Summer Fellowship Report

On

IC Design Using Subcircuit in eSim

Submitted by

Krishan Kumar

Under the guidance of

Prof. Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

June 19, 2024

Contents

1 Introduction **4**
- 1.1 **eSim** . . . 4
  - 1.1.1 KiCad . . . 4
  - 1.1.2 Ngspice . . . 4
  - 1.1.3 KiCad to Ngspice converter . . . 4
  - 1.1.4 Model Builder . . . 5
  - 1.1.5 Subcircuit Builder . . . 5
  - 1.1.6 NGHDL . . . 5
  - 1.1.7 NgVeri . . . 5
  - 1.1.8 Makerchip . . . 5
  - 1.1.9 Verilator . . . 5
  - 1.1.10 SandPiper SaaS . . . 5

2 Abstract **6**
- 2.1 Approach . . . 6
- 2.2 Execution . . . 7

3 Integrated Circuit Design **9**
- 3.1 **MC1709** . . . 9
  - 3.1.1 Description . . . 9
  - 3.1.2 Pin Diagram . . . 9
  - 3.1.3 Schematic Diagram . . . 10
  - 3.1.4 Simulation Circuit . . . 11
  - 3.1.5 NgSpice Plot . . . 12
- 3.2 **MC1741C** . . . 13
  - 3.2.1 Description . . . 13
  - 3.2.2 Pin Diagram . . . 13
  - 3.2.3 Schematic Diagram . . . 14
  - 3.2.4 Simulation Circuit . . . 15
  - 3.2.5 NgSpice Plot . . . 16
- 3.3 **TL431C** . . . 17
  - 3.3.1 Description . . . 17
  - 3.3.2 Pin Diagram . . . 17
  - 3.3.3 Schematic Diagram . . . 18
  - 3.3.4 Simulation Circuit . . . 19
  - 3.3.5 NgSpice Plot . . . 20

3.4 **LM431** . . . 21
3.4.1 Description . . . 21
3.4.2 Pin Diagram . . . 21
3.4.3 Schematic Diagram . . . 22
3.4.4 Simulation Circuit . . . 23
3.4.5 Python Plot . . . 24
3.5 **LH0003** . . . 25
3.5.1 Description . . . 25
3.5.2 Pin Diagram . . . 25
3.5.3 Schematic Diagram . . . 26
3.5.4 Simulation Circuit . . . 27
3.5.5 Python Plot . . . 28
3.6 **LM3080** . . . 29
3.6.1 Description . . . 29
3.6.2 Pin Diagram . . . 29
3.6.3 Schematic Diagram . . . 30
3.6.4 Simulation Circuit . . . 31
3.6.5 Python Plot . . . 32
3.7 **LM709** . . . 33
3.7.1 Description . . . 33
3.7.2 Pin Diagram . . . 33
3.7.3 Schematic Diagram . . . 34
3.7.4 Simulation Circuit . . . 35
3.7.5 Python Plot . . . 36
3.8 **LM725** . . . 37
3.8.1 Description . . . 37
3.8.2 Pin Diagram . . . 37
3.8.3 Schematic Diagram . . . 38
3.8.4 Simulation Circuit . . . 39
3.8.5 Python Plot . . . 40

4 Conclusion and Future Scope **41**

5 Bibliography **42**

Acknowledgment

I would like to extend my heartfelt gratitude to the FOSSEE team for this incredible opportunity. I am extremely thankful to Prof. Kannan M. Moudgalya for guiding and motivating us throughout the FOSSEE fellowship program. Initially, I was uncertain about the internship as I am only in my second year, in the 3rd semester, and lacked confidence in using open-source electronics software. However, eSim provided me with the knowledge, environment, and learning capability to work on the project individually.

I would like to extend my heartfelt gratitude to Mr. Sumanto Kar for being a constant mentor and support throughout my journey. You have helped me not only with eSim subcircuits but also in enhancing my knowledge of studying datasheets, reading research papers, navigating GitHub, and using open-source platforms. Your weekly monitoring of our work, debugging of our errors, and the invaluable insights you provided to optimize our designs have been incredibly beneficial. Thank you for your dedication and guidance. I will always remember our fellow colleagues for their invaluable assistance, exceptional knowledge, and superb skills. The work environment has been incredibly energetic and friendly, fostering collaboration and growth.

Over the past three months, I have learned a great deal through the eSim program. Balancing this online fellowship with my semester coursework has not only enhanced my core skills and proficiency with open-source tools but also improved my time management, discipline, and sense of responsibility. I am eager to apply my knowledge and skills for the betterment of society, contributing to both health and wealth. I am proud to be a part of the esteemed IIT Bombay, FOSSEE-eSim Team.

In closing, I would like to express my profound gratitude to everyone who has supported and guided me throughout this journey. This experience has been immensely enriching and transformative. Thank you to the FOSSEE team, my mentors, and my fellow colleagues for their unwavering support and encouragement. I am honored to be a part of this esteemed community and look forward to contributing to its ongoing success.

Chapter 1

Introduction

FOSSEE (Free/Libre and Open Source Software for Education) is an initiative by the National Mission on Education through Information and Communication Technology (ICT) under the Ministry of Human Resource Development (MHRD), Government of India. This initiative has successfully developed a range of open-source tools and actively promotes their use to enhance the quality of education. By making these resources freely available to everyone, FOSSEE ensures equitable access to educational tools. The software is designed to remain competitive and relevant in comparison to commercial software, continuously evolving to meet the needs of users.

1.1 eSim

eSim is a CAD tool that helps electronic system designers to design, test, and analyze their circuits. The important feature of this tool is that it is open source, allowing users to modify the source as per their needs. The software provides a generic, modular, and extensible platform for experimenting with electronic circuits.

eSim is built using various free/libre and open-source software components including :

1.1.1 KiCad

Integrated software where all functions of circuit drawing, control, layout, library management, and access to the PCB design software are carried out.

1.1.2 Ngspice

Ngspice is a general-purpose circuit simulation program for nonlinear dc, nonlinear transient, and linear ac analysis.

1.1.3 KiCad to Ngspice converter

Analysis parameters,source details are provided through this module. It allows us to add and edit the device models and subcircuits included in the circuit schematic.

1.1.4 Model Builder

This tool provides the facility to define a new model for devices such as:

- Diode
- Bipolar Junction Transistor (BJT)
- Metal Oxide Semiconductor Field Effect Transistor (MOSFET)
- Junction Field Effect Transistor (JFET)
- IGBT
- Magnetic core

1.1.5 Subcircuit Builder

This module allows the user to create a subcircuit for a component. Once the subcircuit for a component is created, the user can use it in other circuits.

1.1.6 NGHDL

A module for mixed signal circuit simulation, is also integrated with eSim. It makes use of VHDL code.

1.1.7 NgVeri

NgVeri, a module for mixed signal circuit simulation, is also integrated with eSim. It makes use of Verilog/System Verilog/Transaction-Level Verilog code.

1.1.8 Makerchip

Makerchip is a cloud-based browser application developed by Redwood EDA to do digital circuit design. One can simulate Verilog/SystemVerilog/Transaction-Level Verilog code in Makerchip.

1.1.9 Verilator

Verilator is a Verilog/SystemVerilog simulator tool. It converts the Verilog/SystemVerilog code to C++ object files. These object files are linked with that of Ngspice thus enabling mixed signal simulation in eSim.

1.1.10 SandPiper SaaS

Sandpiper-saas is a tool developed by Redwood EDA which converts Transaction Level Verilog code to SystemVerilog code. It is used by NgVeri so that it can get the System Verilog code which can be further passed to the Verilator.

Chapter 2

Abstract

The objective of this internship was to design and develop various integrated circuits using the Subcircuit Builder Method in eSim. This involved modeling the ICs with eSim library files and subsequently simulating them with different circuits. The goal was to expand the eSim Subcircuit Library for future use, enhancing its utility and application in educational and practical scenarios.

2.1 Approach

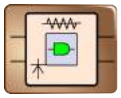
- Identify and research an integrated circuit (IC) that is not currently available in the eSim library.
- Obtain and study the datasheet of the selected IC thoroughly.
- Carefully examine the schematic provided in the datasheet.
- Accurately recreate the schematic in eSim using the Subcircuit Builder Method.
- Model the IC in eSim, ensuring all parameters and configurations match those in the datasheet.
- Simulate the integrated circuit within eSim, testing it with various circuits to verify its functionality.
- Document the process and results to contribute to the future use and expansion of the eSim Subcircuit Library.

2.2 Execution

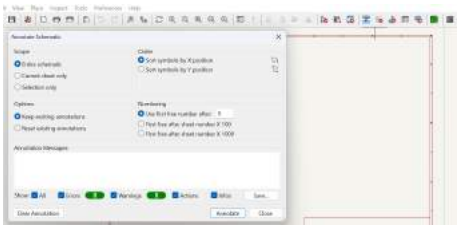
The following steps outline the method for creating a subcircuit in eSim 2.4 for the Windows version.

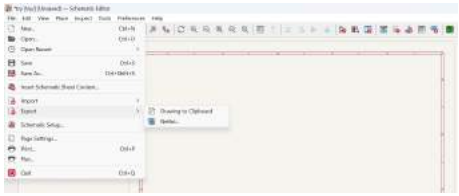
- Upon starting eSim, the initial window that appears is shown in the figure above. When eSim is opened, all files are by default stored in the **eSim Workspace**.

- Click on the **Open Project** icon mentioned above and create a new project in eSim.



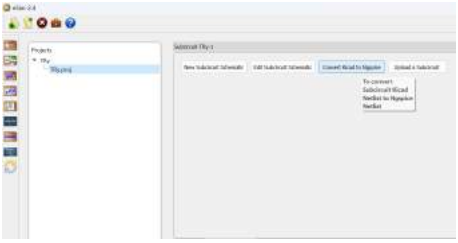
- Now, click on the **Subcircuit** icon mentioned above, which will open a new window. Click on **New Subcircuit Schematic**, enter the **schematic name**, and an **Eeschema** window will appear, allowing you to recreate the diagram mentioned in the datasheet.





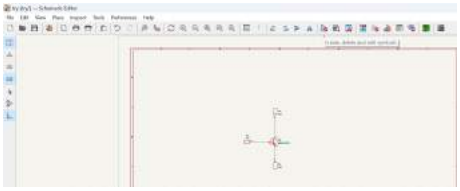
- After completing your design in **Eeschema** based on the datasheet and connecting the **ports** at the respective places, click on the **Annotate Schematic Symbols** icon as shown above to annotate your design.

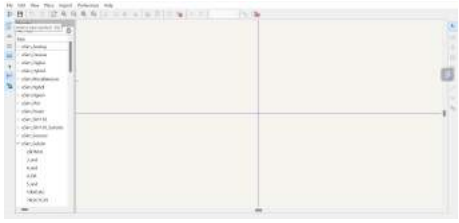
- item Once the design is annotated, click on the **Generate Netlist** icon as shown above. This will create a netlist file which is essential for the simulation and further processing in eSim.



- After generating the netlist, return to the **eSim window** and click on the **Convert KiCad to NgSpice** option.

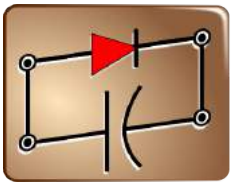
- In the new window that appears, add the **model file** from the built-in model library according to the devices used in your circuit.





- After modeling, get back to the **Schematic Editor** of eSim and click on the **Create, Delete, and Edit Symbol** option as shown in the snapshot.
- A new window called **Symbol Editor** will appear. Select the **eSim Sbckt** option and click on **Create a New Symbol** as mentioned in the snapshot.

- A box called **New Symbol** Box will open. Fill in the details of your device in the box and click OK. You can then create your **pin diagram**.



/

- After completing the pin diagram, click on **KiCad** to create the circuit for simulation. Convert the circuit to **NgSpice**, click on the **Simulation** Symbol, and check the output against the datasheet.

Chapter 3

Integrated Circuit Design

3.1 MC1709

3.1.1 Description

It is a Monolithic Operational Amplifier designed for use as a summing amplifier,integrator, or amplifier with operating characteristics dependent on external feedback components.

Features of MC1709:

1. High Performance Open-Loop Gain Characteristics: A_{vol} = 45,000 Typically.
2. Low Temperature Drift: $\pm 3.0\ \mu V/°C$.
3. Large Output Voltage Swings: ±14V @ 15V Supply.
4. Low Output Impedance: $Z_0 = 150\ \Omega$.

3.1.2 Pin Diagram

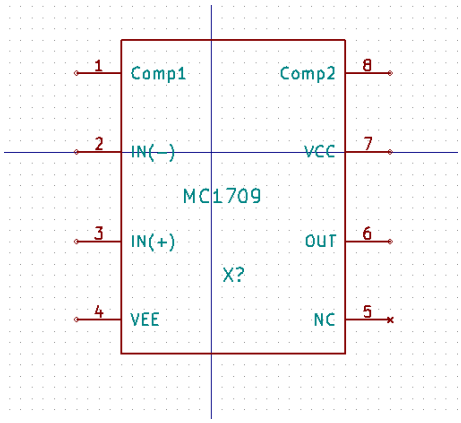


Figure 3.1: Pin Diagram of MC1709

3.1.3 Schematic Diagram

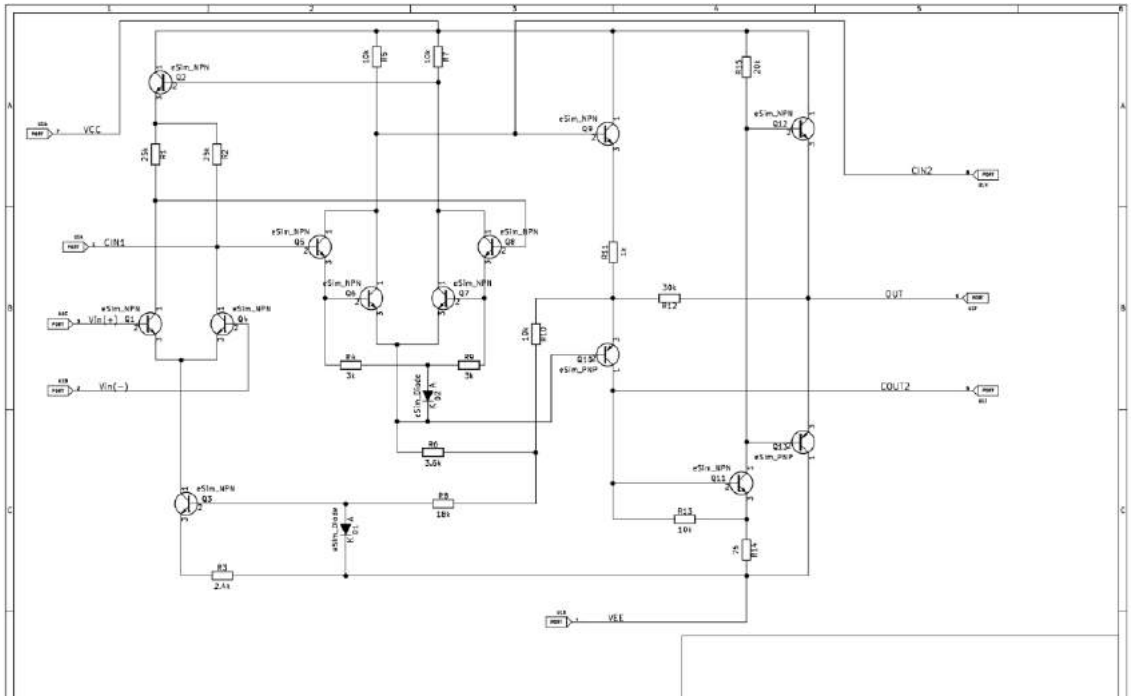


Figure 3.2: Schematic Diagram of MC1709

3.1.4 Simulation Circuit

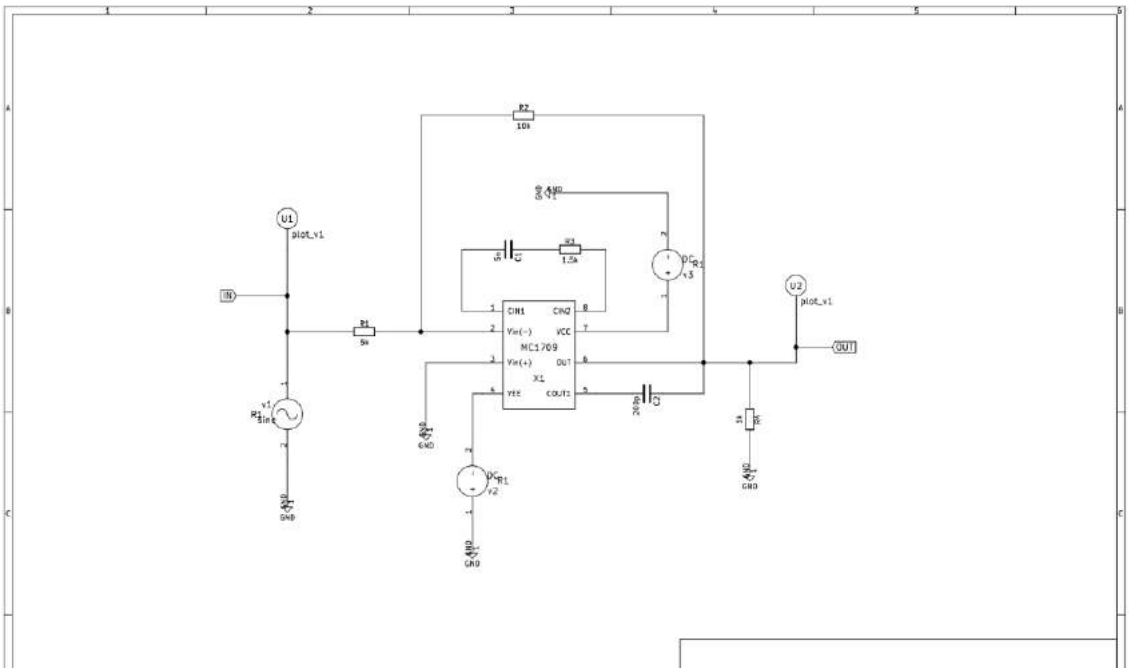


Figure 3.3: Simulation Circuit of MC1709

3.1.5 NgSpice Plot

Input plot

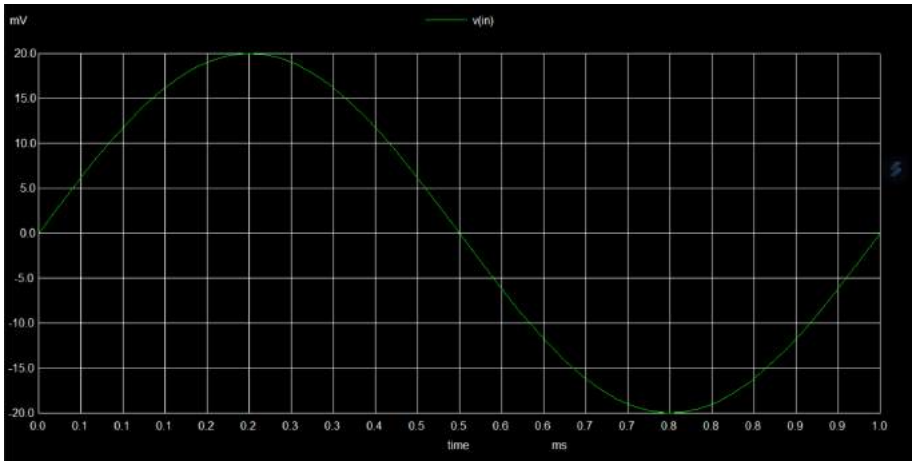


Figure 3.4: Input plot of MC1709

Output plot

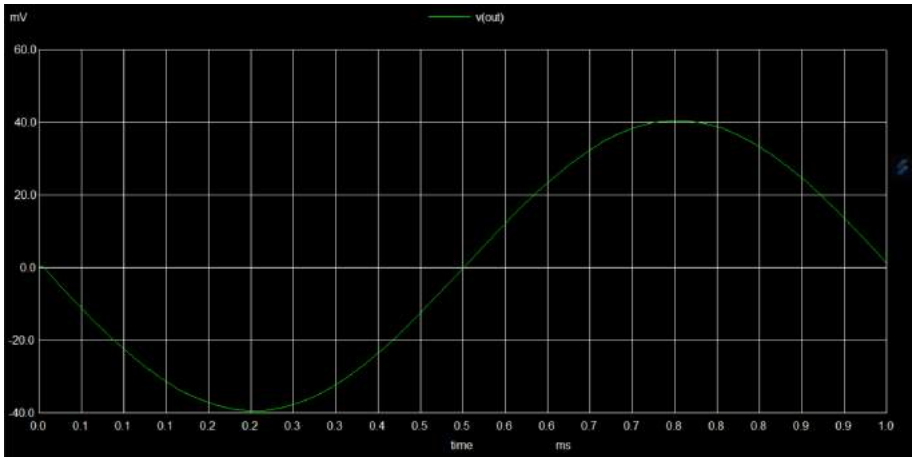


Figure 3.5: Output plot of MC1709

3.2 MC1741C

3.2.1 Description

The MC1741C is an internally compensated, high-performance operational amplifier. It is designed for use as a summing amplifier, integrator, or amplifier with operating characteristics determined by the external feedback components.

Features of MC1741C:

- No frequency compensation required
- Short circuit protection
- Offset voltage null capability
- Wide common mode and differential voltage ranges
- Low power consumption
- No latch-up

3.2.2 Pin Diagram

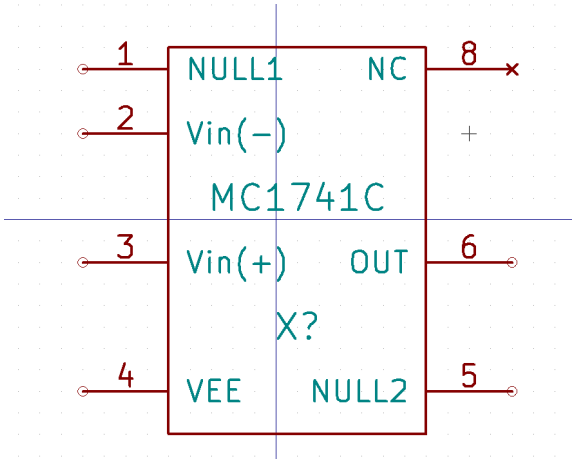


Figure 3.6: Pin Diagram of MC1741C

3.2.3 Schematic Diagram

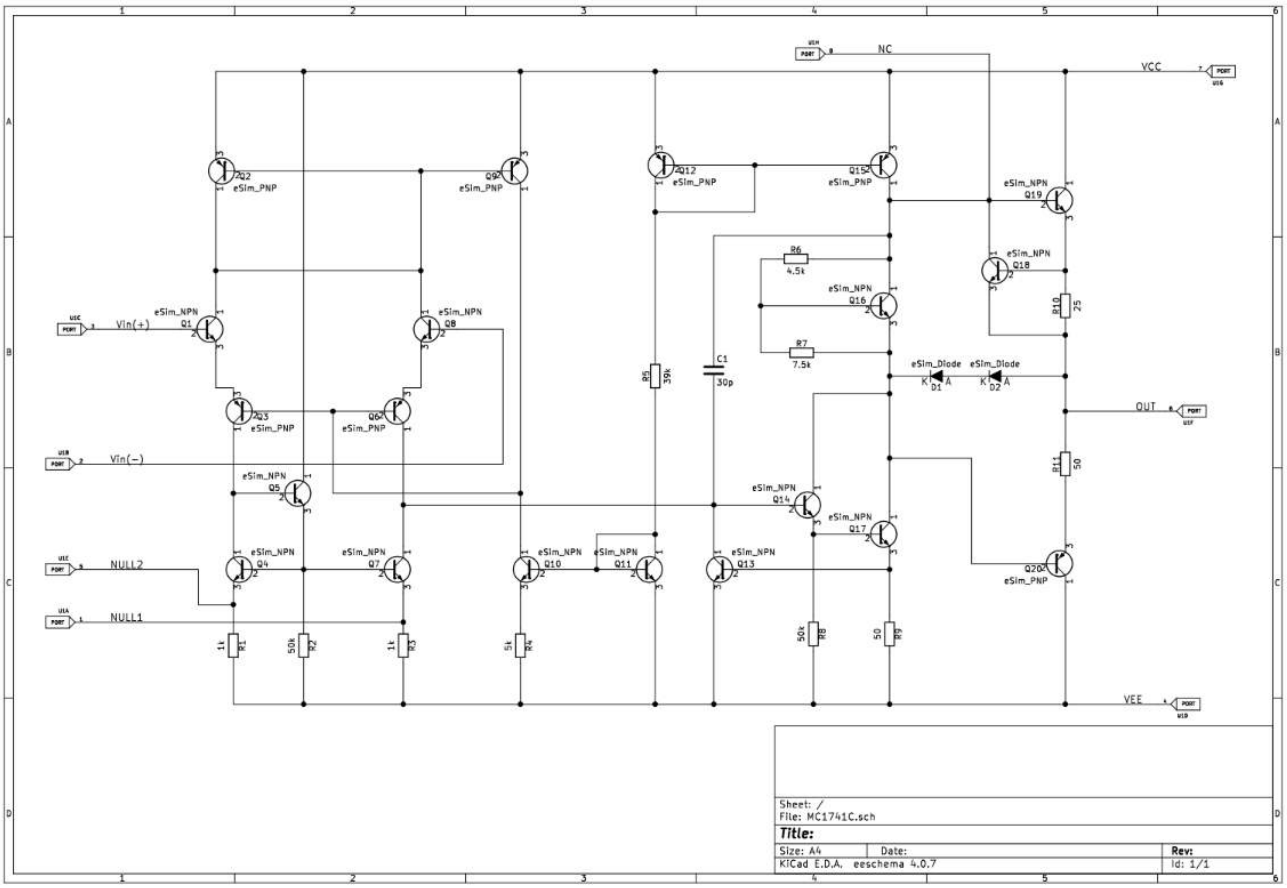


Figure 3.7: Schematic Diagram of MC1741C

3.2.4 Simulation Circuit

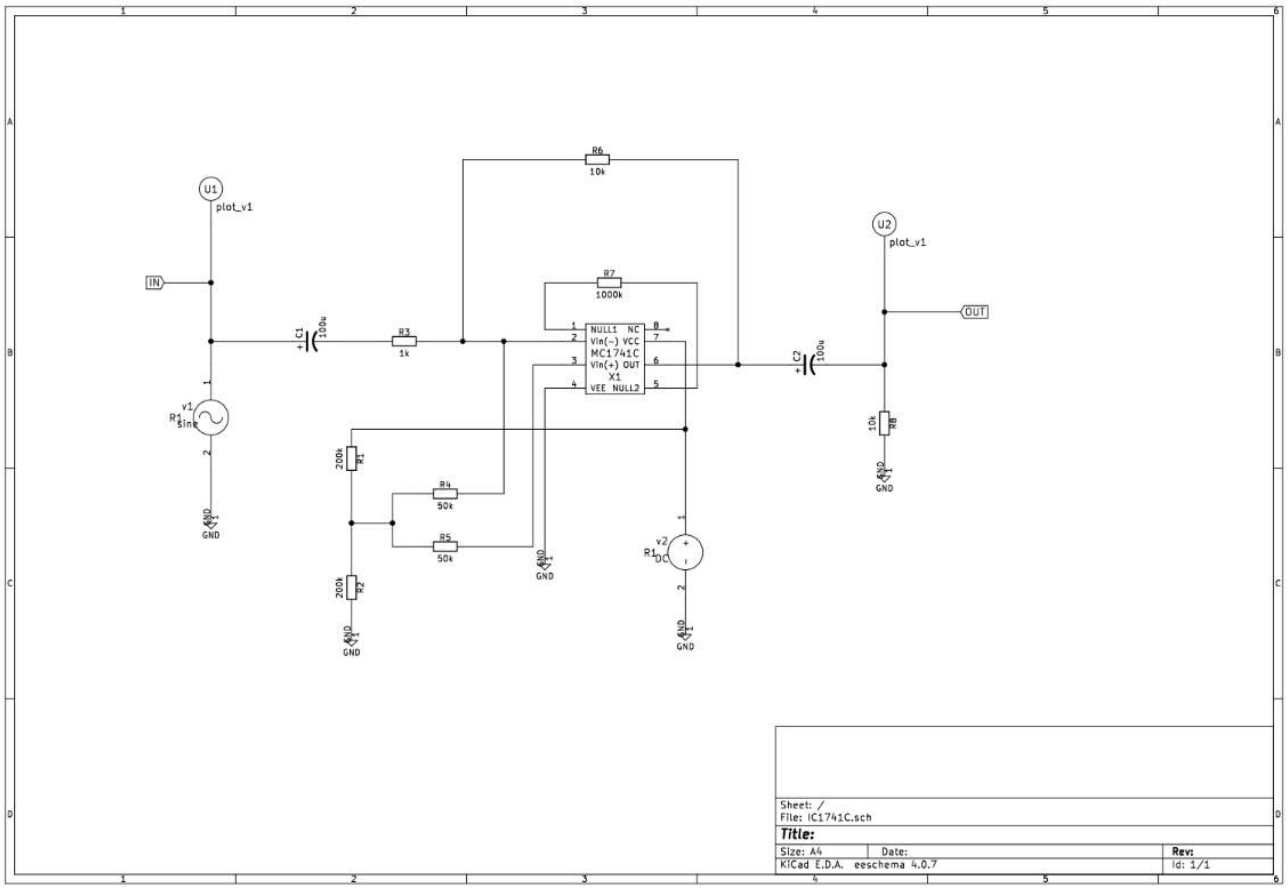


Figure 3.8: Simulation Circuit of MC1741C

3.2.5 NgSpice Plot

Input plot

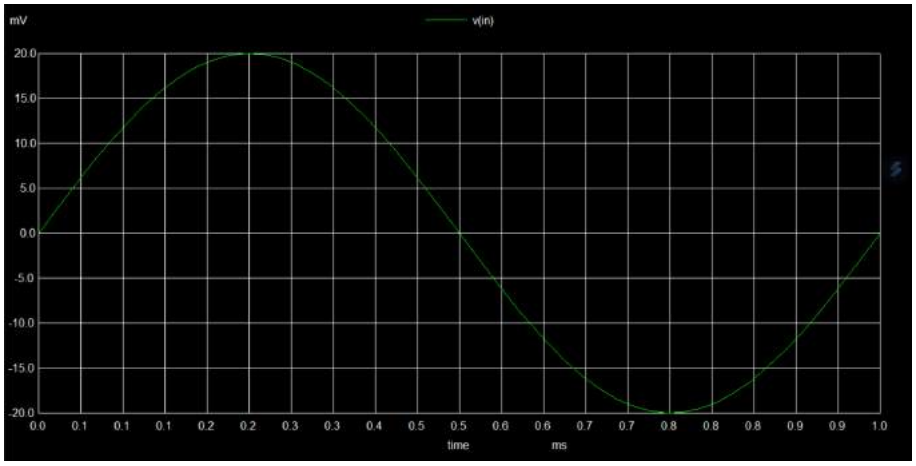


Figure 3.9: Input plot of MC1741C

Output plot

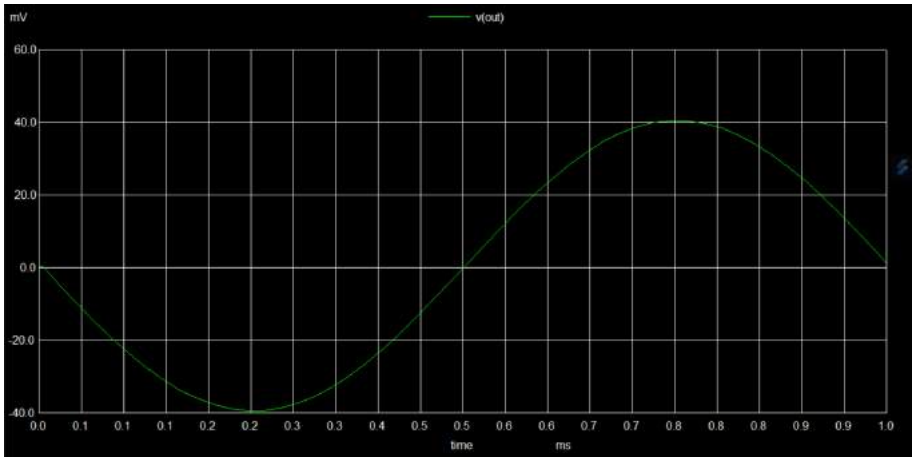


Figure 3.10: Output plot of MC1741C

3.3 TL431C

3.3.1 Description

The TL431C is a 3-terminal adjustable shunt regulator with specified thermal stability over applicable automotive and commercial temperature ranges. The output voltage may be set to any value between V_{REF} (approximately 2.5V) and 36V with two external resistors. These devices have a typical output impedance of 0.2Ω. Active output circuitry provides a very sharp turn-on characteristic, making these devices excellent replacements for Zener diodes in many applications like on-board regulation, adjustable power supplies, and switching power supplies.

Features of TL431C:

- Equivalent full-range temperature coefficient: 30 ppm/°C
- 0.2Ω typical output impedance
- Sink current capability: 1 mA to 100 mA
- Low output noise
- Adjustable output voltage: V_{REF} to 36V

3.3.2 Pin Diagram

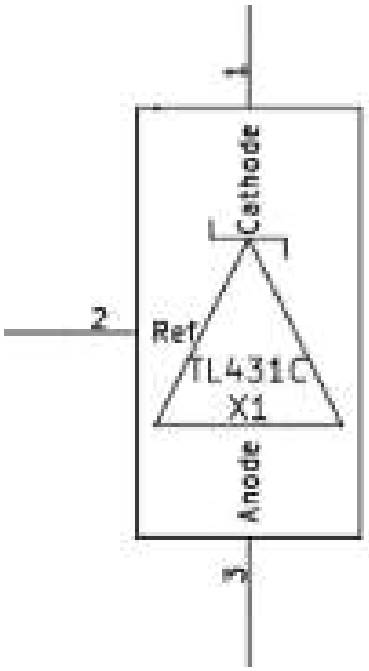


Figure 3.11: Pin Diagram of TL431C

3.3.3 Schematic Diagram

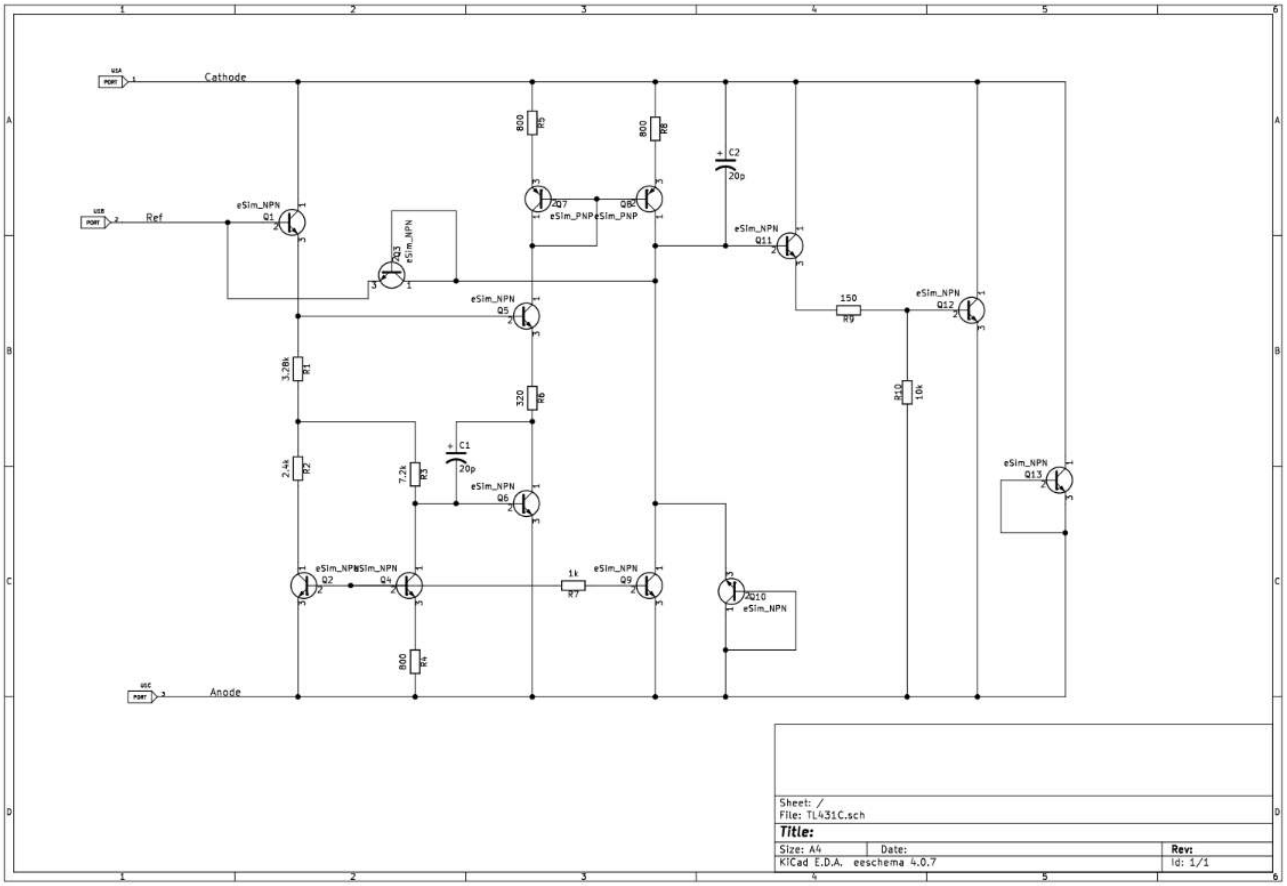


Figure 3.12: Schematic Diagram of TL431C

3.3.4 Simulation Circuit

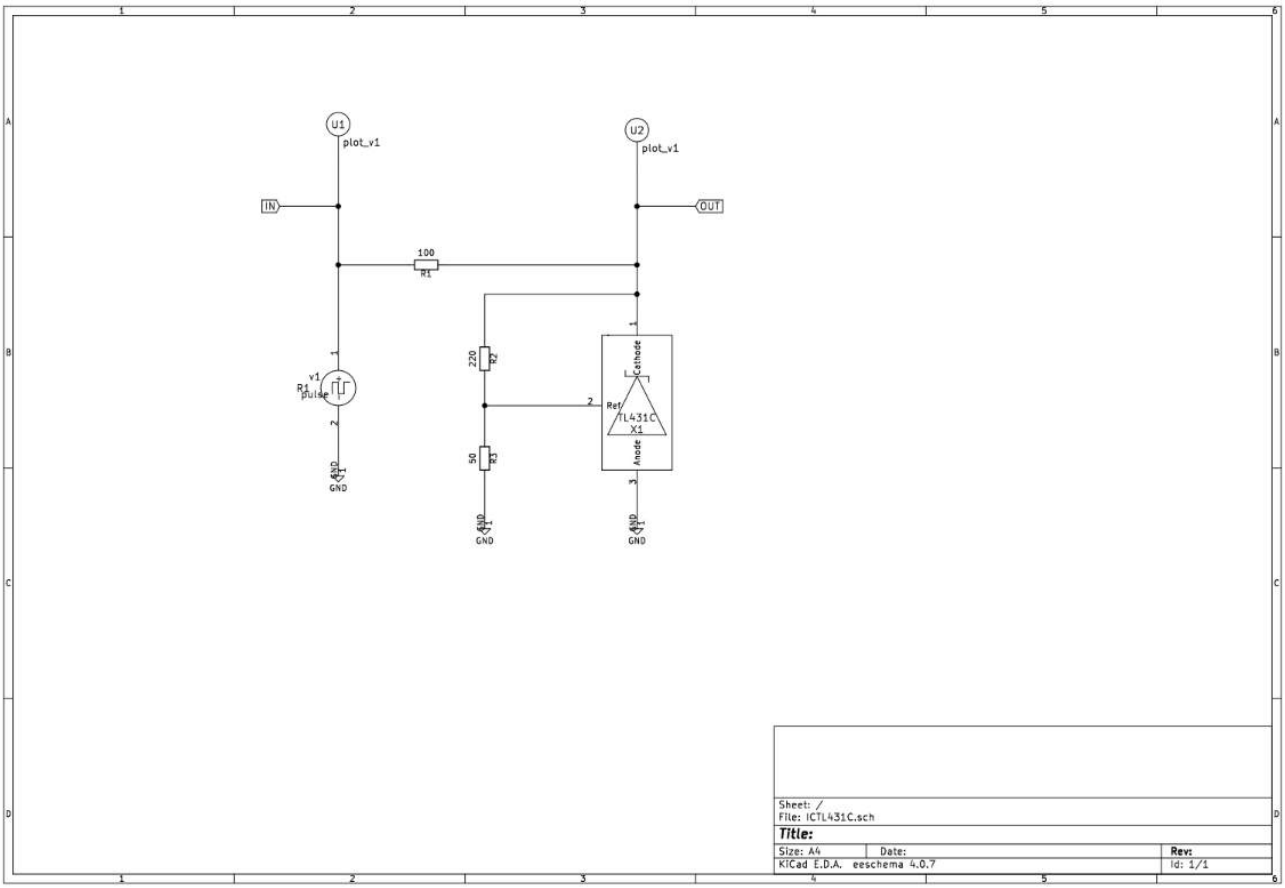


Figure 3.13: Simulation Circuit of TL431C

3.3.5 NgSpice Plot

Input plot

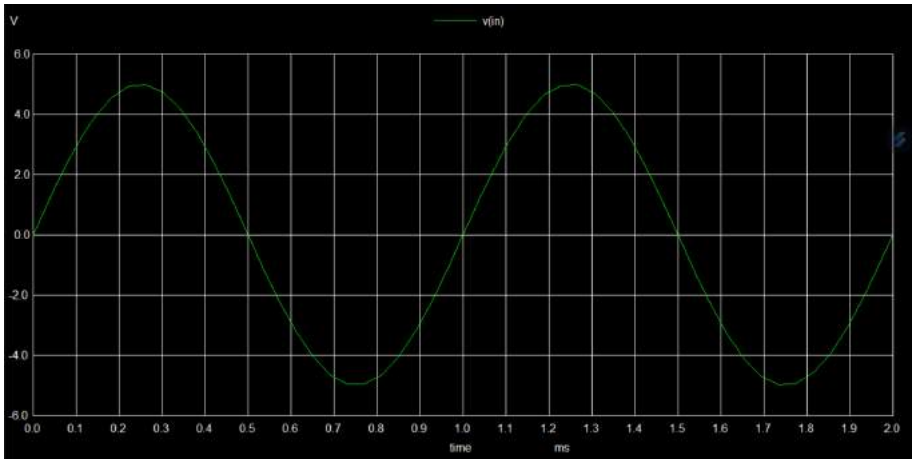


Figure 3.14: Input plot of TL431C

Output plot

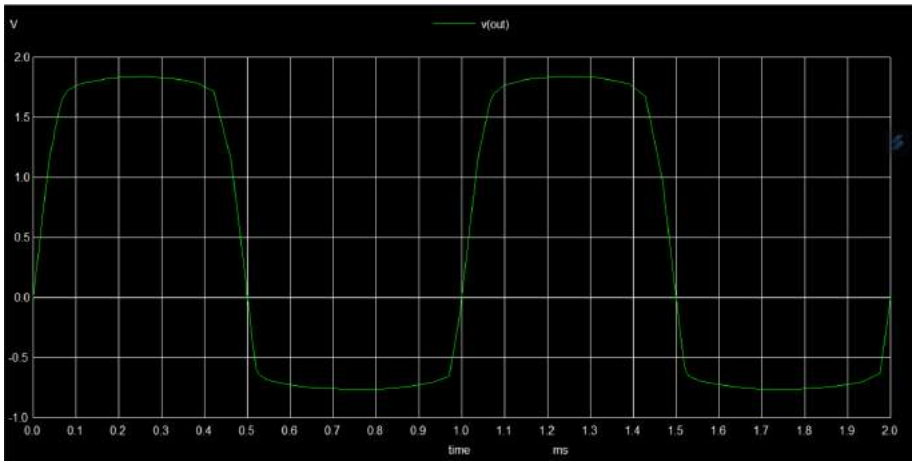


Figure 3.15: Output plot of TL431C

3.4 LM431

3.4.1 Description

The LM431 is a 3-terminal adjustable shunt regulator with ensured temperature stability over the entire temperature range of operation. The output voltage may be set at any level greater than 2.5 V (V_{REF}) up to 36 V merely by selecting two external resistors that act as a voltage divider network. Due to the sharp turn-on characteristics, this device is an excellent replacement for many Zener diode applications.

Features of LM431:

- Average temperature coefficient 50 ppm/°C
- Temperature compensated for operation over the full temperature range
- Programmable output voltage
- Fast turn-on response
- Low-output noise
- Low-dynamic output impedance

3.4.2 Pin Diagram

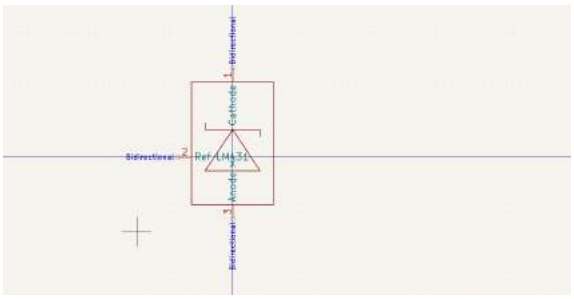


Figure 3.16: Pin Diagram of LM431

3.4.3 Schematic Diagram

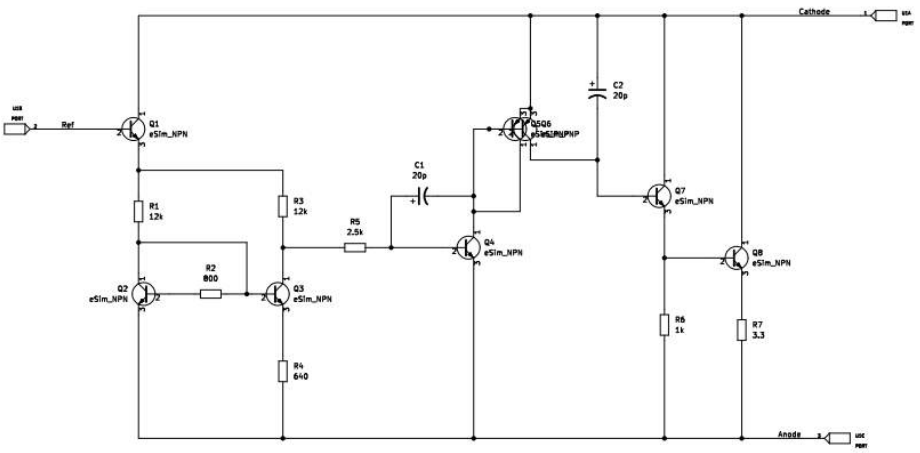


Figure 3.17: Schematic Diagram of LM431

3.4.4 Simulation Circuit

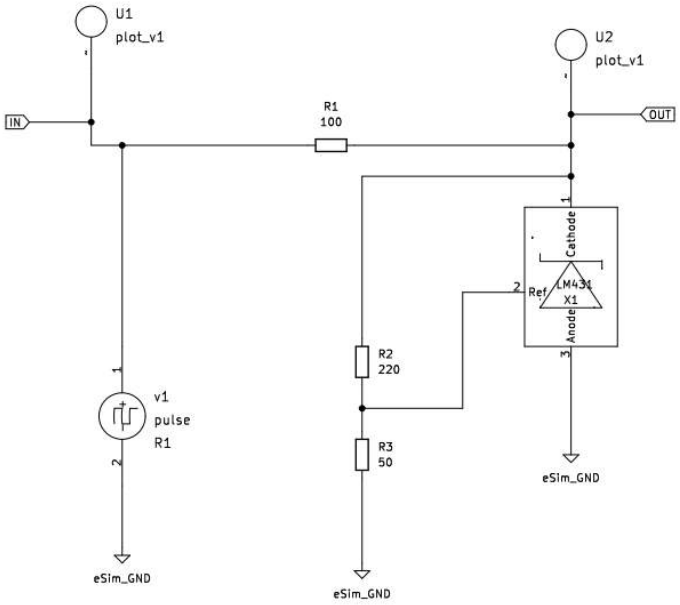


Figure 3.18: Simulation Circuit of LM431

3.4.5 Python Plot

Input plot

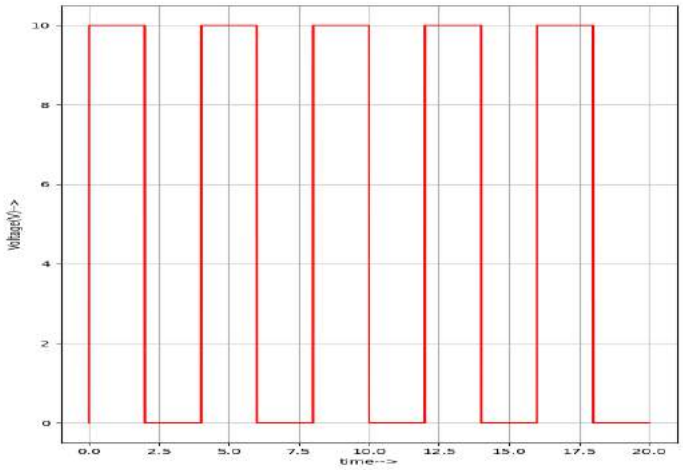


Figure 3.19: Input plot of LM431

Output plot

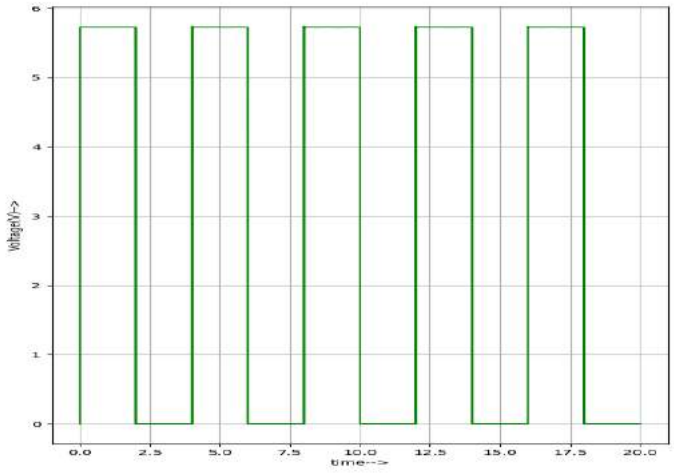


Figure 3.20: Output plot of LM431

3.5 LH0003

3.5.1 Description

The LH0003 is a Wide Bandwidth Operational Amplifier. The LH0003/LH0003C is a general-purpose operational amplifier that features a slew rate of up to 70 V/ s, a gain bandwidth of up to 30 MHz, and high output currents. The LH0003 is specified for operation over the -55°C to 125°C military temperature range. The LH0003C is specified for operation over the 0°C to 85°C temperature range.

- Very low offset voltage: Typically 0.4 mV
- Large output swing: ±10V into 100Ω load
- High CMRR: Typically 90 dB
- Good large signal frequency response: 50 kHz to 400 kHz depending on compensation

3.5.2 Pin Diagram

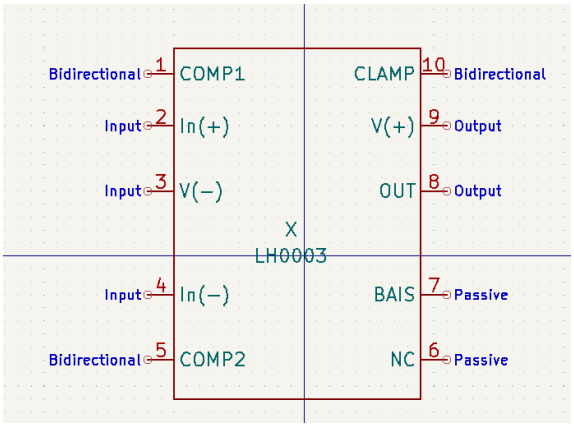


Figure 3.21: Pin Diagram of LH0003

3.5.3 Schematic Diagram

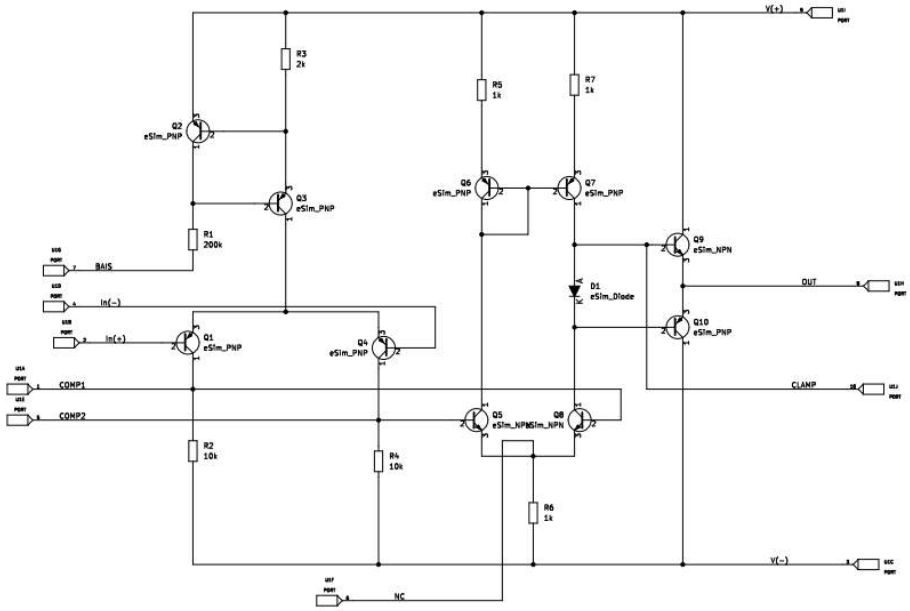


Figure 3.22: Schematic Diagram of LH0003

3.5.4 Simulation Circuit

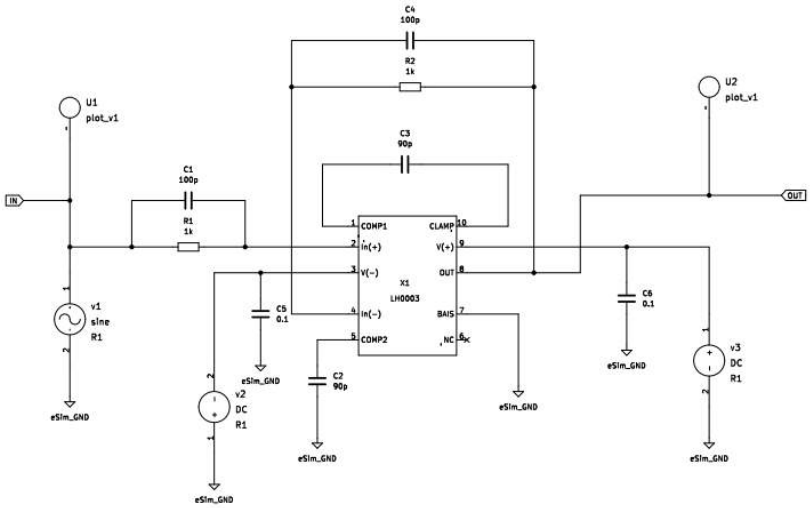


Figure 3.23: Simulation Circuit of LH0003

3.5.5 Python Plot

Input plot

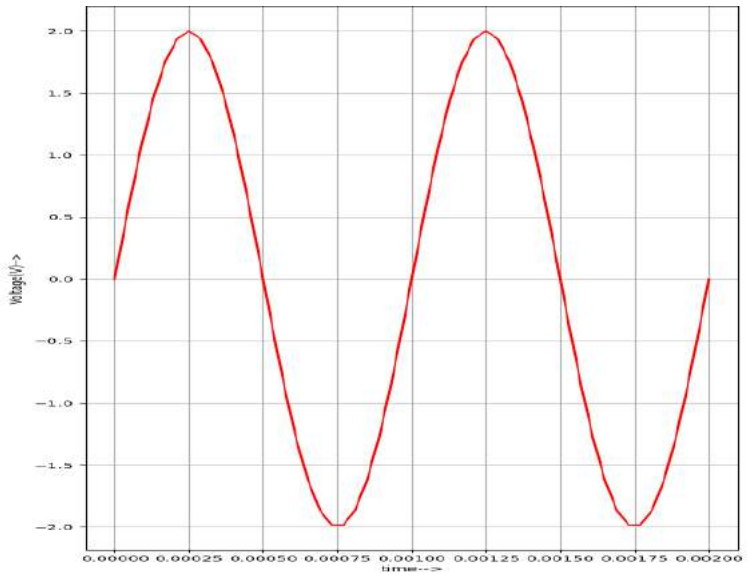


Figure 3.24: Input plot of LH0003

Output plot

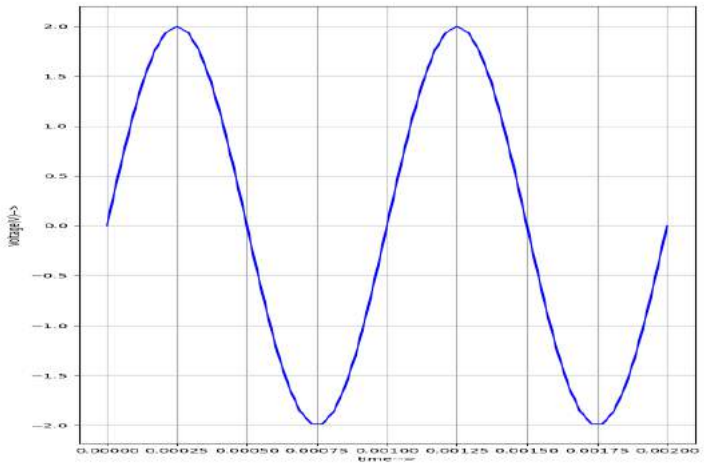


Figure 3.25: Output plot of LH0003

3.6 LM3080

3.6.1 Description

The LM3080 is an Operational Transconductance Amplifier. The LM3080 is a programmable transconductance block intended to fulfill a wide variety of variable gain applications. The LM3080 has differential inputs and high impedance push-pull outputs. The device has high input impedance and its transconductance (g_m) is directly proportional to the amplifier bias current (I_{ABC}). High slew rate together with programmable gain make the LM3080 an ideal choice for variable gain applications such as sample and hold, multiplexing, filtering, and multiplying.

Features of LM3080:

- Slew rate (unity gain compensated): 50 V/ s
- Fully adjustable gain: 0 to g_m # RL limit
- Extended g_m linearity: 3 decades
- Flexible supply voltage range: ±2V to ±18V
- Adjustable power consumption

3.6.2 Pin Diagram

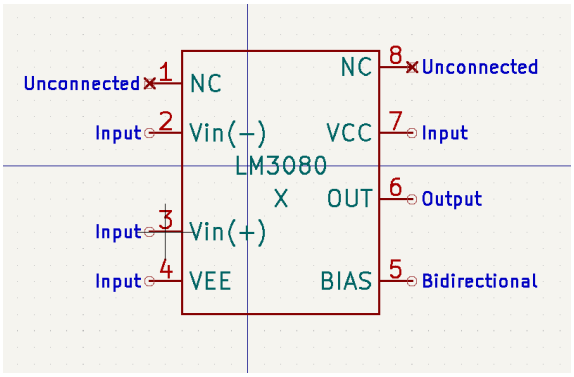


Figure 3.26: Pin Diagram of LM3080

3.6.3 Schematic Diagram

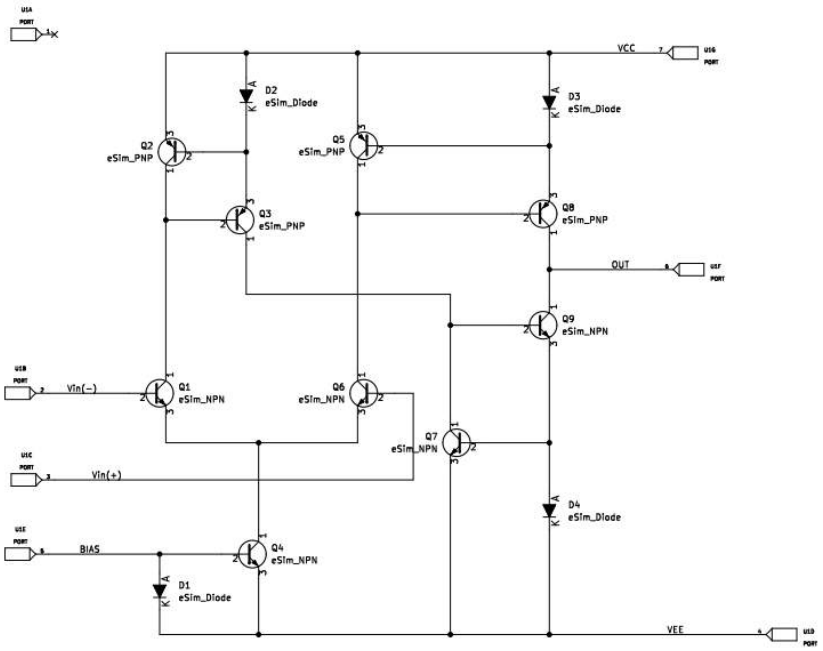


Figure 3.27: Schematic Diagram of LM3080

3.6.4 Simulation Circuit

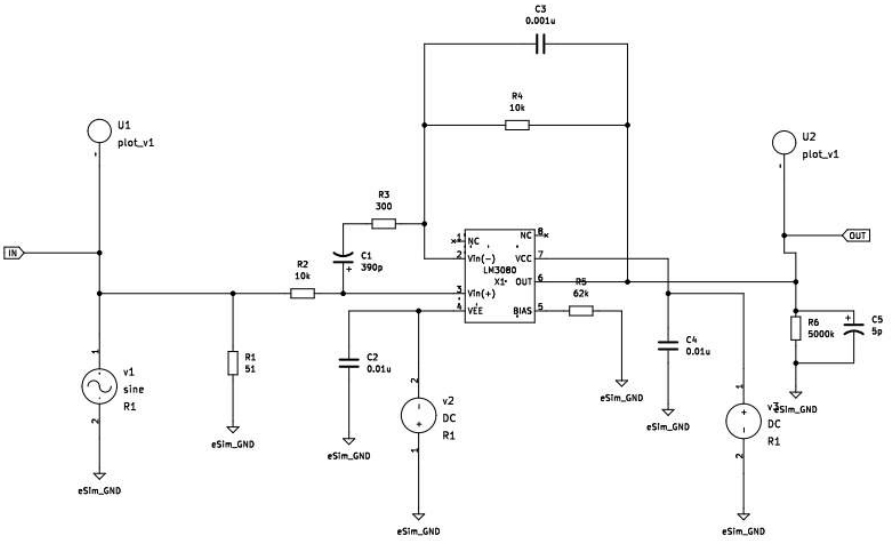


Figure 3.28: Simulation Circuit of LM3080

3.6.5 Python Plot

Input plot

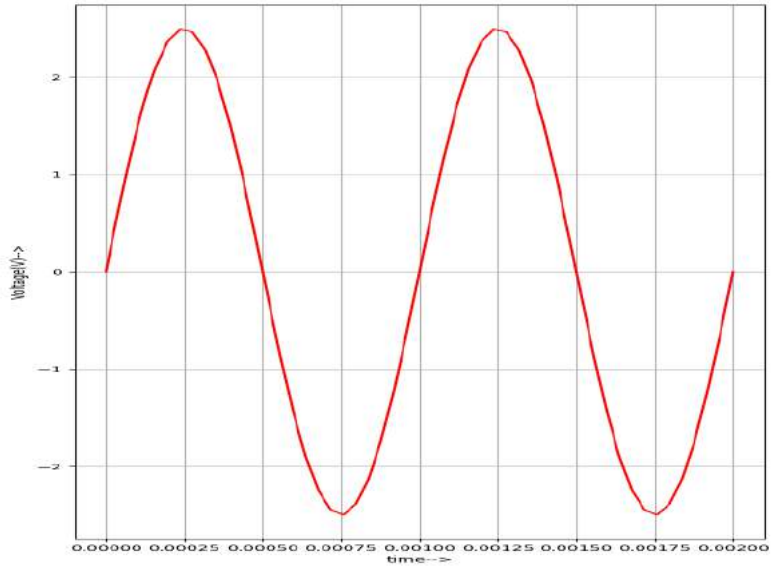


Figure 3.29: Input plot of LM3080

Output plot

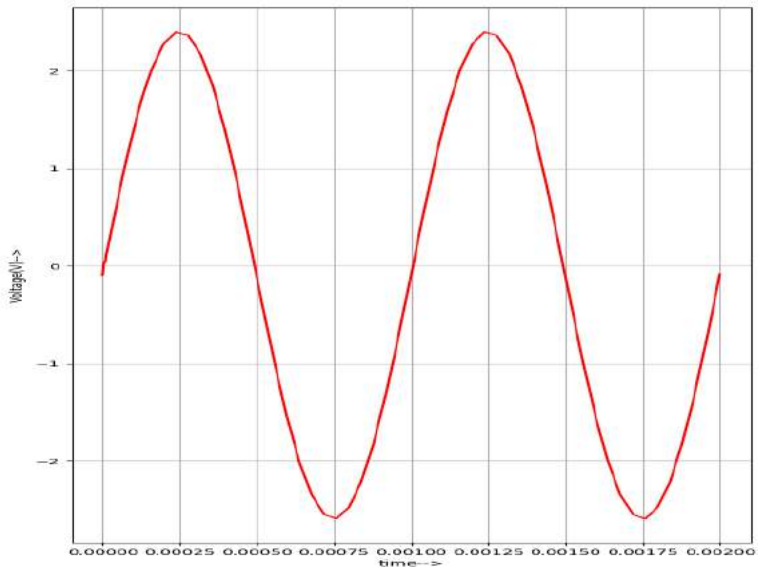


Figure 3.30: Output plot of LM3080

3.7 LM709

3.7.1 Description

The LM709 series is a monolithic operational amplifier intended for general-purpose applications. Operation is completely specified over the range of voltages commonly used for these devices. The design, in addition to providing high gain, minimizes both offset voltage and bias currents. Further, the class-B output stage gives a large output capability with minimum power drain. External components are used to frequency compensate the amplifier. Although the unity-gain compensation network specified will make the amplifier unconditionally stable in all feedback configurations, compensation can be tailored to optimize high-frequency performance for any gain setting. The LM709C is the commercial-industrial version of the LM709. It is identical to the LM709 except that it is specified for operation from 0°C to +70°C.

Features of LM709:

- High gain
- Minimizes offset voltage and bias currents
- Large output capability with minimum power drain
- Frequency compensation with external components
- Unconditionally stable in all feedback configurations with unity-gain compensation network
- Optimizable high-frequency performance for any gain setting

3.7.2 Pin Diagram

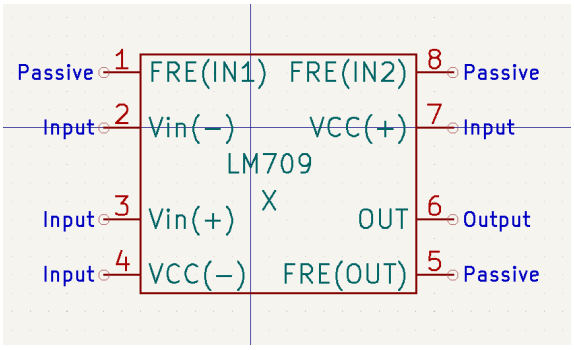


Figure 3.31: Pin Diagram of LM709

3.7.3 Schematic Diagram

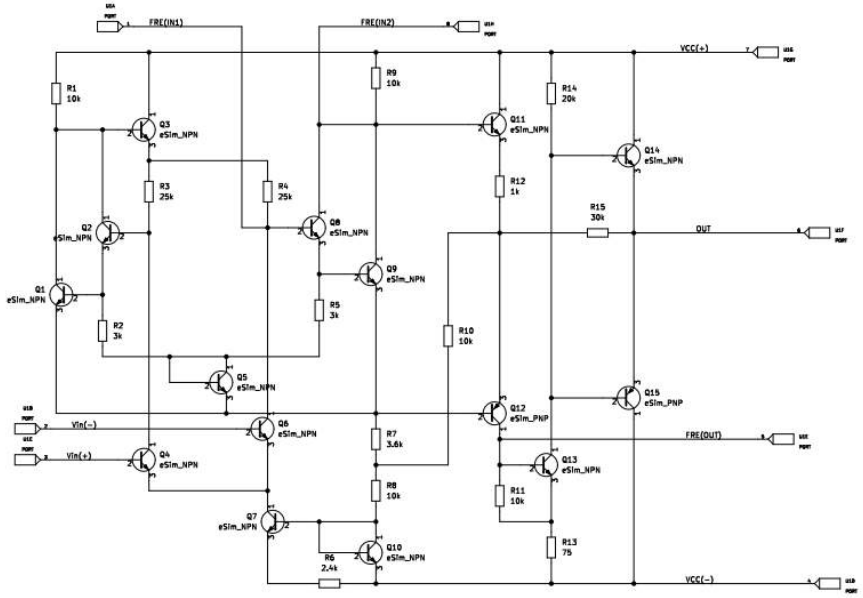


Figure 3.32: Schematic Diagram of LM709

3.7.4 Simulation Circuit

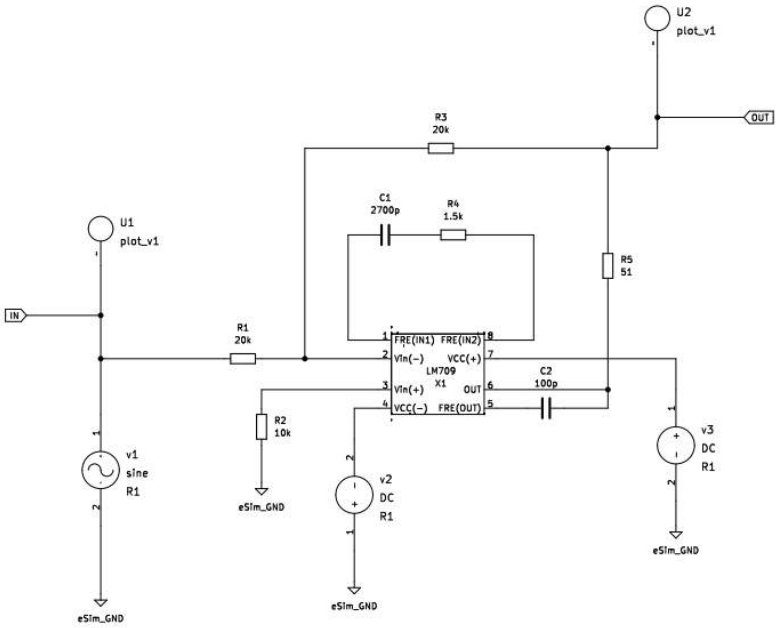


Figure 3.33: Simulation Circuit of LM709

3.7.5 Python Plot

Input plot

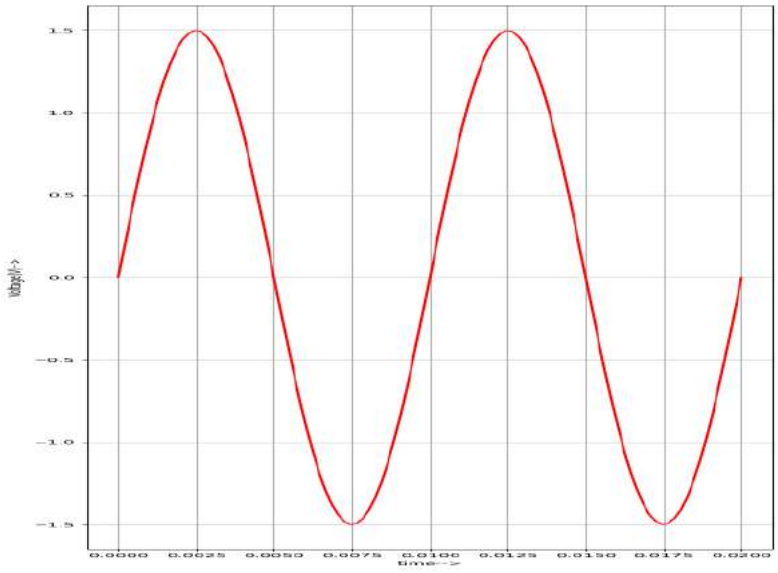


Figure 3.34: Input plot of LM709

Output plot

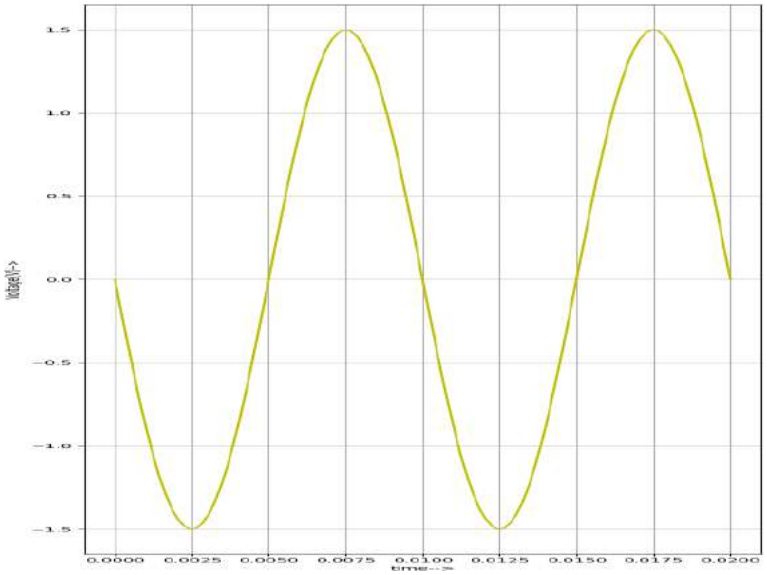


Figure 3.35: Output plot of LM709

3.8 LM725

3.8.1 Description

The LM725/LM725A/LM725C are operational amplifiers featuring superior performance in applications where low noise, low drift, and accurate closed-loop gain are required. With high common mode rejection and offset null capability, it is especially suited for low-level instrumentation applications over a wide supply voltage range. The LM725A has tightened electrical performance with higher input accuracy and like the LM725, is guaranteed over a -55°C to $+125^\circ$C temperature range.

- High open loop gain: 3,000,000
- Low input voltage drift: 0.6 µV/$^\circ$C
- High common mode rejection: 120 dB
- Low input noise current: 0.15 pA/$\sqrt{\text{Hz}}$
- Low input offset current: 2 nA
- High input voltage range: ±14V
- Wide power supply range: ±3V to ±22V

3.8.2 Pin Diagram

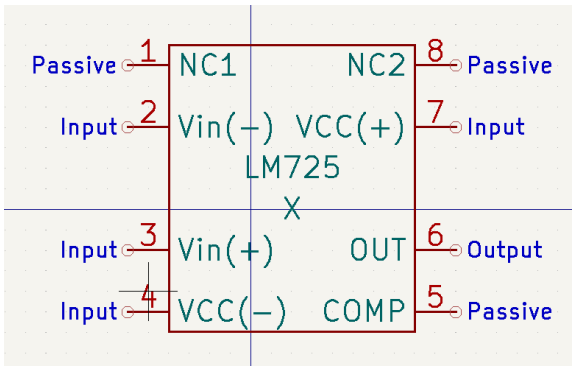


Figure 3.36: Pin Diagram of LM725

3.8.3 Schematic Diagram

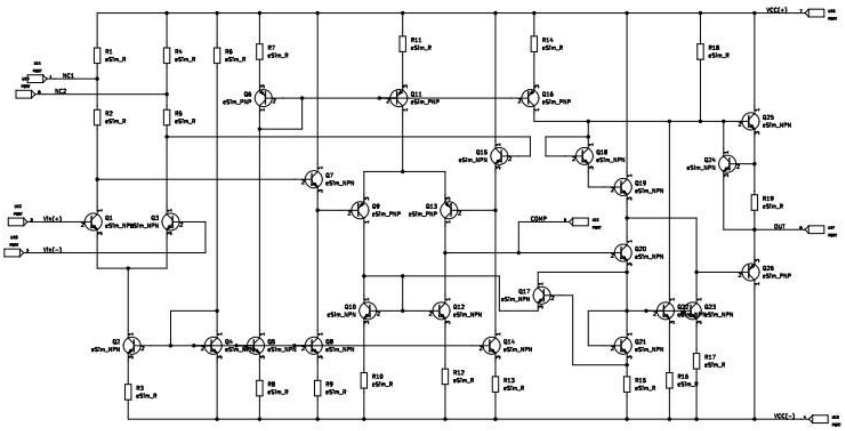


Figure 3.37: Schematic Diagram of LM725

3.8.4 Simulation Circuit

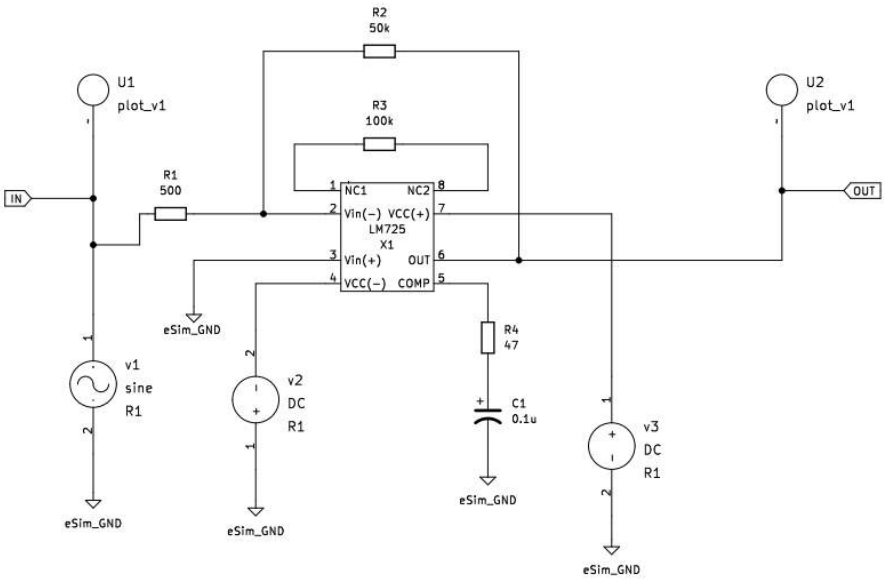


Figure 3.38: Simulation Circuit of LM725

3.8.5 Python Plot

Input plot

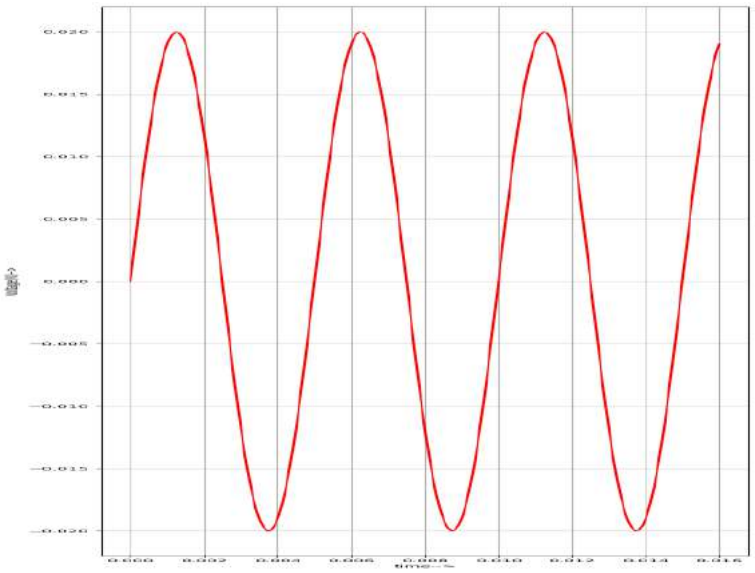


Figure 3.39: Input plot of LM725

Output plot

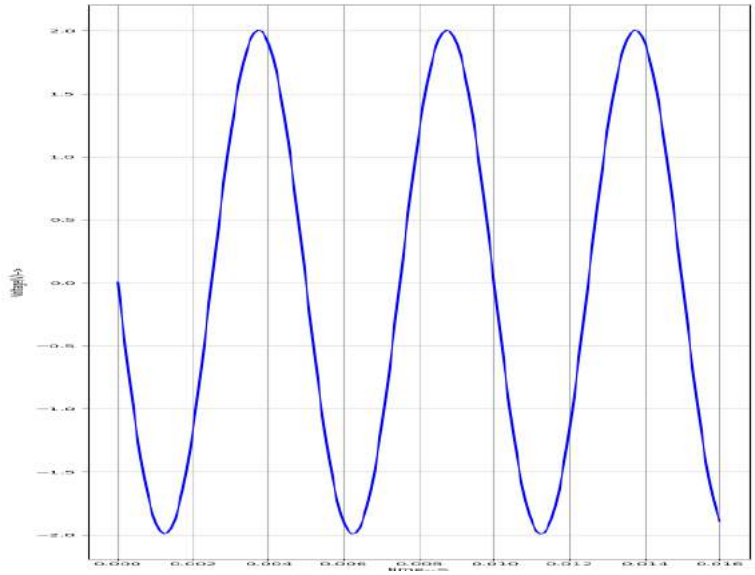


Figure 3.40: Output plot of LM725

Chapter 4

Conclusion and Future Scope

I have successfully achieved the development of various subcircuits in Analog Circuit Design, including Operational Amplifiers, Positive and Negative Shunt Voltage Regulators, and Operational Transconductance Amplifiers. Each Integrated Circuit (IC) model was meticulously developed according to the specifications outlined in their official datasheets. The outputs of each IC were verified and tested successfully using their respective test circuits.

However, I have encountered significant issues with DC shift in some of the ICs. Despite extensive efforts to address this problem, certain ICs continue to experience DC shift. Consequently, I have placed these ICs on hold, hoping that this issue can be resolved in the future. If any colleagues manage to resolve this DC shift problem, it would greatly alleviate many challenges in IC design.

Developers and students can utilize these ICs in their projects and circuit models as modular units. With the ongoing development and expansion of the device model library in eSim, we anticipate the creation of more ready-to-use IC models for integration into eSim. Each Integrated Circuit I design is incorporated into the eSim built-in library for the benefit of future users. In today's electronics world, reusing existing components instead of building everything from scratch is essential for saving time and resources. This approach not only streamlines the design process but also enhances efficiency and innovation in electronic circuit development.

Chapter 5

Bibliography

- **FOSSEE:**
  https://fossee.in/
- **eSim:**
  https://static.fossee.in/esim/manuals/eSim_Manual_2.4.pdf
- **MC1709:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/130010/MOTOROLA/MC1709.html
- **MC1741C:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/1423773/ONSEMI/MC1741C.html
- **TL431C:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/19758/PHILIPS/TL431C.html
- **LM431:**
  https://www.ti.com/lit/ds/symlink/lm431.pdf
- **LH0003:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/143277/NSC/LH0003.html
- **LM3080:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/8835/NSC/LM3080.html
- **LM709:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/9015/NSC/LM709.html
- **LM725:**
  https://www.alldatasheet.com/datasheet-pdf/pdf/9022/NSC/LM725.html
- **Report:**
  https://www.papeeria.com/p/4293c7fec204afc25e9556c18e972784