

# Designing and Simulation of Differential Amplifier

...

By Sudeep Saurabh

# Introduction

For this project, I wanted to create a practical application , so I studied various real world applications of the circuits I could create in LTspice using a CMOS inverter. I chose to make a CMOS differential amplifier because I wondered how a person controls the music and sound equipment.

# Project Objective

- My project objective is to design a CMOS differential amplifier
- Convert the design into a Component.
- Test the component
- To use the designed component as Volume Controller.

# Software and Components

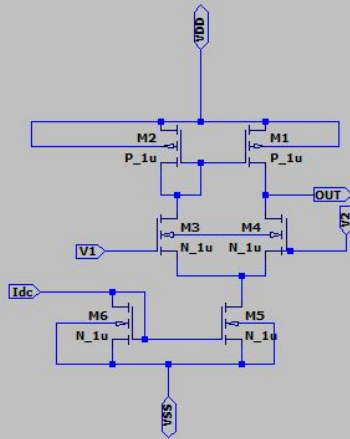
## Software used

I had decided to use LTSpice XVII to design and simulate my project. LTSpice is a SPICE-based analog electronic circuit simulator computer software, produced by semiconductor manufacturer Analog Devices. It is the most widely distributed and used SPICE software in the industry.

## Components Required

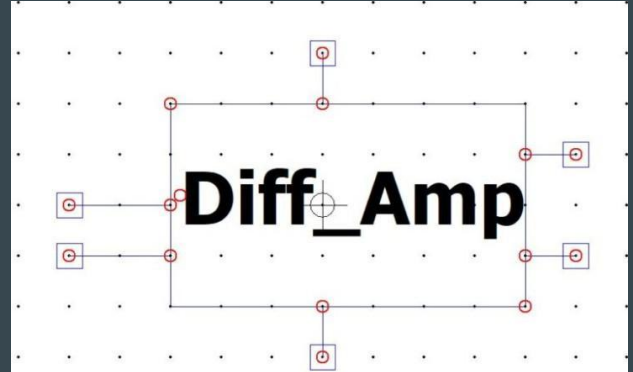
- NMOS
- PMOS
- Voltage source
- Wire
- Ground.

# Schematic of Differential Amplifier



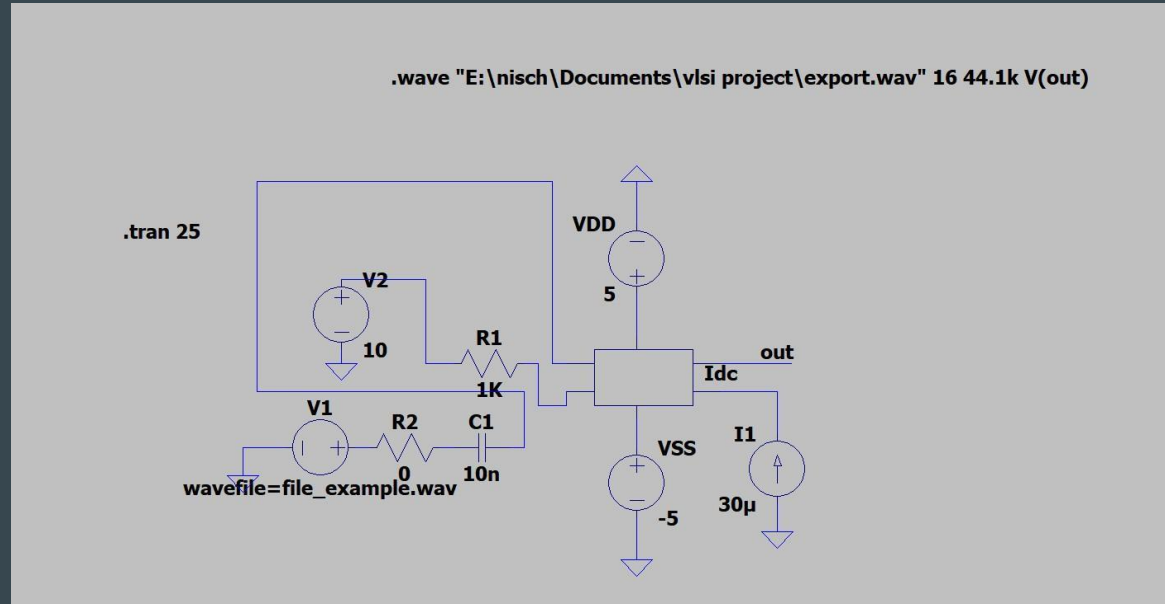
```
.MODEL N_1u NMOS LEVEL = 3
+ TOX = 200E-10    NSUB = 1E17    GAMMA = 0.5
+ PHI = 0.7      VTO = 0.8      DELTA = 3.0
+ UO = 650      ETA = 3.0E-6    THETA = 0.1
+ KP = 120E-6    VMAX = 1E5     KAPPA = 0.3
+ RSH = 0        NFS = 1E12     TPG = 1
+ XJ = 500E-9    LD = 100E-9
+ CGDO = 200E-12 CGSO = 200E-12 CGBO = 1E-10
+ CJ = 400E-6    PB = 1        MJ = 0.5
+ CJSW = 300E-12 MJSW = 0.5
*

.MODEL P_1u PMOS LEVEL = 3
+ TOX = 200E-10    NSUB = 1E17    GAMMA = 0.6
+ PHI = 0.7      VTO = -0.9     DELTA = 0.1
+ UO = 250      ETA = 0        THETA = 0.1
+ KP = 40E-6     VMAX = 5E4     KAPPA = 1
+ RSH = 0        NFS = 1E12     TPG = -1
+ XJ = 500E-9    LD = 100E-9
+ CGDO = 200E-12 CGSO = 200E-12 CGBO = 1E-10
+ CJ = 400E-6    PB = 1        MJ = 0.5
+ CJSW = 300E-12 MJSW = 0.5
*
*
```

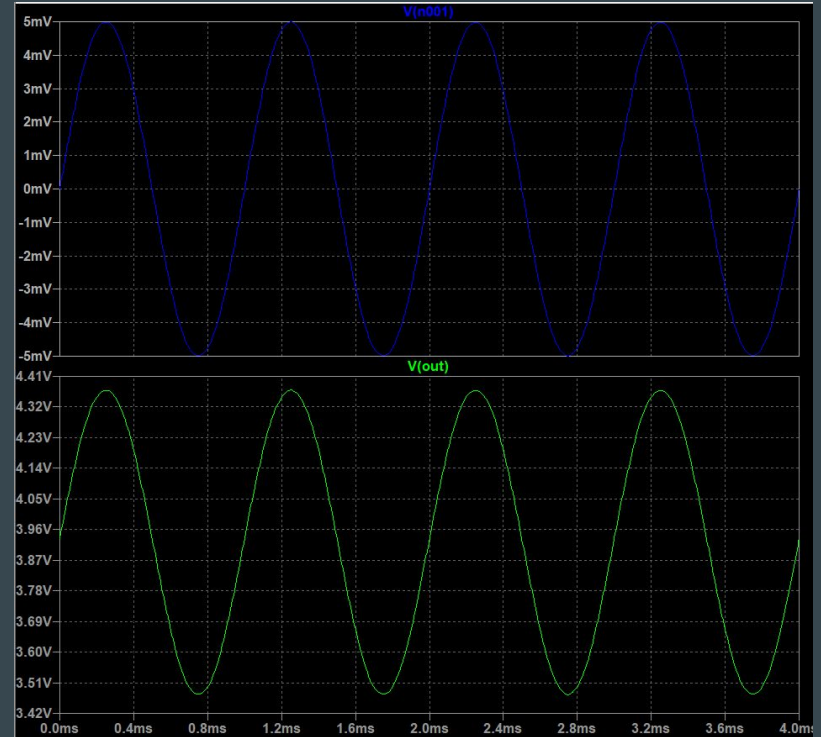
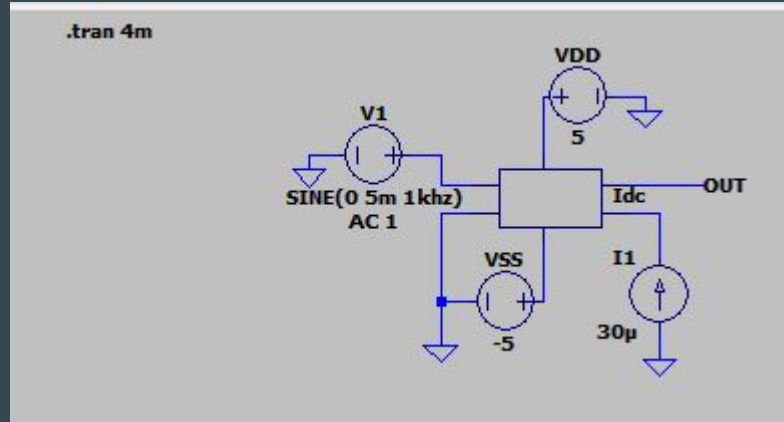


# Schematic of Differential Amplifier

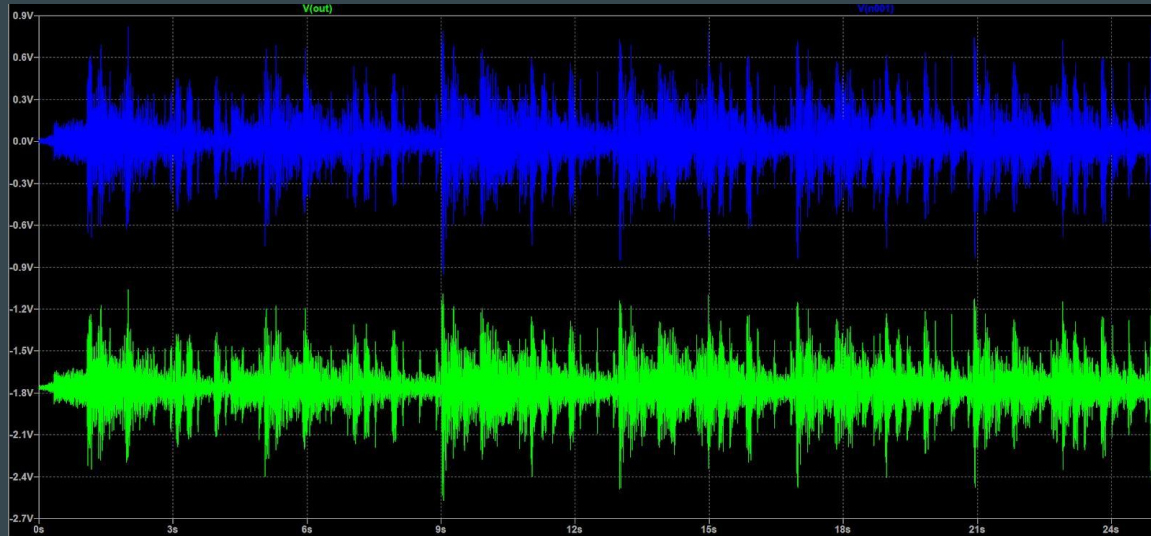
LTspice can produce a sound output in .WAV format. In my simulation, the output sound is stored as export.wav. Sample at the rate of 16 bits. 44.1K is the number of samples per simulated second.



# Simulation of Volume Controller



# Simulation of Volume Controller





# Conclusion

I was able to send an input audio file in .wav format in the circuit, I was able to do this because of the feature of LTSPICE, then I got the output also in .wav form where I was able to hear the low volume version of the original audio file . Hence my project was successfully done.