Winter Internship Report

On

Including Integrated Circuit (IC) Subcircuit on eSim

Submitted by

Hrittika Ghosh

Under the guidance of

Prof. Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

April 20, 2023

Acknowledgment

I would like to convey my gratitude to the entire FOSSEE team for providing me with this esteemed Fellowship opportunity. This was a truly enlightening experience working with an open source organization such as this where I could learn about a new simulation software like eSim and acquire plenty of knowledge about approaching a problem and troubleshooting to find solutions. I deeply look forward to cherishing this learning experience in the forthcoming years of my career.

I would like to extend my sincere regards to Prof. Kannan M. Moudgalya for his vision and mission to take this initiative forward and providing me with a chance to be a small part of this project as an intern.

I also want to express my warm gratitude to the entire FOSSEE team including our mentors Mr. Sumanto Kar, Mr. Rahul Paknikar, and Mrs. Usha Vishwanathan for constantly guiding and mentoring me throughout the duration of my internship. It is with their support that I have been able to fulfil my project demands successfully. Whenever I faced an issue, our mentors were always accessible to help me debug through it and find the correct solution. My learnings from them have been most valuable and I shall carry it forward in my future.

A special thanks to all my fellow interns who have kept me motivated throughout the Fellowship to work together for this project in our respective capacities. Overall, it was a delightful experience interning at FOSSEE and contributing to its growth and I take away some great insights and knowledge from it. Also, as a beginner enthusiastic about the semiconductor industry, this internship is a milestone for me in my pursuit of a successful career.

Table of Contents

Chapter 1

Introduction

FOSSEE (Free/Libre and Open Source Software for Education) is an initiative taken by the National Mission on Education through Information and Communication Technology (ICT), Ministry of Human Resource Development (MHRD), Government of India which has successfully developed various opensource tools and promotes the use of these tools in improving the quality of education and helping every individual avail these sources free of cost. The software is being developed in such a way that it can stay relevant with respect to the commercial softwares.

1.1 eSim

eSim is a free/libre and open source EDA tool for circuit design, simulation, analysis and PCB design developed by FOSSEE, IIT Bombay. It is an integrated tool built using free/libre and open source software such as KiCad, Ngspice, NGHDL and GHDL.

1.2 NgSpice

Ngspice is a general purpose circuit simulation program for nonlinear dc, nonlinear transient, and linear ac analysis. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, lossless and lossy transmission lines (two separate implementations), switches, uniform distributed RC lines, and the five most common semiconductor devices: diodes, BJTs, JFETs, MESFETs, and MOSFET.

Chapter 2

Task Chosen

Implementing some standard Integrated Circuits (IC) as subcircuits into the eSim library.

2.1 Approach

The approach used by me to implement the problem statement is to first look into the datasheets of some prominent Integrated Circuits manufactured by companies like Texas Instruments, Analog Devices, NXP Semiconductors among others. The ICs are so chosen such that there is a variety of utilities served. For example, the ones I have included range from precision amplifiers, comparators, and drivers to audio amplifiers etc. The subcircuits built are then tested to verify basic circuit configurations through NgSpice simulations. The point-by-point roadmap of the process is as follows:

1. Browse through datasheets of relevant ICs that are not previously included into the eSim library.
2. Check for the detailed schematic of the IC and implement the same in the eSim subcircuit builder.
3. Convert the subcircuit from Kicad to NgSpice and create a pin diagram for the IC so that it gets added to the library.
4. Then, the next part involves verifying the subcircuit through a test circuit. We create a new project and build a relevant circuit to test the IC.
5. Convert the test circuit from Kicad to NgSpice and simulate it. We verify the waveforms from the datasheet and assess if the IC operates as per standards.

The same process is followed for all other subcircuits.

Chapter 3

Subcircuit Builder Method

Subcircuit is a way to implement hierarchical modelling. Once a subcircuit for a component is created, it can be used in other circuits. eSim provides an easy way to create a subcircuit. The following section deals with the step-by-step method of creating a subcircuit.

3.1 Subcircuit Creation

The steps to create subcircuit are as follows:

1. After opening the Subcircuit tool, click on New Subcircuit Schematic button. It will ask the name of the subcircuit. Enter the name of subcircuit (without any spaces) and click OK.
2. After clicking OK button it will open KiCad schematic. Draw your circuit which will be later used as a subcircuit.
3. Once you complete the circuit, assign port to the node of your circuit which will be used to connect with the main circuit. PORTS can be found in the components section in the editor. These act as linkages to inputs and outputs of the main circuit.
4. As the next step, save the subcircuit and generate a KiCad netlist for the same.
5. Now, to use this as a subcircuit, create a block in the KiCad Eescchema and follow the below steps:
   i. Go to library viewer of Eschema
   ii. Choose the current working library as the eSim_Subckt. iii. Click on create a new component with reference X.
   iv. Start drawing the subcircuit block. Update and save it
6. Close the Eeschema window and click on Convert KiCad to Ngspice button in subcircuit builder tool. This will convert the KiCad spice netlist to Ngspice netlist.

   And it will save your subcircuit into eSim repository, which you can add in your main circuit.

Chapter 4

Subcircuits Implemented

There are total seven Integrated Circuits(IC) implemented by me as subcircuits on eSim. Following are the schematic and description of each of them.

4.1 INA823 IC

Developed by Texas Instruments, the INA823 is an integrated instrumentation amplifier that offers low power consumption and operates over a wide, single-supply or dualsupply range. A single external resistor sets any gain from 1 to 10,000. The device provides low input offset voltage, low offset voltage drift, low input bias current, and low current noise while remaining cost-effective. Additional circuitry protects the inputs against overvoltage up to ±60 V.

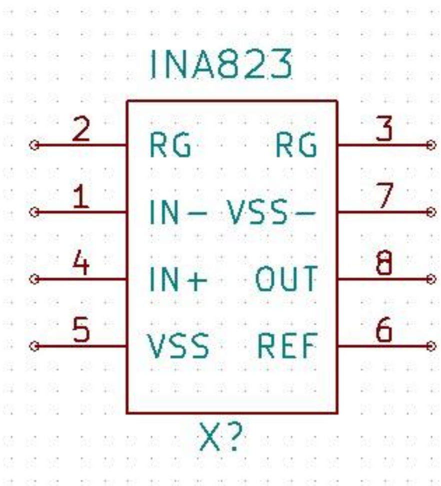
Some of the common applications of INA823 are:

1. Surgical equipment
2. Flow transmitter
3. Process Analytics

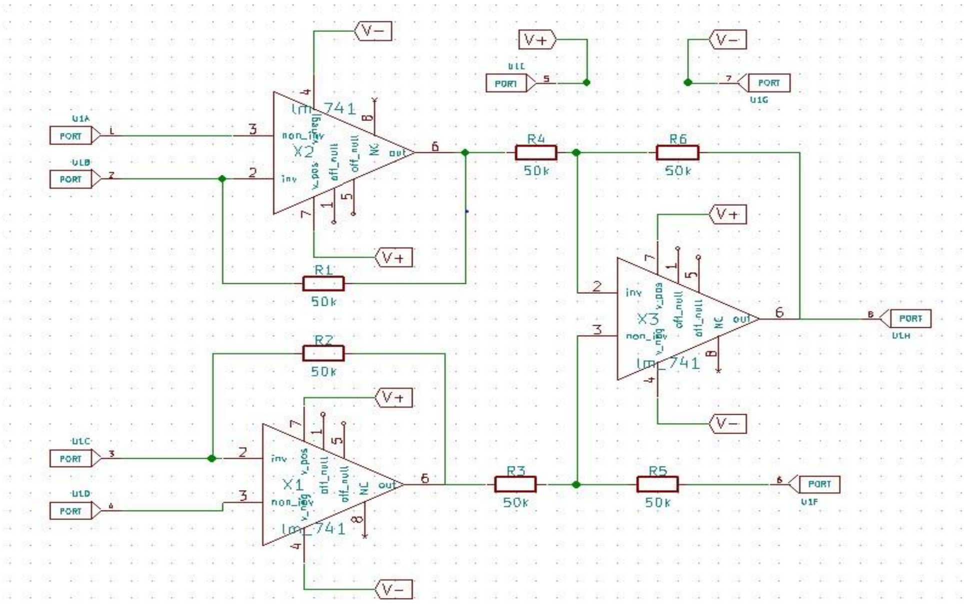
Some of the salient features of INA823 are:

1. Very low input voltage noise 2.
Input overvoltage protection
3. Low input bias current etc.

4.1.1 Pin Diagram



4.1.2 Subcircuit Schematic



4.2 LOG101 IC

The LOG101, manufactured by Texas Instruments is a versatile integrated circuit that computes the logarithm or log ratio of an input current relative to a reference current. The LOG101 is tested over a wide dynamic range of input signals. In log ratio applications, a signal current can come from a photodiode, and a reference current from a resistor in series with a precision external reference. The output signal at VOUT is trimmed to 1V per decade of input current allowing seven decades of input current dynamic range.

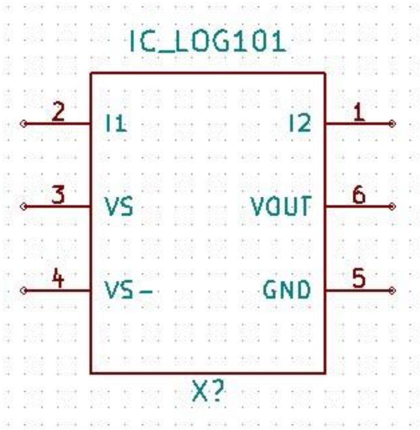
Some of the common applications of LOG101 are:

1. Photodiode signal compression amps
2. Analog signal compression with ADCs
3. Log Ratio Computation for analytical, industrial or communication purposes.

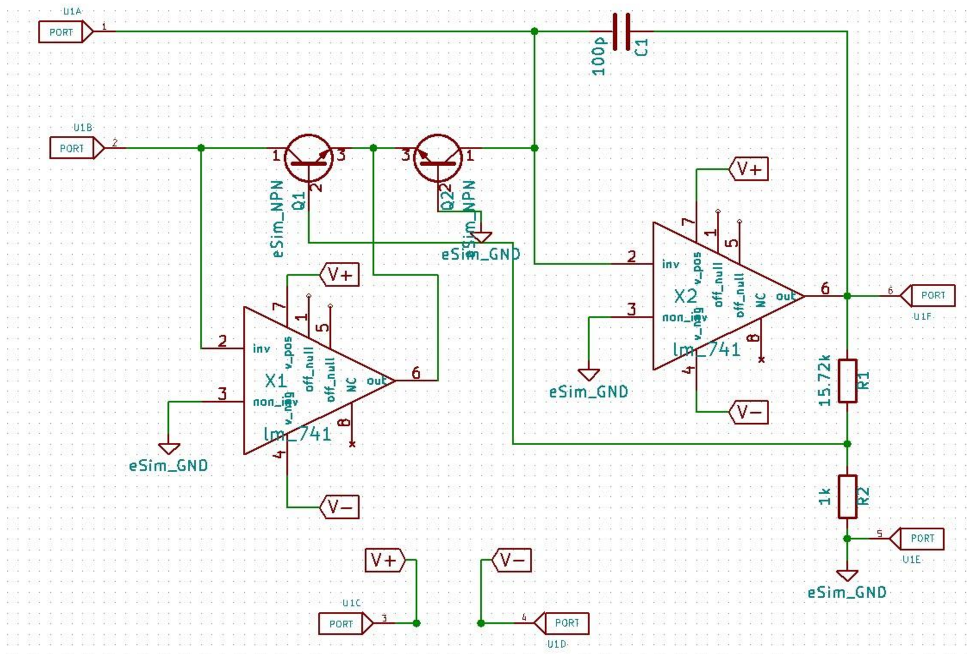
Some of the salient features of LOG101 are:

1. High Accuracy
2. Wide input dynamic range
3. Low Quiescent Current, around 1mV.

4.2.1 Pin Diagram



4.2.2 Subcircuit Schematic



4.3 OPA827 IC

The OPA827 series of JFET operational amplifiers combine outstanding DC precision with excellent AC performance. These amplifiers offer low offset voltage (150 µV, maximum), very low drift over temperature (0.5 µV/°C, typical), low-bias current (3 pA, typical), and very low 0.1-Hz to 10-Hz noise (250 nV_{PP}, typical).

Some of the common applications of OPA827 are:

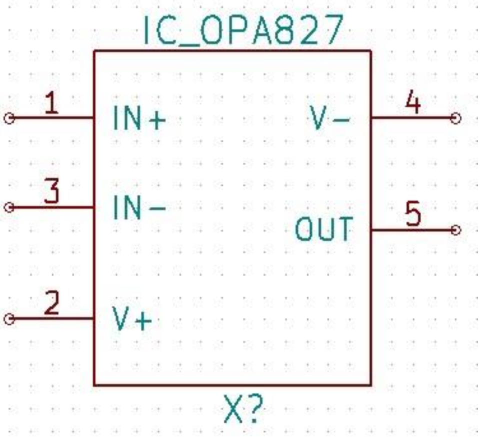
1. DAC Output Buffers
2. ADC Drivers

3. PLL Filters

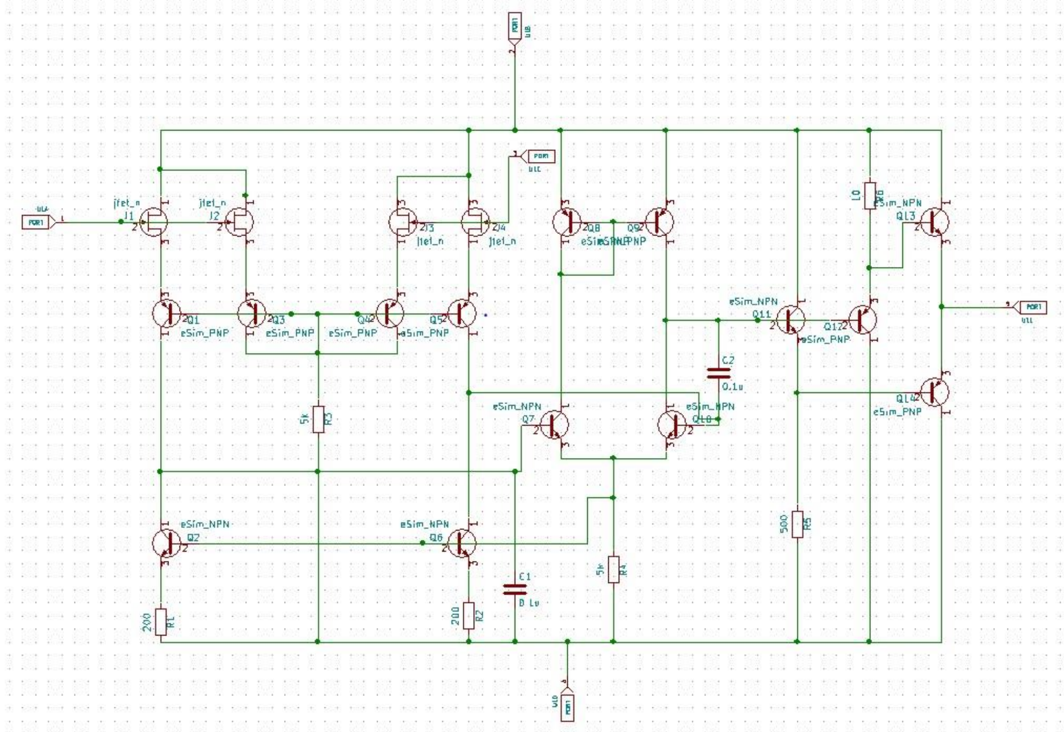
Some of the salient features of OPA827 are:

4. Very low input voltage noise density
5. Low input offset voltage
6. Moderately high GBW

4.3.1 Pin Diagram



4.3.2 Subcircuit Schematic



.

4.4 OPA862 IC

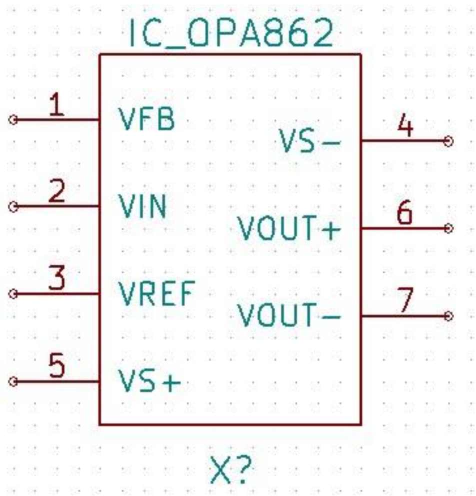
The OPA862 is a single-ended to differential analogto-digital converter (ADC) driver with high input impedance for directly interfacing with sensors. The device only consumes 3.1-mA quiescent current for an output-referred noise density of 8.3 nV/√ Hz in a gain of 2-V/V configuration. A fully differential amplifier configured in a gain of 1 V/V with 1-kΩ resistors must be less than 1 nV/√ Hz to achieve the OPA862 equivalent output-referred noise density of 8.3 nV/√ Hz. Some of the common applications of OPA862 are:

1. Data Acquisition
2. Test and Measurement
3. Medical Instrumentation

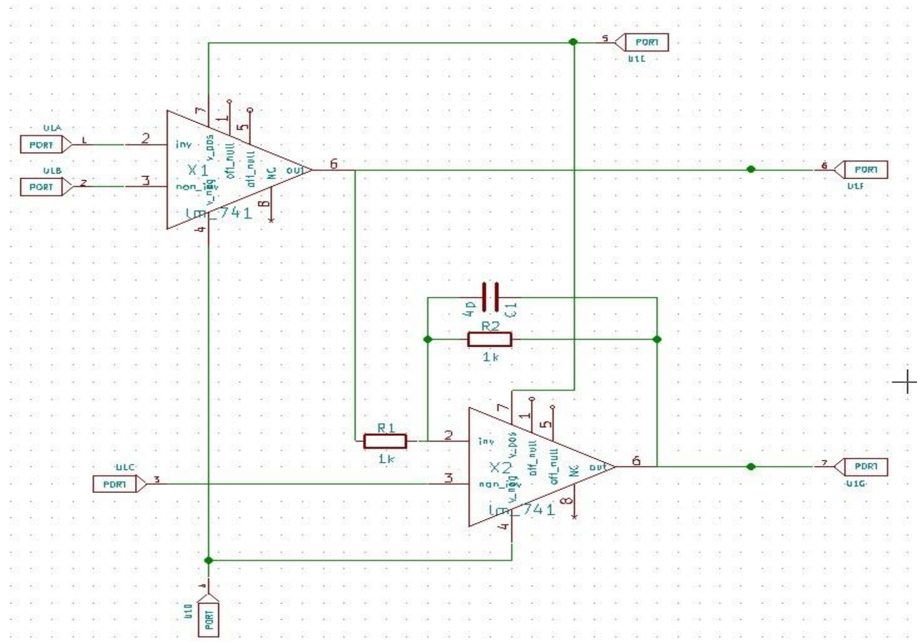
Some of the salient features of OPA862 are:

1. Very high GBW
2. High input impedance 3. Wide Supply Range etc.

4.4.1 Pin Diagram



4.4.2 Subcircuit Schematic



4.5 ULN2803A IC

The ULN2803A is a high-voltage, high-current Darlington transistor array. The device consists of eight npn Darlington pairs that feature high-voltage outputs with commoncathode clamp diodes for switching inductive loads. The collector-current rating of each Darlington pair is 500 mA. The Darlington pairs may be connected in parallel for higher current capability.

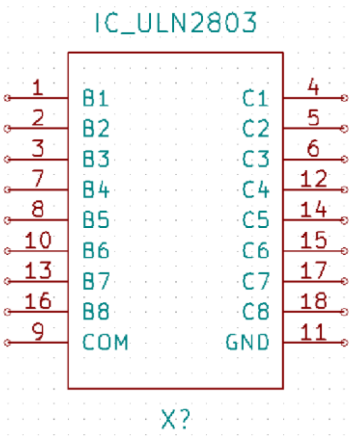
Some of the common applications of ULN2803A are:

1. Hammer drivers
2. Relay drivers
4. Lamp drivers

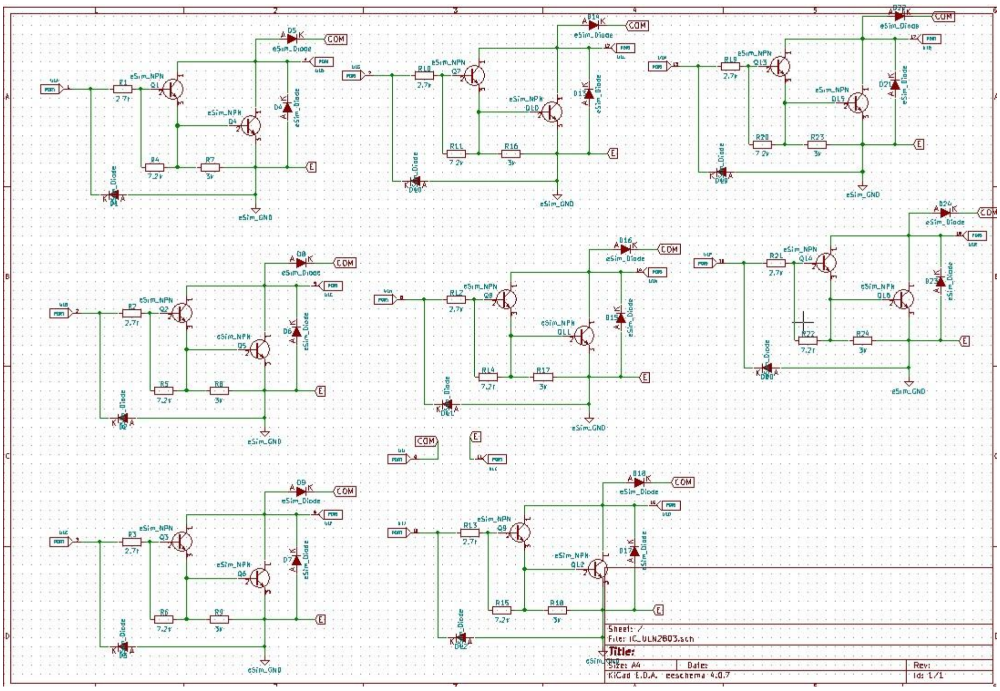
Some of the salient features of ULN2803A are:

1. 500-mA Rated Collector Current (Single Output)
2. High voltage output
7. Inputs compatible with various types of logic.

4.5.1 Pin Diagram



4.5.2 Subcircuit Schematic



4.6 LM397 IC

The LM397 device is a single voltage comparator with an input common mode that includes ground. The LM397 is designed to operate from a single 5-V to 30-V power supply or a split power supply. Its low supply current is virtually independent of the magnitude of the supply voltage.

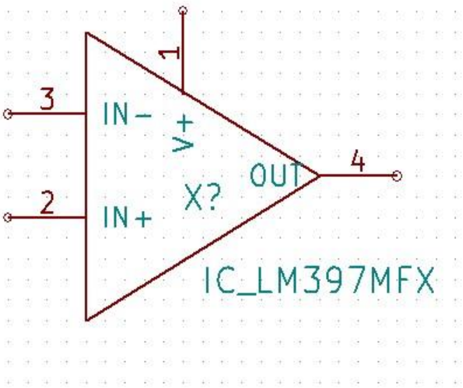
Some of the common applications of LM397 are:

1. A/D Converters
2. Peak Detectors

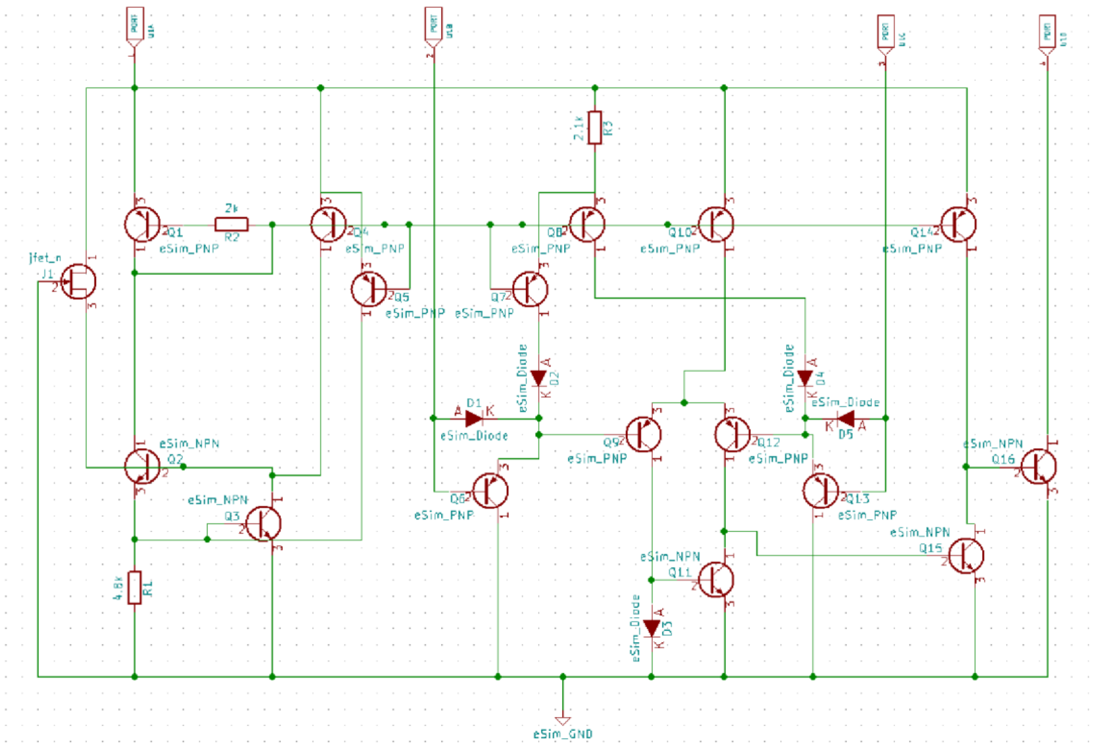
Some of the salient features of LM397 are:

7. Low input bias current
8. Low input offset voltage 8. Low input offset current etc.

4.6.1 Pin Diagram



4.6.2 Subcircuit Schematic



4.7 LM386 IC

The LM386M-1 and LM386MX-1 are power amplifiers designed for use in low voltage consumer applications. The gain is internally set to 20 to keep external part count low, but the addition of an external resistor and capacitor between pins 1 and 8 will increase the gain to any value from 20 to 200.

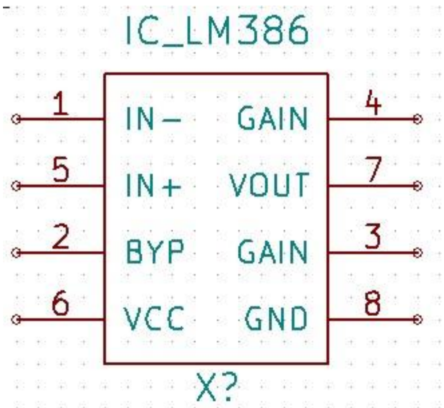
Some of the common applications of LM386 are:

1. AM-FM Radio Amplifiers
2. Ultrasonic drivers
3. Power Converters

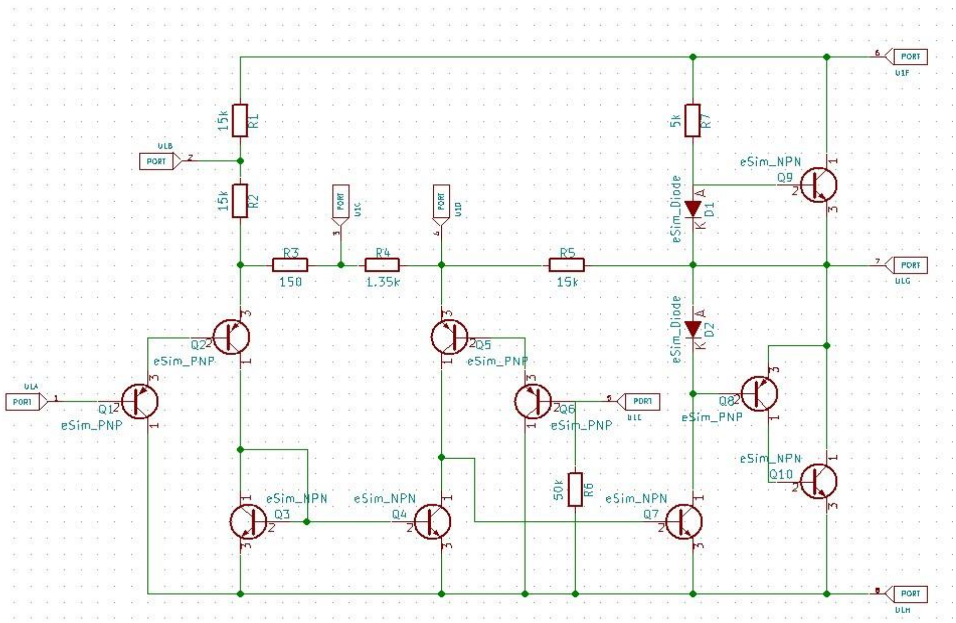
Some of the salient features of LM386 are:

1. Wide Supply Voltage Range
2. Minimum External Parts
3. Ground Referenced Input

4.7.1 Pin Diagram



4.7.2 Subcircuit Schematic



Chapter 5

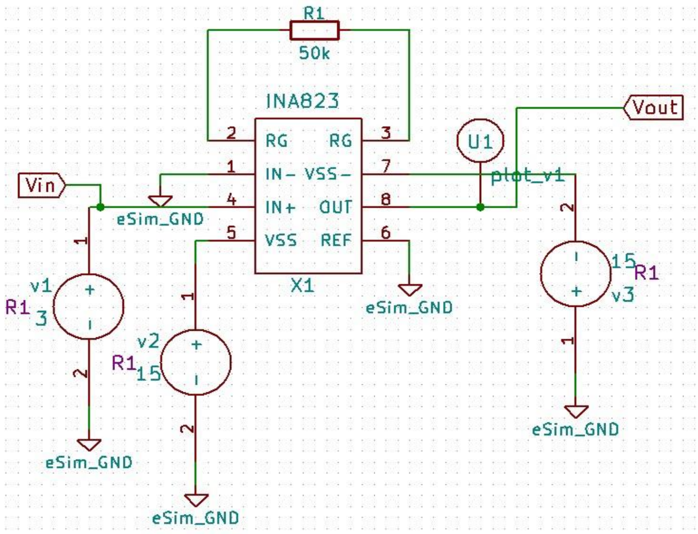
NgSpice Simulations of Subcircuits Tested

For each of the subcircuits, an NgSpice simulation is performed for a test circuit using the IC subcircuit and the results are verified in accordance with the equations and transfer functions in the datasheet. In the following section, I describe the nature of the test circuit and the waveforms obtained as a result of the simulations performed. For the sake of clarity, the waveforms attached are taken from python plots.

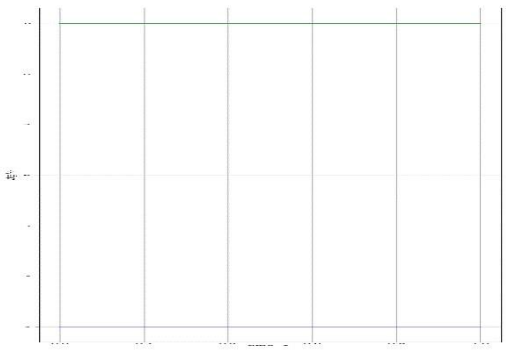
5.1 INA823

INA823 is tested as a simple amplifier configuration. The gain of the device is given by: Gain = 1 + (100/RG). In the test circuit, RG = 50k ohms ➐ Gain = 3 ➐ Vout = 3*Vin = 9V

5.1.1 Test Schematic



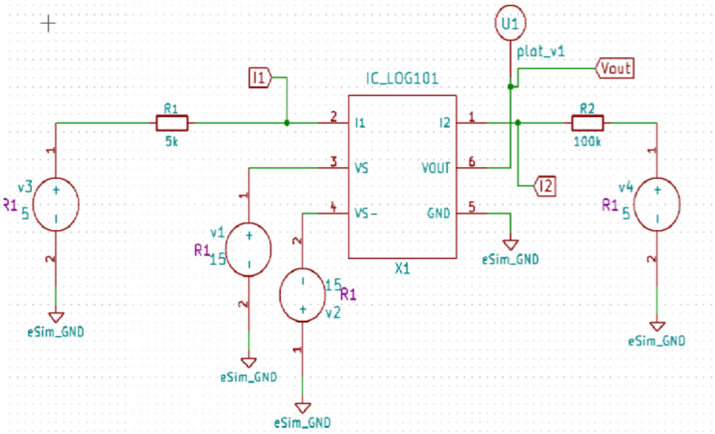
5.1.2 Waveforms



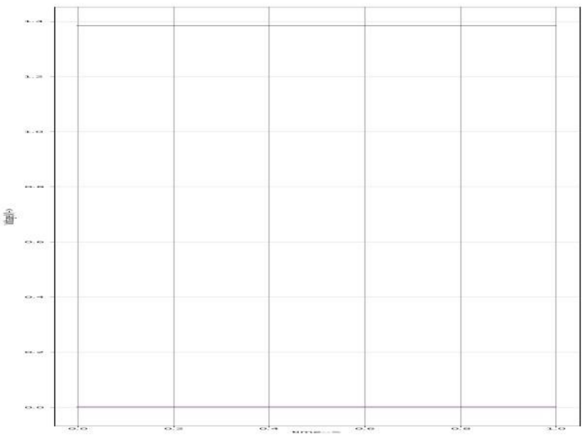
5.2 LOG101

LOG101 is tested as a current log ratio amplifier configuration. The output voltage is given by V0 = (1V) log(I1/I2). Here, I1 = 1mA and I2 = 10uA ➐ V0 = 2V (Approx.)

5.2.1 Test Schematic



5.2.2 Waveforms

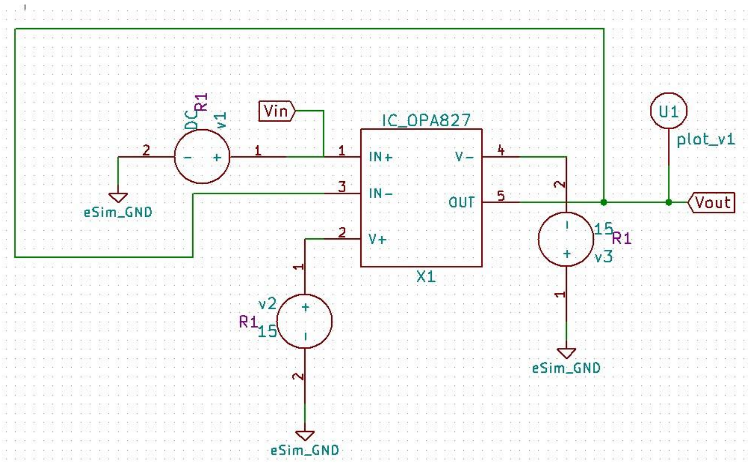


5.3 OPA827

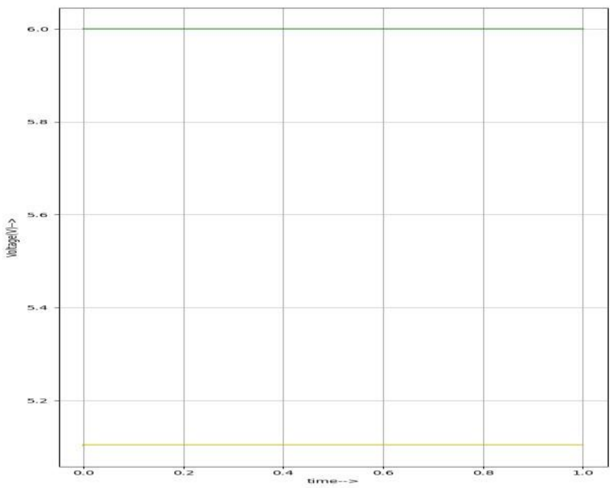
OPA827 is tested as a non-inverting buffer circuit in this case.

Here, Vin = 6V ➐ Vout = 6V.

5.3.1 Test Schematic



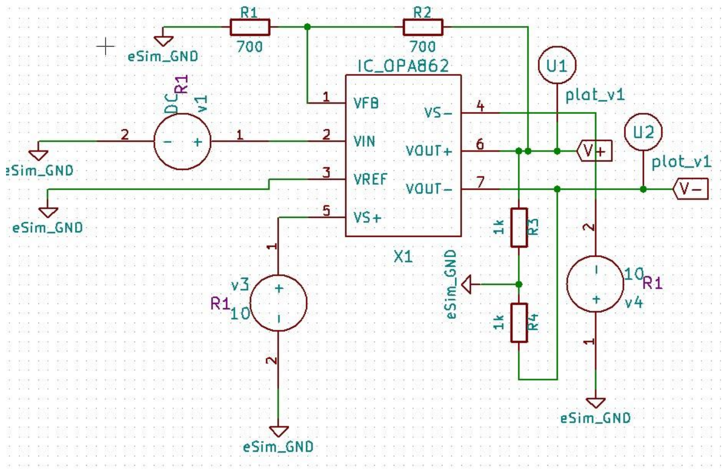
5.3.2 Waveforms



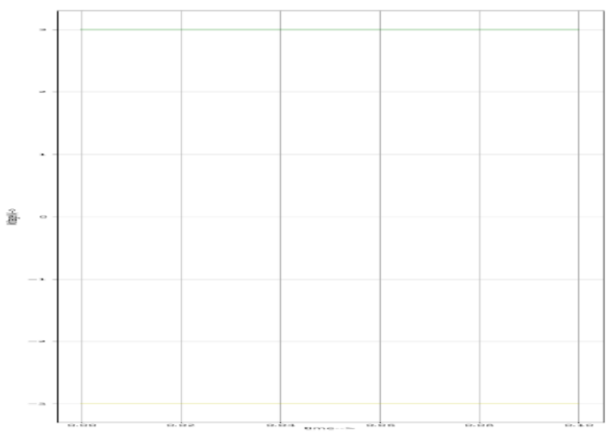
5.4 OPA862

OPA862 is a dual output differential amplifier and the output is tested as the difference of V+ and V-. The output is given as Vout = V+ - V- = 2*Vin*(1+ Rf/Rg) - (2*Vref). Here, Vref = 0V, Rf = Rg , Vin = 1.5V ❼ Vout = 6V

5.4.1 Test Schematic



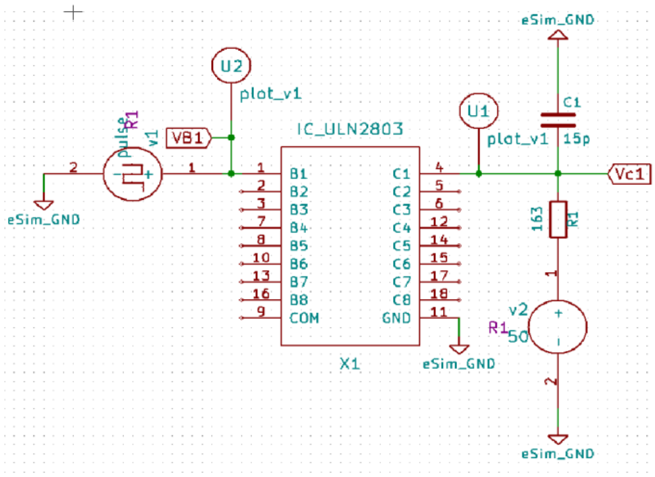
5.4.2 Waveforms



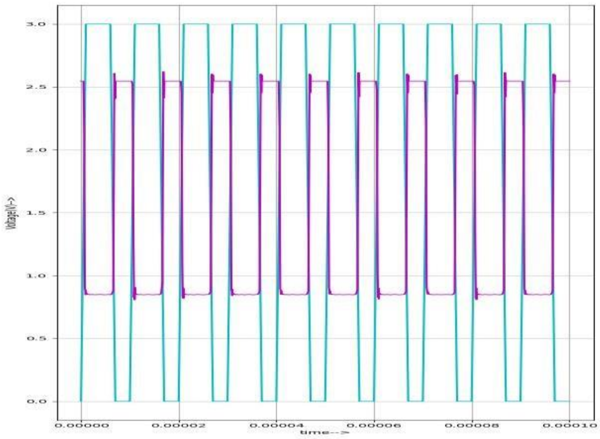
5.5 ULN2803A

ULN2803A is tested as a circuit from the datasheet that performs as an inverter.

5.5.1 Test Schematic



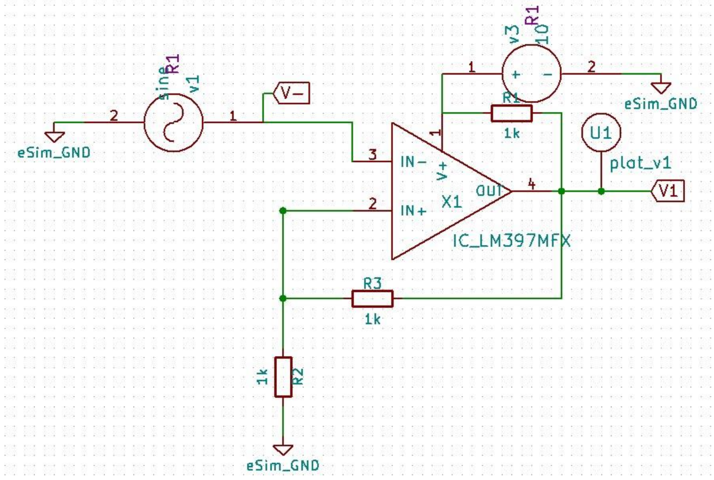
5.5.2 Waveforms



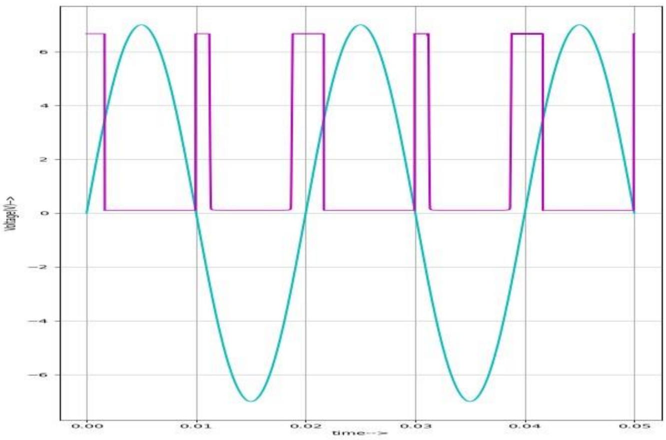
5.5 LM397

LM397 IC is tested in a Schmitt trigger configuration with a sinusoidal input at the inverting terminal and as a result, we get a pulse wave at the output. When non inverting voltage is greater than inverting voltage, we get a logic high and logic low when vice-versa.

5.6.1 Test Schematic



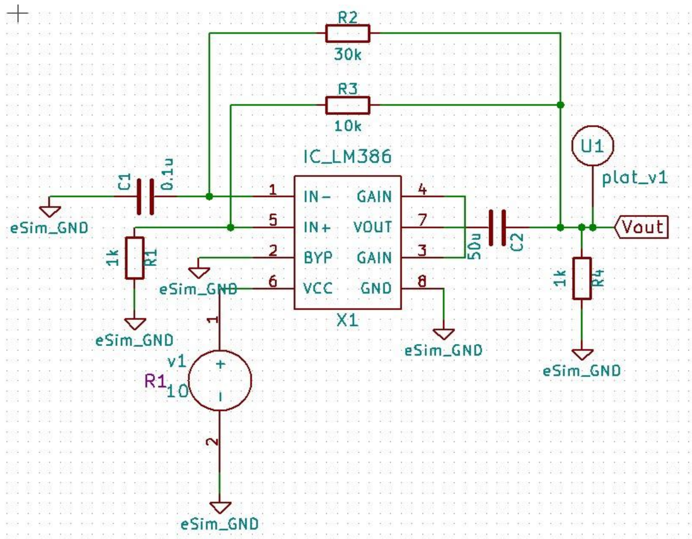
5.6.2 Waveforms



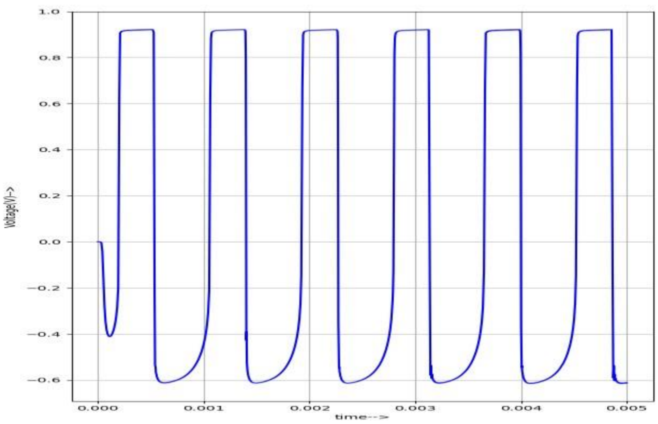
5.6 LM386

LM386 IC is an audio power amplifier circuit tested as a square wave oscillator in this case. The frequency of oscillation is tuned to be 1kHz.

5.7.1 Test Schematic



5.7.2 Waveforms



Bibliography

[1] INA823 Datasheet https://www.ti.com/lit/ds/symlink/ina823.pdf?ts=1675887066862&ref_url=https%253A%252F%252Fwww.ti.com%252Famplifiercircuit%252Finstrumentation%252Fproducts.html

[2] LOG101 Datasheet https://www.ti.com/lit/ds/symlink/log101.pdf?ts=1676301524134&ref_url=https%253A%252F%252Fwww.ti.com%252Famplifier-circuit%252Fspecialfunction%252Flogarithmic%252Fproducts.html

[3] OPA827 Datasheet https://www.ti.com/lit/ds/symlink/opa827.pdf?ts=1676380164910&ref_url=https%253A%252F%252Fwww.google.com%252F

[4] OPA862 Datasheet https://www.ti.com/lit/ds/symlink/opa862.pdf?ts=1678101969346&ref_url=https%253A%252F%252Fwww.ti.com%252Famplifier-circuit%252Ffullydifferential%252Fproducts.html

[5] ULN2803A Datasheet https://www.electroschematics.com/wp-content/uploads/2013/07/uln2803a-datasheet.pdf

[6] LM397 Datasheet https://www.ti.com/lit/ds/symlink/lm397.pdf?ts=1679980519407&ref_url=https%253A%252F%252Fwww.google.co.in%252F

[7] LM386 Datasheet https://www.ti.com/lit/ds/symlink/lm386.pdf

[8] FOSSEE eSim Manual https://esim.fossee.in/resource/book/esimusermanual.pdf