

NOISE ANALYSIS CIRCUIT SIMULATION USING eSim

Anjali Papansingh Thakur

Department Of Electrical Engineering

Shri Guru Gobind Singhji Institute of Engineering and Technology, at Nanded

ABSTRACT

Noise is a form of electrical or electromagnetic energy that can degrade signal quality, impacting digital, analog, and communication systems. This paper presents a noise analysis method using Multisim, a simulation tool that models circuit noise. Instead of relying on traditional AC models, Multisim utilizes noise models for each resistor and semiconductor device to perform the analysis. The tool calculates the noise contribution from each component at a specified output node, providing detailed insights into the circuit's noise characteristics.

Keywords - noise analysis; model; noise figure; setting parameters; Multisim

INTRODUCTION

In electronic systems, noise is a significant issue that limits the measurement accuracy and signal quality. It can severely impact the power, speed, and linearity of circuits. To address these challenges, analog circuit designers frequently analyse circuit noise and its effects on semiconductor devices to optimize circuit performance.

Multisim provides a variety of analysis methods that use simulation to generate data for the desired analyses. These methods range from basic to highly sophisticated and often involve performing one analysis as part of another automatically. Noise analysis and noise figure analysis using Multisim are straightforward and effective techniques for optimizing electronic systems.

Working Principal

Noise analysis in electronic circuits involves assessing the impact of unwanted electrical disturbances. Multisim uses noise models for components like resistors and transistors to simulate their behaviour, including thermal, shot, and flicker noise. This approach replaces traditional AC models to calculate the total noise at a specified output node, accounting for each component's contribution.

The analysis provides noise voltage or current values, enabling engineers to evaluate signal integrity. By identifying key noise sources, engineers can optimize component selection and circuit layout to reduce noise, ensuring reliable circuit performance.

- **Circuit schematic**

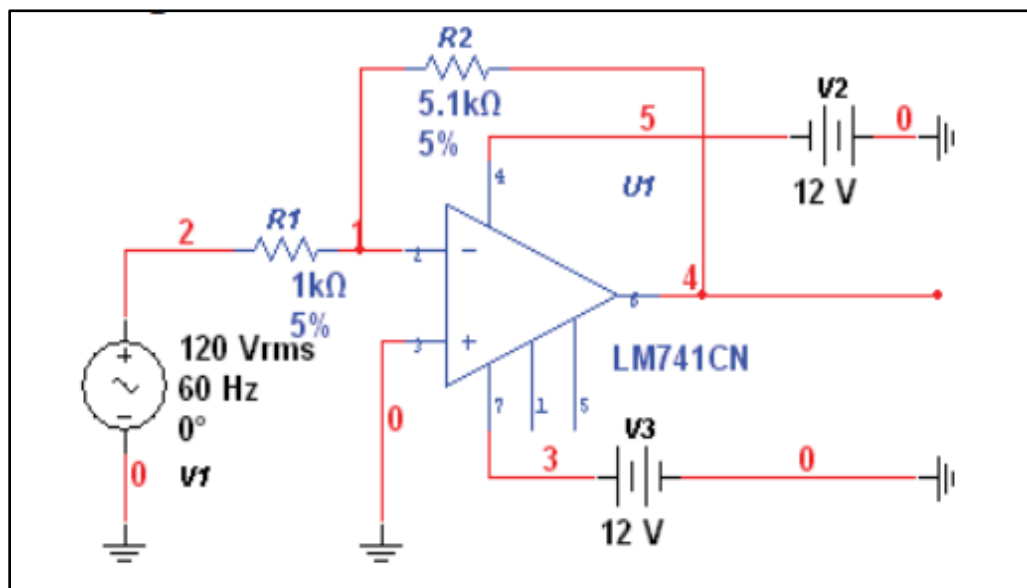


Figure 1: Noise analysis circuit

- **eSim Schematic**

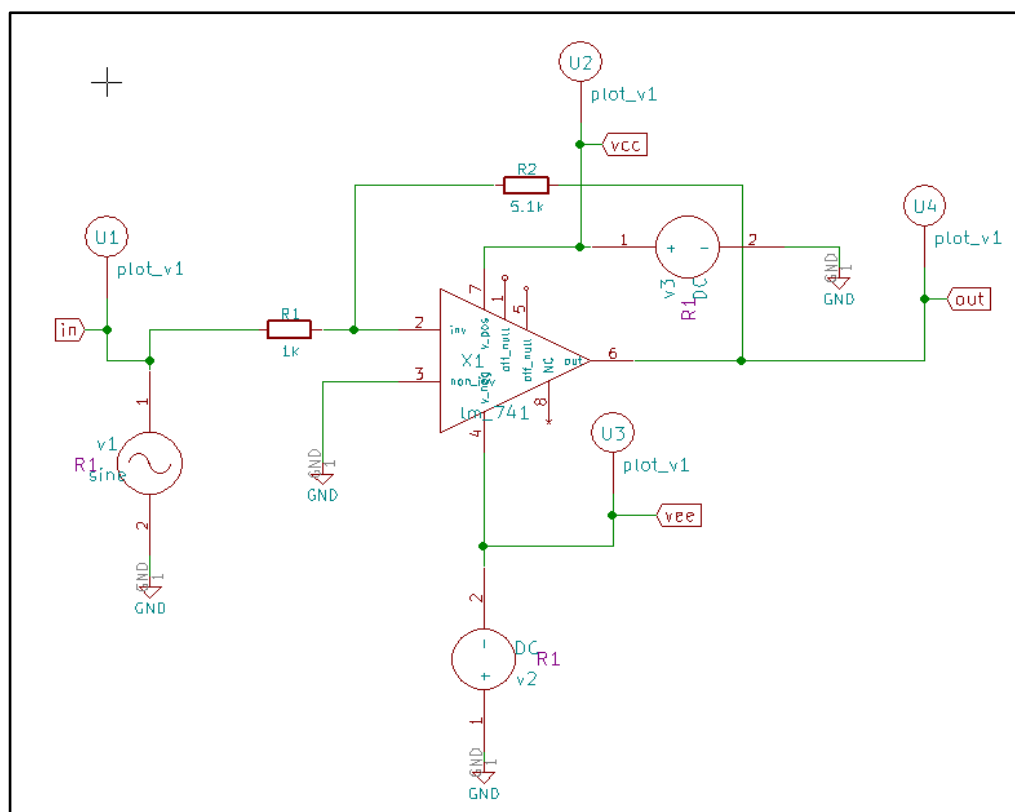


Figure 2: schematic of Noise analysis circuit

- **Input Waveform**

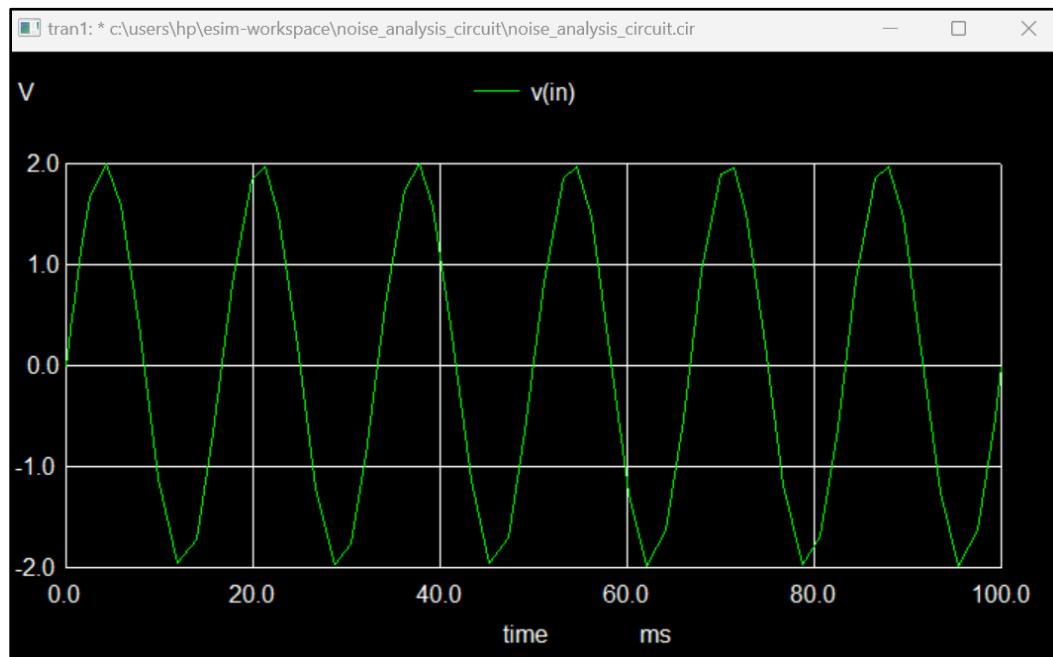


Figure 3: Input waveform- Noise analysis circuit

- **Output Waveform**

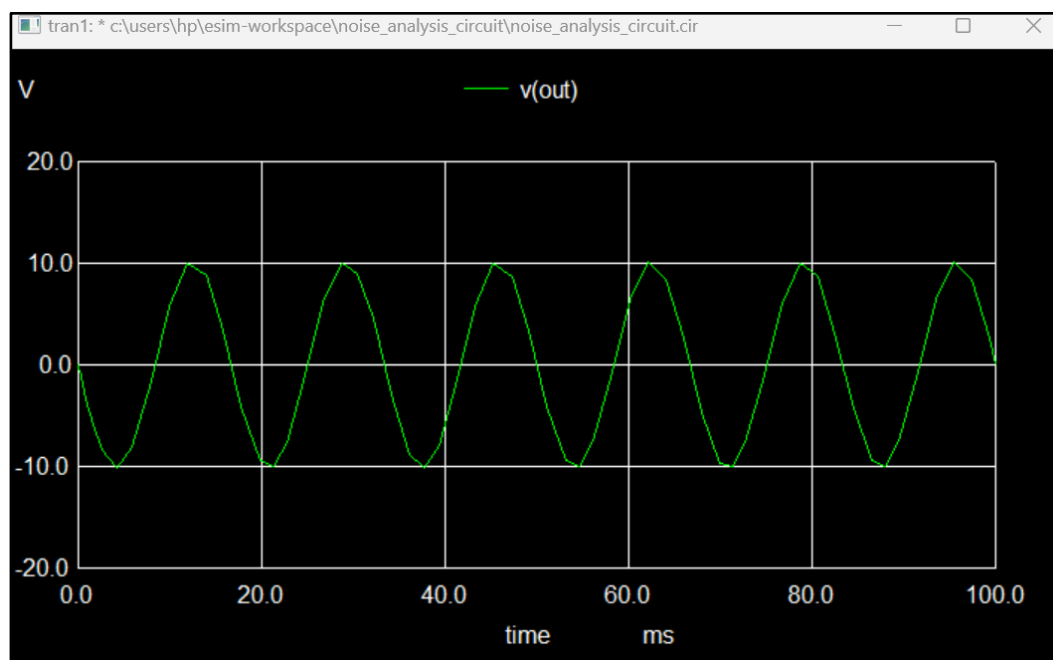


Figure 4: output waveform-Noise analysis circuit

Conclusion

The noise analysis method based on Multisim is practical and effective. The analysis results can provide some reference for circuit optimization for Analog circuit designers.

Noise analysis is crucial for improving electronic circuit performance by pinpointing and addressing unwanted disturbances. Multisim allows for the modelling and simulation of noise from different components to evaluate its effect on signal integrity. By targeting major sources of noise, engineers can boost the reliability and efficiency of circuits, ensuring they function effectively in practical applications.

References:

- [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.
- [3] National Instruments, Electronics Workbench Group. NI Multisim™ User Manual, 2008.
- [4] Interactive Image Technology Ltd. Multisim7 User Guide, Canada, 2003.
- [5] A. J. Peyton V. Walsh: Analogue electronics with Op Amps: a source book of practical, Cambridge University Press, New York, 1993