Design and Implementation of Instrumentation Amplifier at Nanoscale

eSim Research Migration Project

Ganderla Chaithanya
Rajiv Gandhi University of Knowledge Technologies, Nuzvid

Abstract

The instrumentation amplifier is a key component in analog signal processing, primarily used to amplify low-level differential signals in the presence of high common-mode noise. This project presents the design and implementation of a high-precision, three-op-amp-based instrumentation amplifier suitable for biomedical and sensor applications. The amplifier offers high input impedance, excellent common-mode rejection ratio (CMRR), and accurate gain control through a single resistor. Implemented using simulation tools such as eSim and Ngspice, the circuit demonstrates consistent linear amplification and stable performance across varying input conditions. The results validate the amplifier's suitability for precise data acquisition systems requiring low-noise, high-gain signal conditioning.

1 Introduction

Instrumentation amplifiers are specialized differential amplifiers designed to offer precise, low-noise amplification of small signals, often in the presence of large common-mode voltages. These amplifiers are widely used in applications such as biomedical signal acquisition, industrial process control, and sensor interfacing, where signal integrity and accuracy are critical.

2 Theory of Operation

An instrumentation amplifier typically consists of three operational amplifiers (Op-Amps) arranged in a two-stage configuration:

- The first stage consists of two buffer amplifiers (Op-Amp 1 and Op-Amp with high input impedance to prevent loading the signal source.
- The second stage is a differential amplifier (Op-Amp 3) that subtracts the signals from the two inputs and provides the final amplified output.

2.1 Circuit Diagram

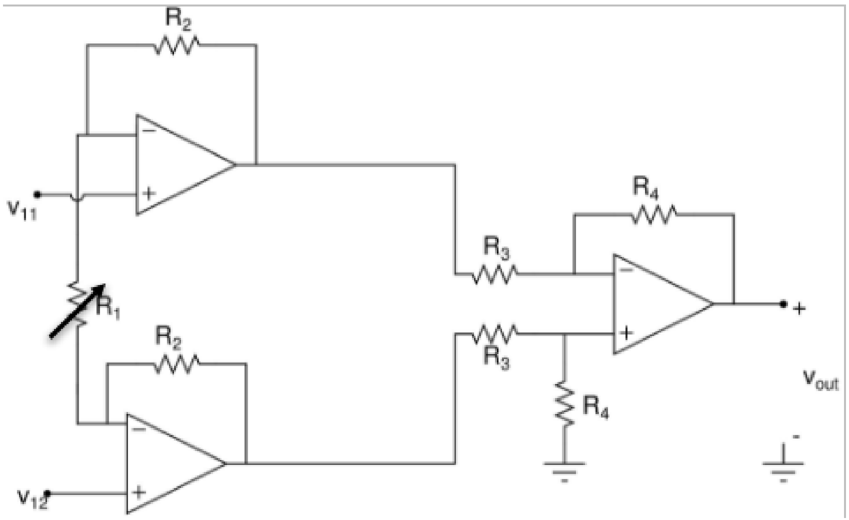

Figure 1: Instrumentation Amplifier

2.2 Circuit Schematic in eSim

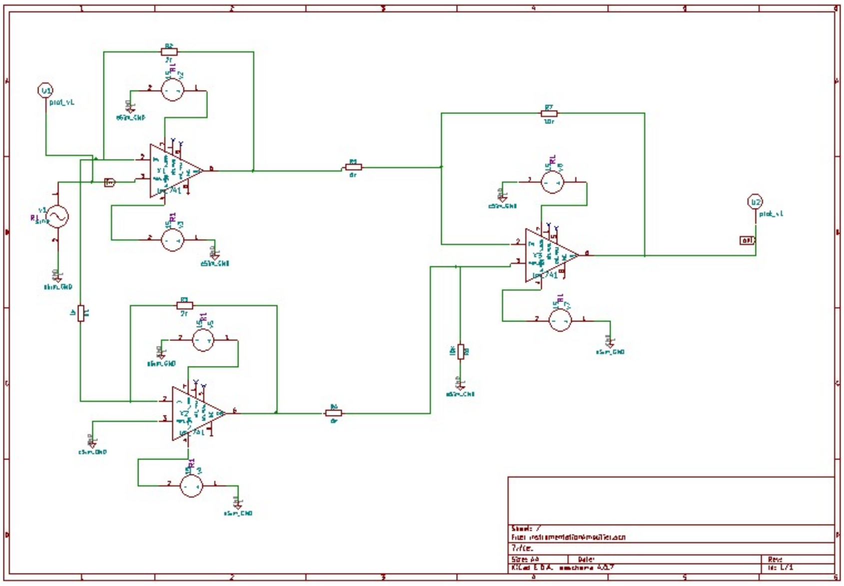

Figure 2: Instrumentation Amplifier Schematic in eSim

2.3 Analysis and Results of Flash ADC in Ngspice

The designed 3-bit Flash ADC was simulated in Ngspice, an open-source mixed-signal circuit simulator, to analyze its performance. The key aspects of the analysis includes transient simulations to verify the correct operation of the ADC under different input conditions.

3 Ngspice Simulation

Ngspice was used to simulate the behavior of the designed instrumentation amplifier circuit to verify its functionality and performance.

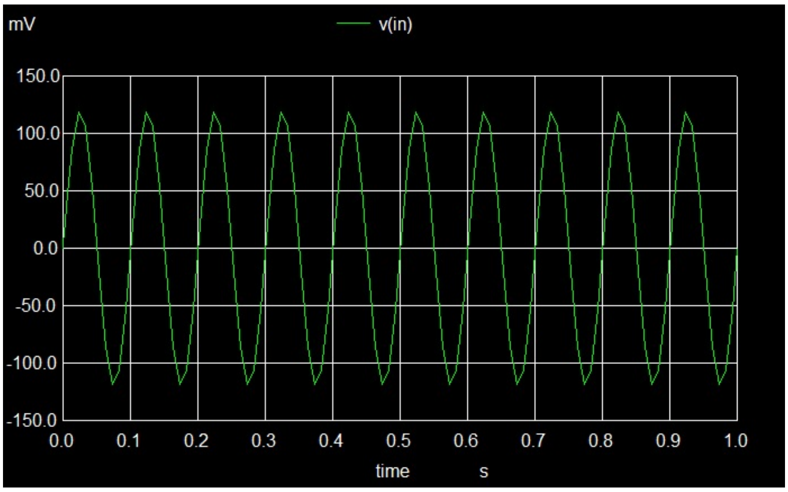

Figure 3: Input V_{in} in Ngspice

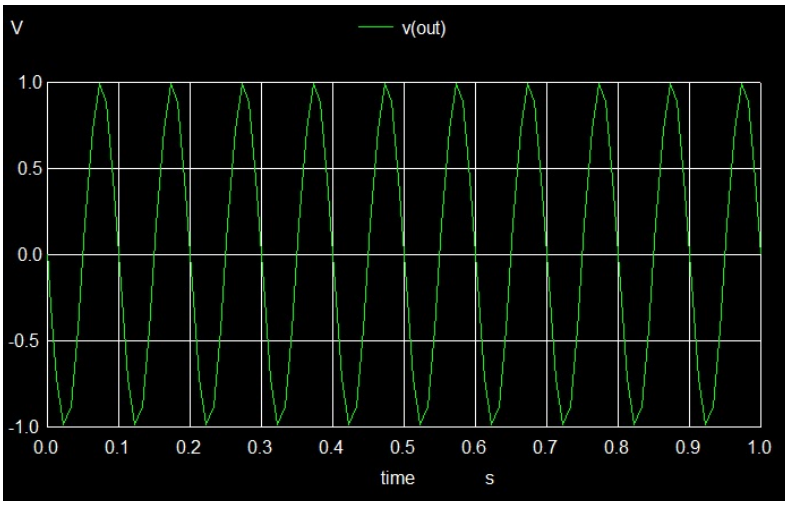

Figure 4: Output from Ngspice

4 Python plots

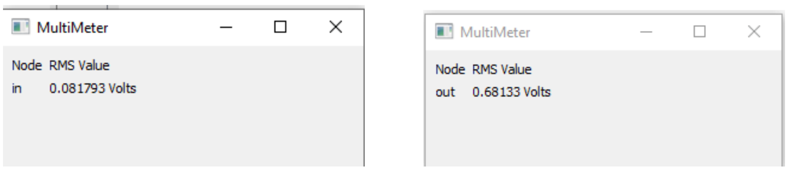

Figure 5: Multimeter Readings from Python Plot

5 Conclusion

The design and simulation of the instrumentation amplifier have successfully demonstrated its ability to accurately amplify low-level differential signals while effectively rejecting common-mode noise. The three-op-amp configuration provided high input impedance, stable gain control, and excellent linearity, making it suitable for precision signal processing applications such as sensor interfacing and biomedical instrumentation.

References

[1] Dr.M.Nizamuddin, Design and Implementation of Instrumentation Amplifier at Nanoscale 2017, Available at : https://www.ijariit.com/manuscript/design-performance-analysis-instrumentation-amplifier-nanoscale/)