TITLE: Noise Analysis for Electronic Circuit simulation using eSim.

STUDENT NAME: Anjali Papansingh Thakur.

MENTOR NAME: Prof. S.M. Gudhe.

COLLEGE: Shri Guru Gobind Singhji Institute Of Engineering And Technology, Nanded.

PROBLEM STATEMENT:

Noise in electronic circuits poses a significant challenge, impacting the performance and dependability of devices used in fields like communications, medical technology, and precision measurement. This noise stems from various sources, including thermal fluctuations, electromagnetic interference, and the inherent properties of semiconductor materials, which can cause signal distortion and data errors.

This project aims to use Multisim, a versatile circuit simulation software, to perform an in-depth analysis of noise in electronic circuits. By simulating different circuit components, such as amplifiers and filters, the project will pinpoint major noise sources and evaluate their effects on circuit behavior. The goal is to develop design strategies that effectively reduce noise, thereby enhancing the performance and reliability of electronic systems in practical applications.

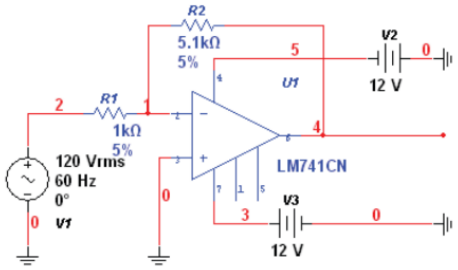

Figure 1. the noise analysis circuit

-

TITLE OF THE PAPER: Noise Analysis for Electronic Circuit Using Multisim

REFERENCE:

[1] P. R. Gray and R. G. Meyer Analysis and Design of Analog Integrated circuit 3rd Edition, John Wiley & Sons, New York, 1993.

[2] P. L. Jones P. J. Spreadbury: Analogue electronic circuit and systems, Cambridge University Press, New York, 1991.