

TITLE : Design and Simulation of a CMOS Bandgap Voltage Reference

STUDENT NAME : MUHIL.T.A

COLLEGE NAME : Sri Eshwar College of Engineering

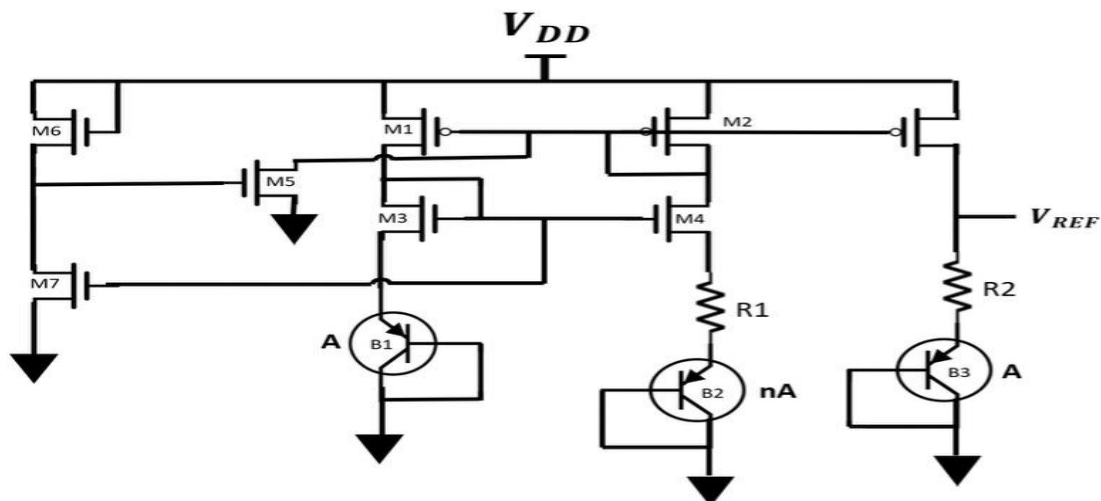
Problem Statement :

The project focuses on the design and simulation of a CMOS-based bandgap reference circuit using eSim (NgSpice). The bandgap reference is a fundamental building block in analog and mixed-signal integrated circuits, providing a stable reference voltage independent of supply voltage, temperature variations, and process changes. The aim is to study the working principle, simulate the circuit behavior, and verify the temperature stability of the output reference voltage.

Abstract :

A bandgap voltage reference is an essential circuit used in analog, digital, and mixed-signal ICs for providing a temperature-independent and supply-insensitive reference voltage. In this project, a CMOS-based bandgap reference circuit is designed and simulated using eSim (NgSpice) with SKY130 device models. The circuit combines the complementary-to-absolute-temperature (CTAT) and proportional-to-absolute-temperature (PTAT) voltages to achieve a stable reference of approximately 1.2 V. The performance is analyzed through DC sweep, transient analysis, and temperature variation from -40°C to $+125^{\circ}\text{C}$. The results demonstrate that the proposed circuit achieves a stable voltage reference, making it suitable for low-power and precision applications such as ADCs, DACs, and voltage regulators.

Proposed Circuit :



References :

1. R. J. Widlar, "New Developments in IC Voltage Regulators," IEEE Journal of Solid-State Circuits, vol. 6, no. 1, pp. 2–7, Feb. 1971.
2. B. Razavi, Design of Analog CMOS Integrated Circuits, McGraw-Hill, 2001.
3. P. E. Allen and D. R. Holberg, CMOS Analog Circuit Design, 3rd Edition, Oxford University Press, 2012.