margin: 60°

### 4.5.3 Subcircuit:

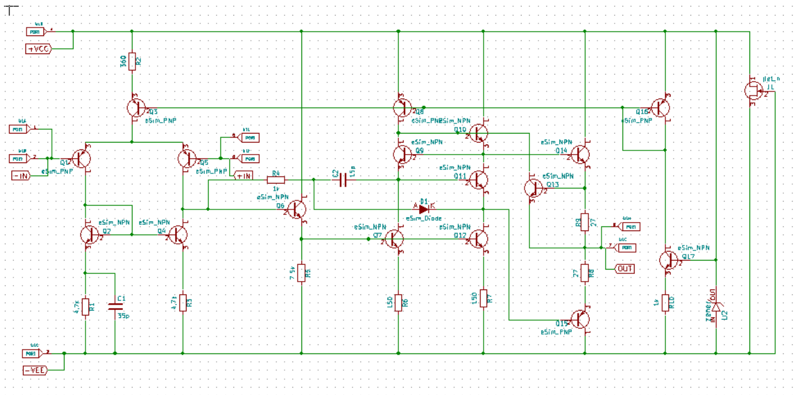

Figure 4.13: LM833 Subcircuit in Eeschema

### 4.5.4 Test Circuit:

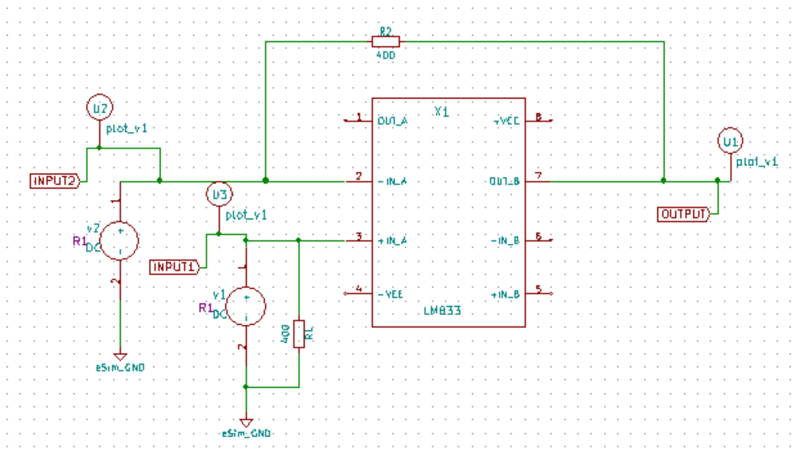


Figure 4.14: LM833 Test Circuit

### 4.5.5 Output:

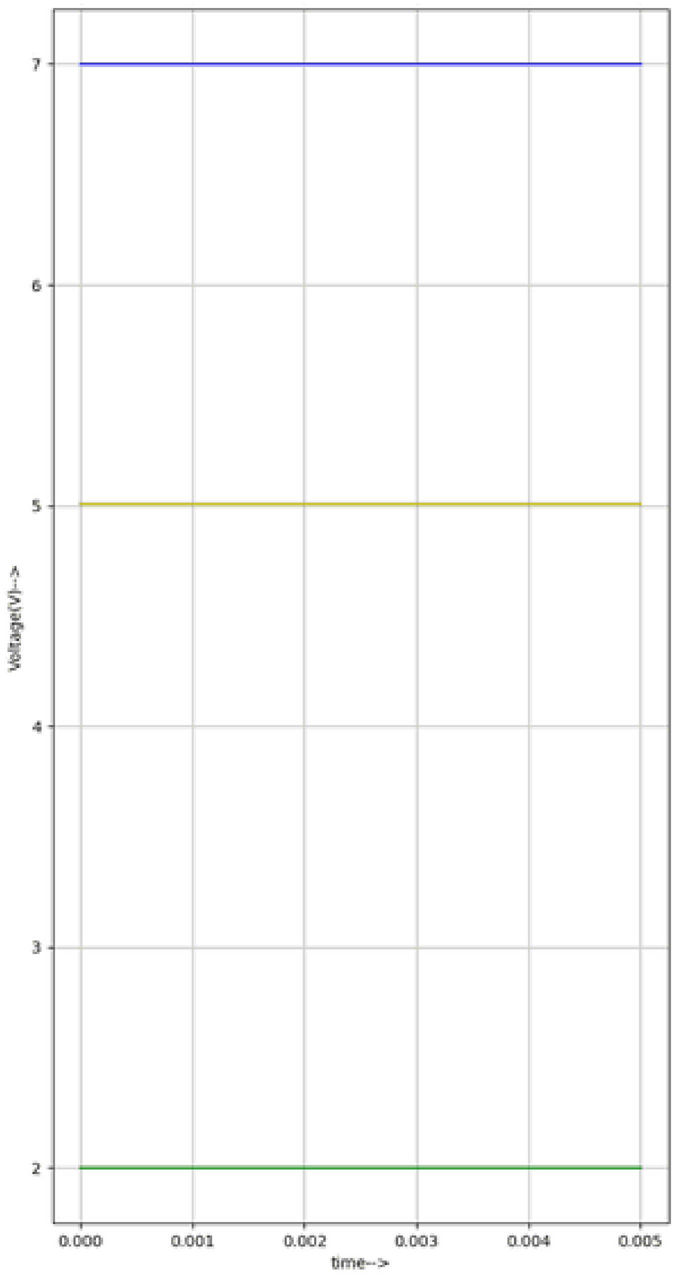


Figure 4.15: LM833 Simulation Output

## 4.6 LT1003

- **Type of IC:** 5V, 5A Voltage Regulator
- **Pin Count:** 3 pins

### 4.6.1 Description

The LT1003 is a 5V voltage regulator capable of delivering up to 5A output current. Though obsolete, it was widely used in power regulation tasks requiring high current and accurate voltage stability.The LT1003 is Linear Technology's advanced design, process and test techniques for improved quality and reliability over similar device types.

### 4.6.2 Key Features:

- High current output (5A)
- Good line regulation
- Improved initial accuracy
- Obsolete, replaced in modern designs
- 40 Watt Capability

### 4.6.3 Subcircuit:

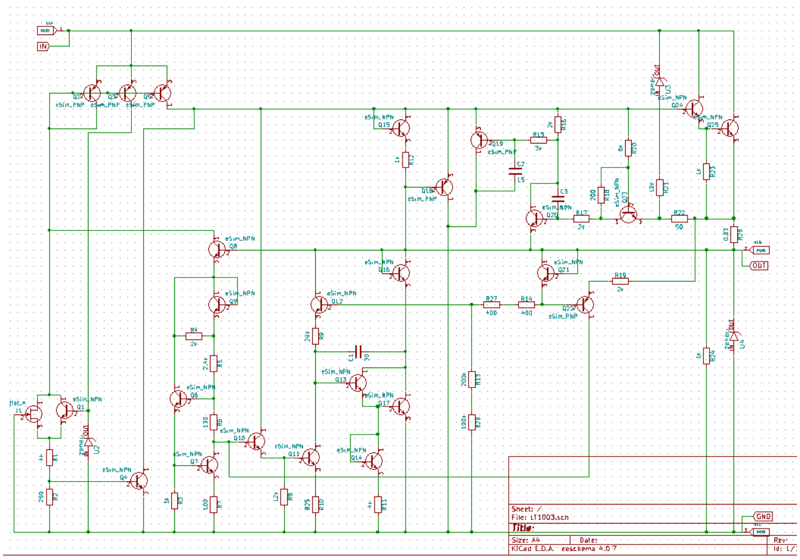

Figure 4.16: LT1003 Subcircuit in Eeschema

### 4.6.4 Test Circuit:

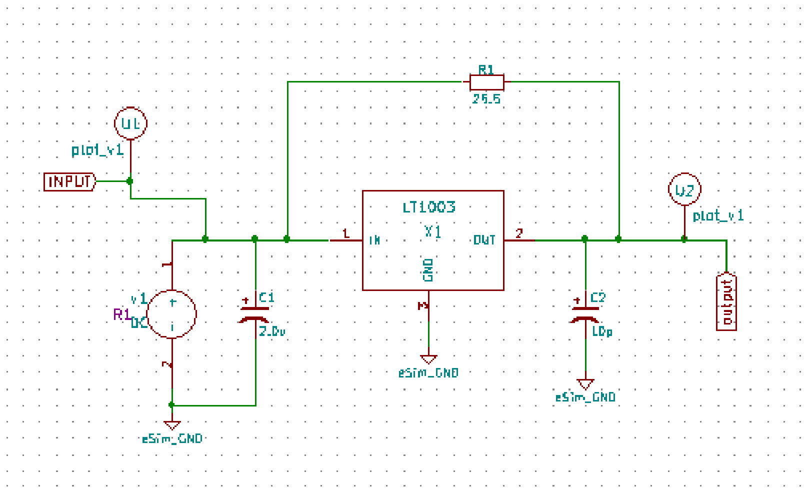

Figure 4.17: LT1003 Test Circuit

### 4.6.5 Output:

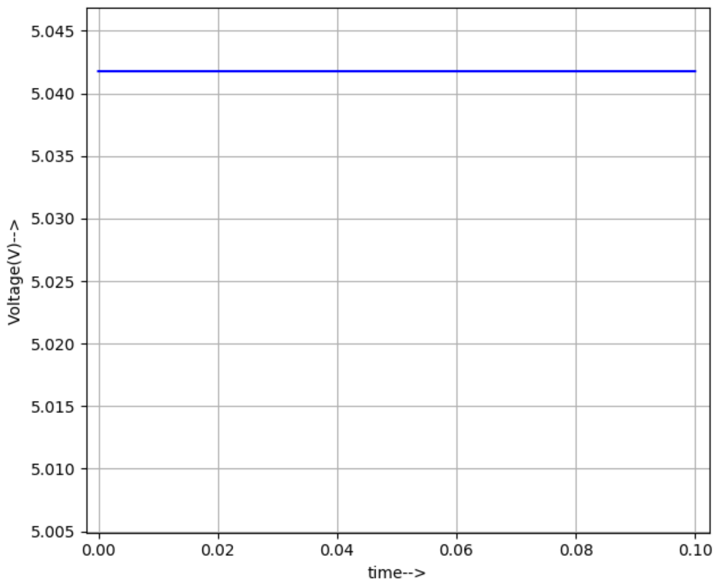


Figure 4.18: LT1003 Simulation Output

## 4.7 LT1460

- **Type of IC:** Micropower Precision Series Voltage Reference
- **Pin Count:** 3 pins

### 4.7.1 Description

The LT1460-5 is a micropower bandgap reference that combines very high accuracy and low drift with low power dissipation and small package size. This series reference uses curvature compensation to obtain a low temperature coefficient and trimmed precision thin-film resistors to achieve high output accuracy. The reference will supply up to 20mA, making it ideal for precision regulator applications, yet it is almost totally immune to input voltage variations.Reverse battery protection keeps the reference from conducting current and being damaged.

### 4.7.2 Key Features:

- Low power consumption
- High Accuracy: 0.075% Max
- Low Supply Current: 175mA Max
- Temperature Coefficient Guaranteed to 125°C
- Minimum Output Current: 20mA
- No Output Capacitor Required

### 4.7.3 Subcircuit:

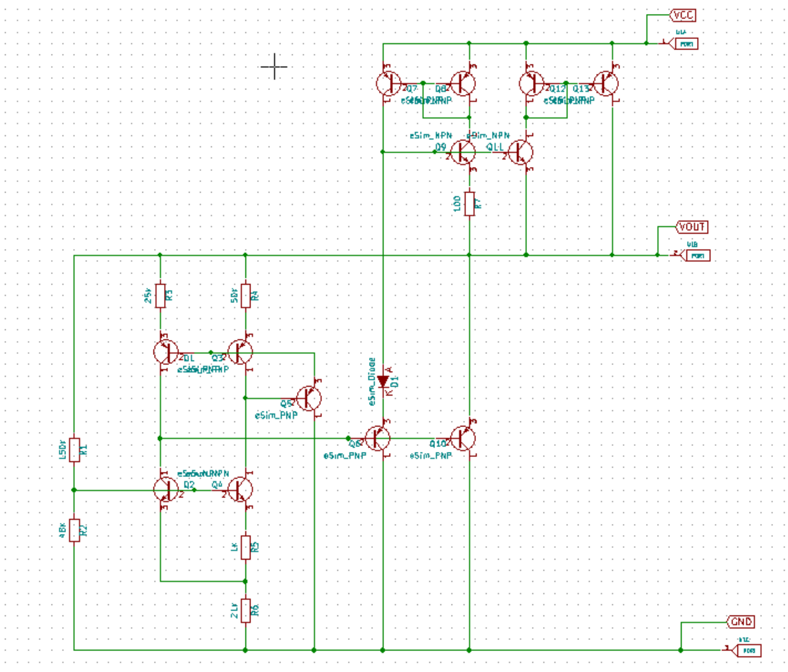


Figure 4.19: LT1460 Subcircuit in Eeschema

### 4.7.4 Test Circuit:

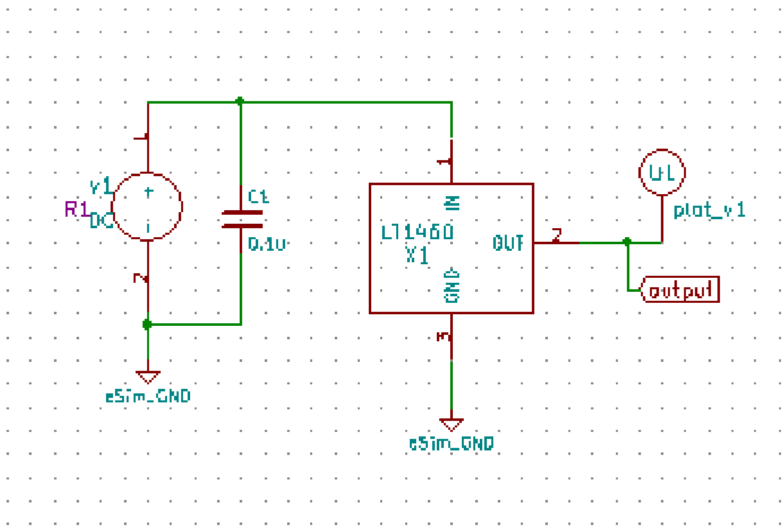


Figure 4.20: LT1460 Test Circuit

### 4.7.5 Output:

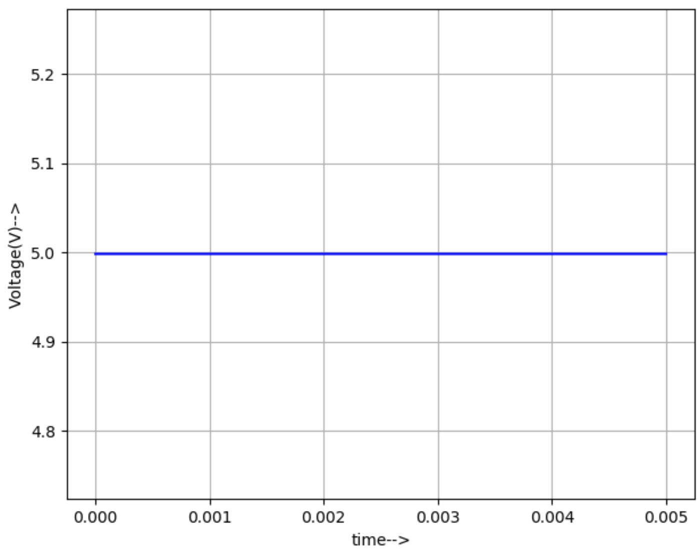


Figure 4.21: LT1460 Simulation Output

## 4.8 LM1403

- **Type of IC:** Precision 2.50V Voltage Reference
- **Pin Count:** 3 pins

### 4.8.1 Description

The LM1403 is a precision, monolithic, temperature-compensated voltage reference. The LM1403 makes use of thin-film technology enhanced by the discrete laser trimming of resistors to achieve excellent Temperature coefficient. The trim scheme is such that individual resistors are cut open rather than being trimmed (partially cut), to avoid resistor drift caused by electromigration in the trimmed area. The LM1403 also provides excellent stability vs. changes in input voltage and output current. The output is current-limited and is short circuit proof.

### 4.8.2 Key Features:

- Low 400 mA operating current
- Temperature coefficient as low as 11 ppm/°C
- Low output impedance
- Single-supply operation

### 4.8.3 Subcircuit:

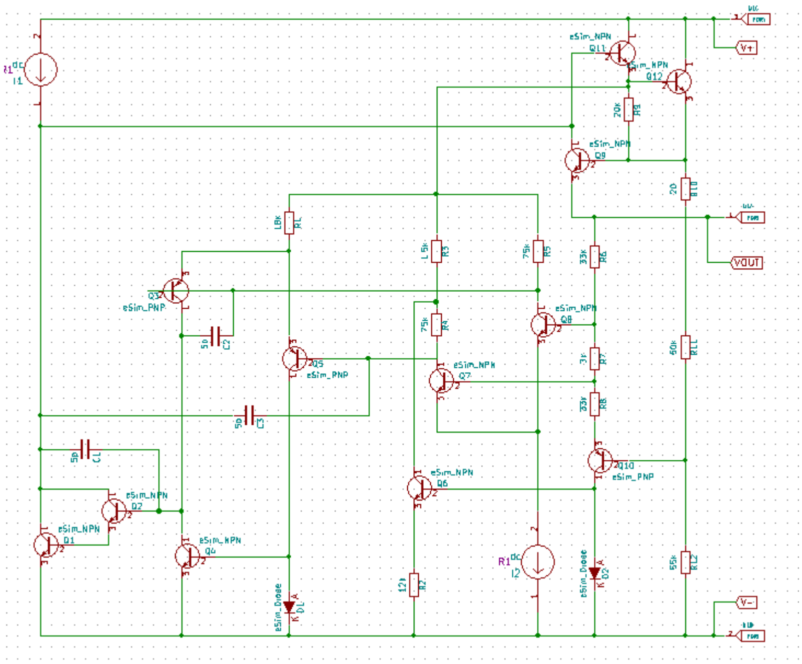


Figure 4.22: LM1403 Subcircuit in Eeschema

### 4.8.4 Test Circuit:

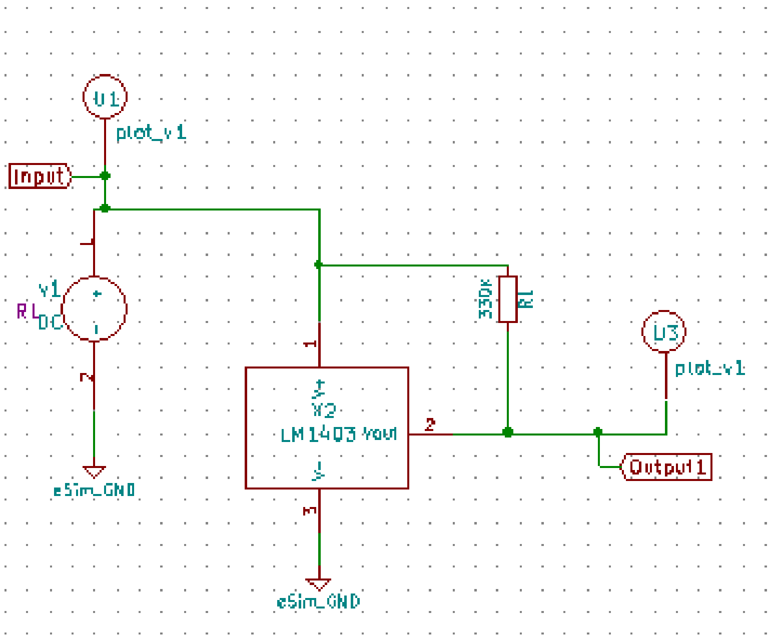


Figure 4.23: LM1403 Test Circuit

### 4.8.5 Output:

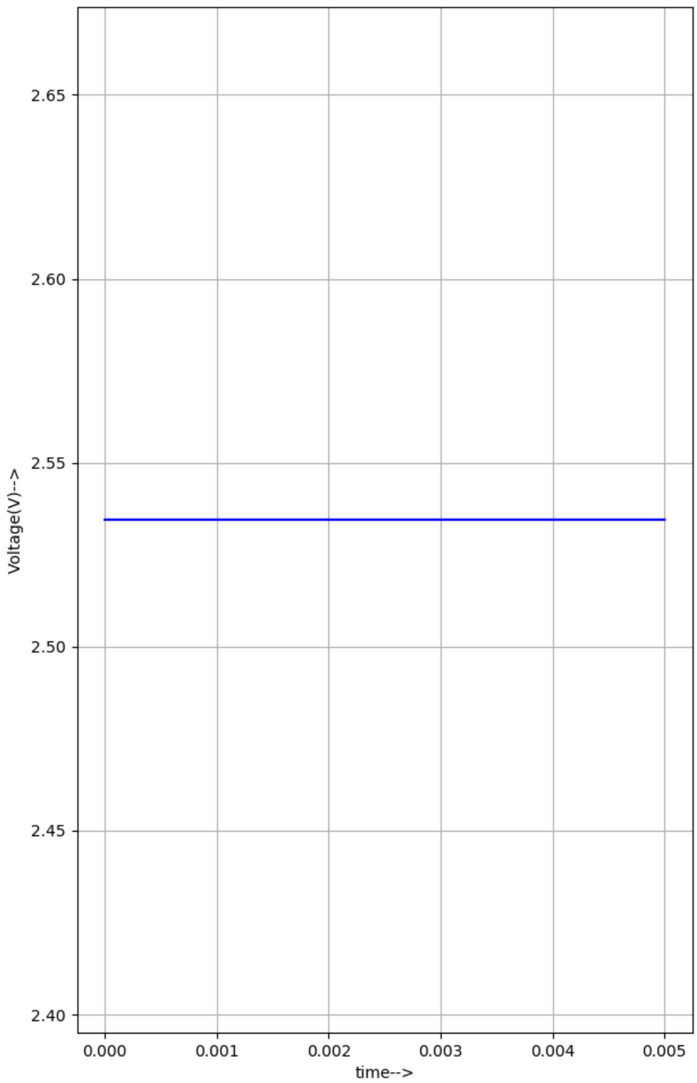


Figure 4.24: LM1403 Simulation Output

# Chapter 5

# Conclusion and Future Scope

## 5.1 Conclusion

The internship offered valuable hands-on experience with open-source electronic design automation using the eSim platform developed by FOSSEE, IIT Bombay. Through this project, multiple analog ICs were successfully modeled, verified, and integrated into test circuits using a structured simulation workflow. The internship enhanced my understanding of SPICE-based simulation, subcircuit modeling, schematic creation, and waveform analysis.

Furthermore, improvements such as pop-up suppression, multithreading support, and background Ngspice execution contributed to a smoother user experience within eSim. These tasks not only reinforced core electronics concepts but also deepened my confidence in working with real-world design tools and collaborative open-source environments.

## 5.2 Future Scope

While significant progress was made, there are several opportunities to expand and refine the work carried out during this internship:

- **Expansion of IC Library:** Additional analog, digital, and mixed-signal ICs can be modeled and validated to enrich eSim's component library.
- **Automated Model Verification:** Tools can be developed to automate the validation of subcircuit behavior against datasheet specifications.
- **Improved UI/UX Integration:** Further enhancement of user feedback mechanisms during simulation—such as real-time progress indicators and simulation logs—can be explored.
- **Community Collaboration:** Documenting the modeling process and contributing models to the open-source repository can help future users adopt and improve eSim more easily.

- **Digital IC Modeling:** Integrating Verilog or VHDL-based digital IC modeling using NGHDL or NgVeri within eSim can broaden its scope for academic and project-based use.

Overall, this internship serves as a strong foundation for continued contributions to open-source EDA tools and for applying these skills in academic research or industry-oriented electronic design projects.

# Bibliography


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

[2] FOSSEE Spoken tutorial.
URL:https://spoken-tutorial.org/tutorial%02search/?search_foss=eSim&search_language=English

[3] LT1009
URL:https://www.ti.com/lit/ds/symlink/lt1009.pdf?ts=1748874104871

[4] LT1004
URL:https://www.ti.com/lit/ds/symlink/lt1004-1.2.pdf

[5] LM311-N
URL:https://www.ti.com/lit/ds/symlink/lm311.pdf

[6] LM329
URL:https://www.analog.com/media/en/technical-documentation/data-sheets/129329fd.pdf

[7] LM833-N
URL:https://www.ti.com/lit/ds/symlink/lm833.pdf

[8] LT1003
URL:https://www.alldatasheet.com/datasheet-pdf/view/70238/LINER/LT1003.html

[9] LT1460
URL:https://www.alldatasheet.com/datasheet-pdf/view/70449/LINER/LT1460-5.html

[10] LM1403
URL:https://www.alldatasheet.com/datasheet-pdf/view/194463/NSC/LM1403.html