

West Virginia University LCSEE

EE 355L

Spring 2023

Section: 001

Date: 8/28/23

Lab 1: Intro To LTSpice

Submitted by:
Azain Uqaily

Objective

The objective of this lab is to introduce the operational amplifier. LM741 is the component that will be used a lot throughout the semester. For this lab, three different input signals will be amplified.

Procedure

This lab requires no physical circuit to be built, only LTSpice simulation. The first step is to create the op amp circuit. After the circuit is built, the input voltage is fed to the non inverting input of the op amp. Three different waves of amplitude 1v will be simulated. They are a sine wave, a triangular wave and a square wave. For triangular and square waves, the pulse input will be used.

Calculation

No calculations were performed for this lab.

Circuit Diagram(s)

The same circuit is used for all three signals, the only difference is the input voltage is changed and displayed to show parameters for V3.

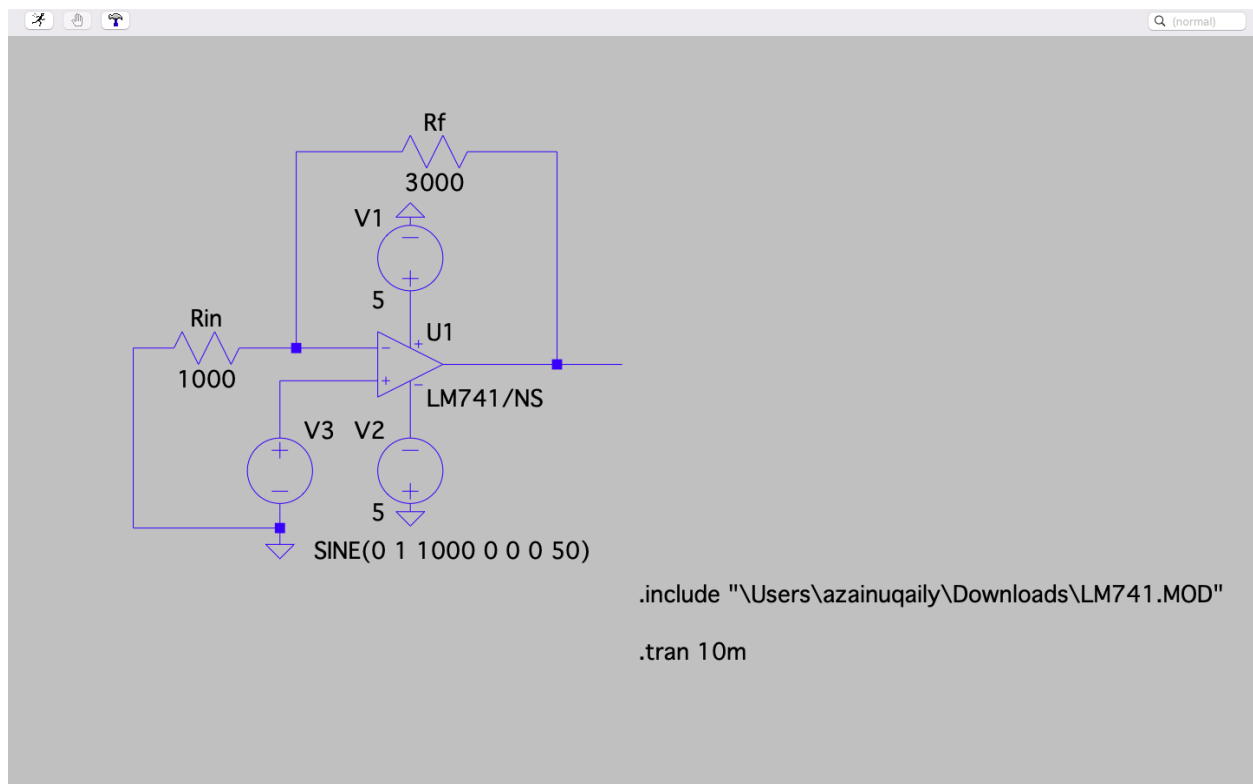


Figure 1: Sine Wave

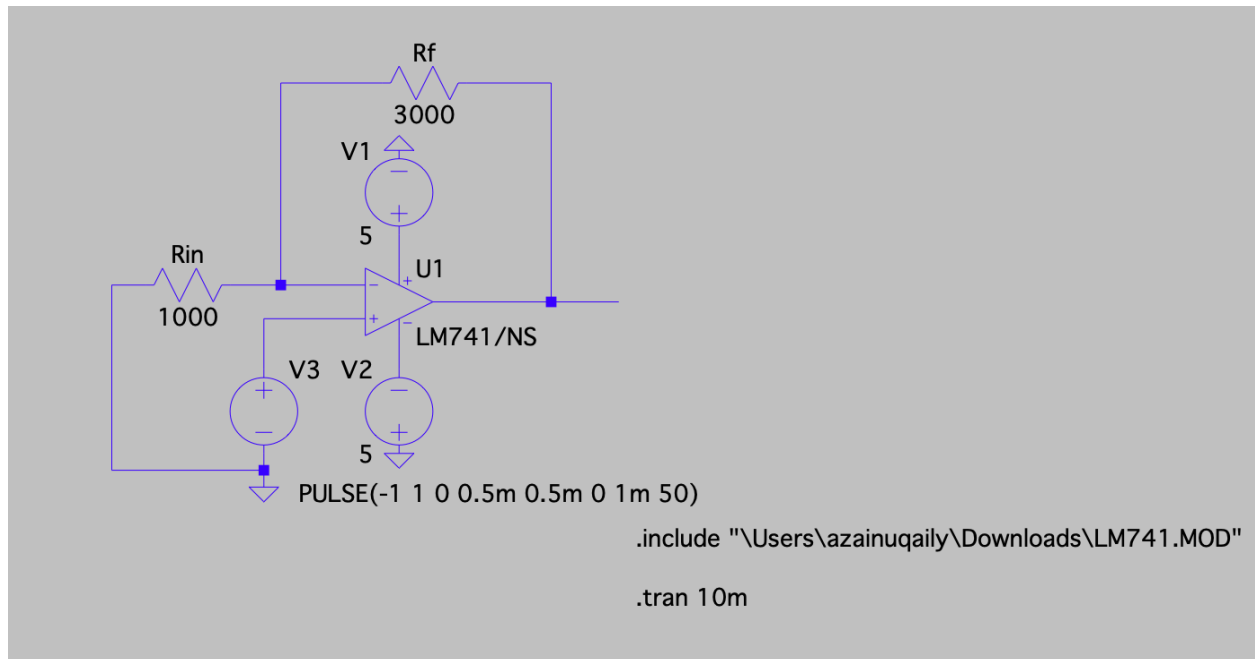


Figure 2: Triangular Wave

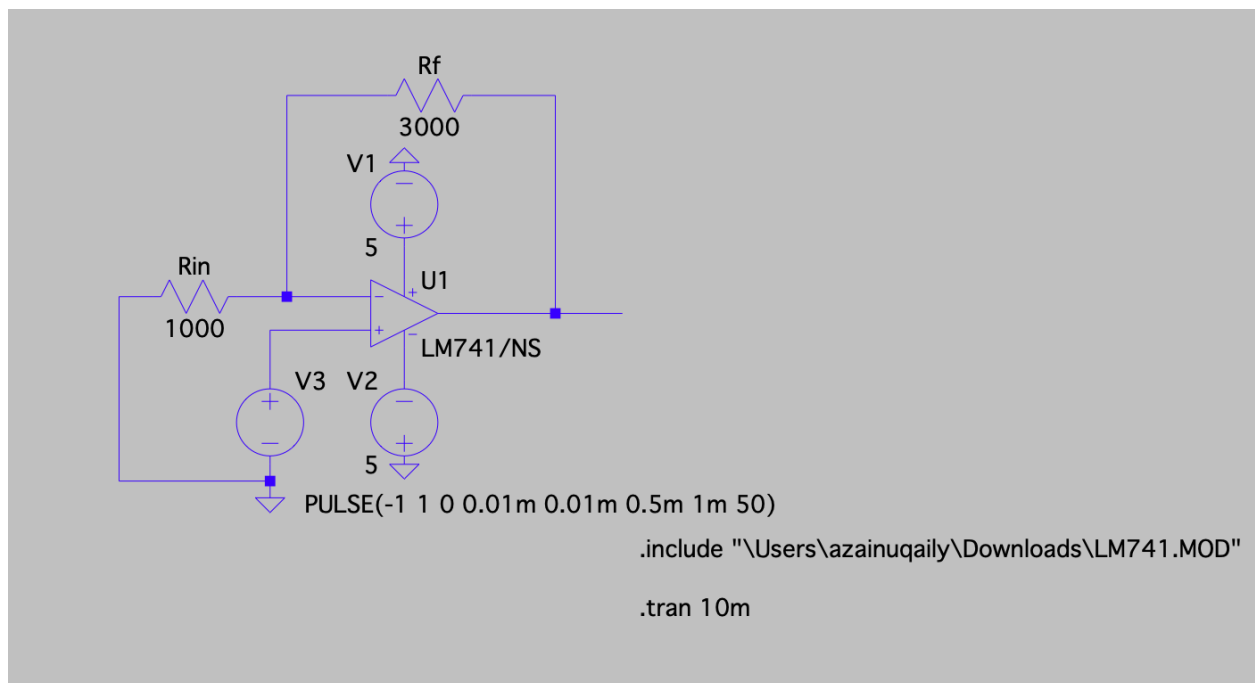


Figure 3: Square Wave

Results & Discussion (Values, Graph & Wave forms)

For each simulation, the green graph represents the input voltage, and the blue graph represents the output voltage.

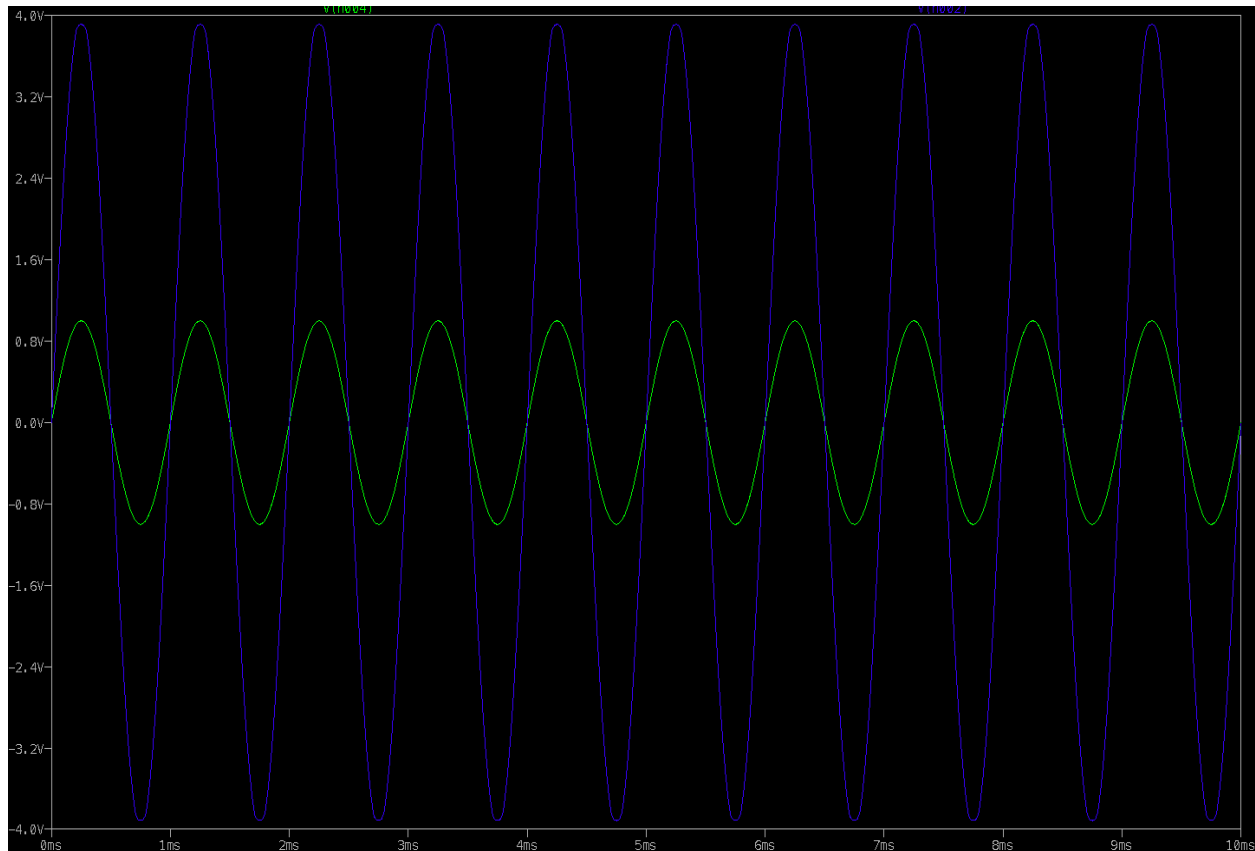


Figure 4: Sine Wave

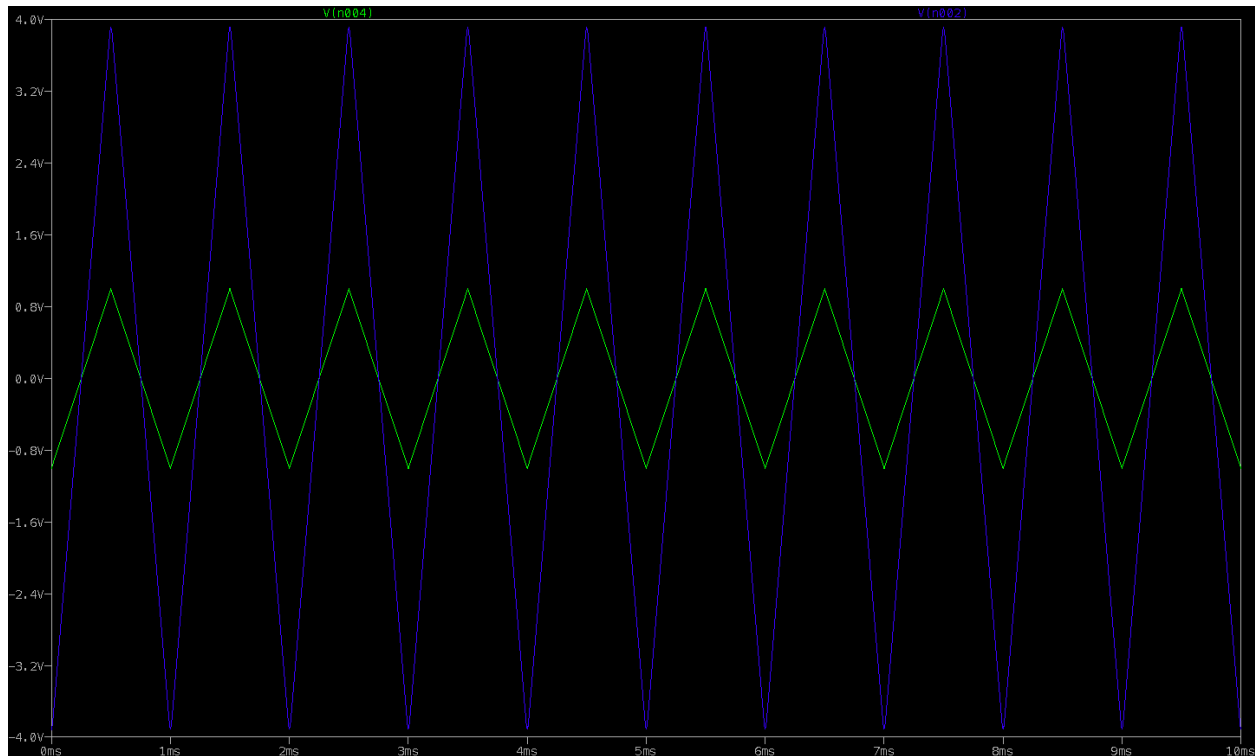


Figure 5: Triangular Wave

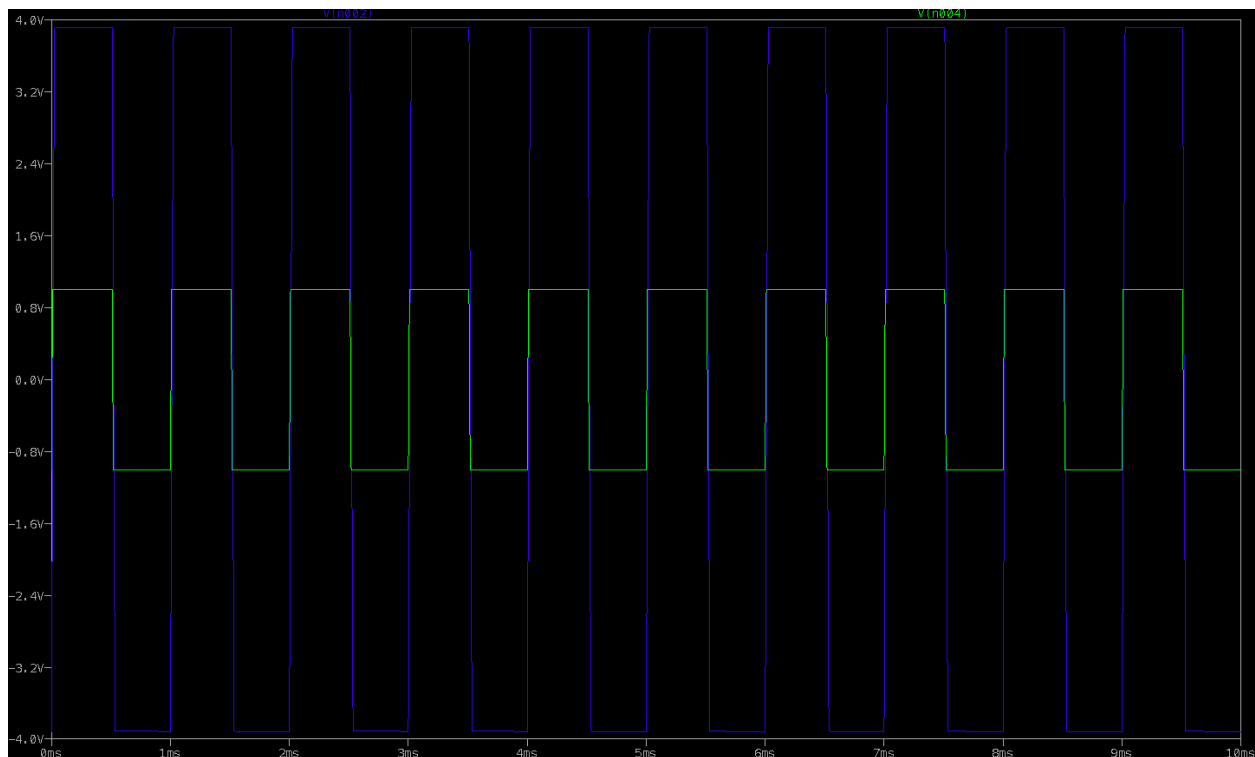


Figure 6: Square Wave

Conclusion

As observed in all three output waveforms, the circuit was able to successfully amplify each of the input signals such that the input amplitude of 1v was increased to 4v. With a constant change of the voltage, we can say that the gain of our simulated amplifier is $A=4$. It took the input amplitude of 1 v and amplified it to 4 v.

Overall, this lab was a good introduction to the semester and in the future we look to explore more configurations on LM741 in action with different applications.