University of Salahaddin – Hawler
College of Engineering
Department of Electrical Engineering (4th Year)

Laboratory Manual for

Microelectronics

Lectured by:
Ara Abdulsatar Assim

Contact:
ara.assim@su.edu.krd

Autumn Semester
2022

Download the lectures, lab manual & books here:

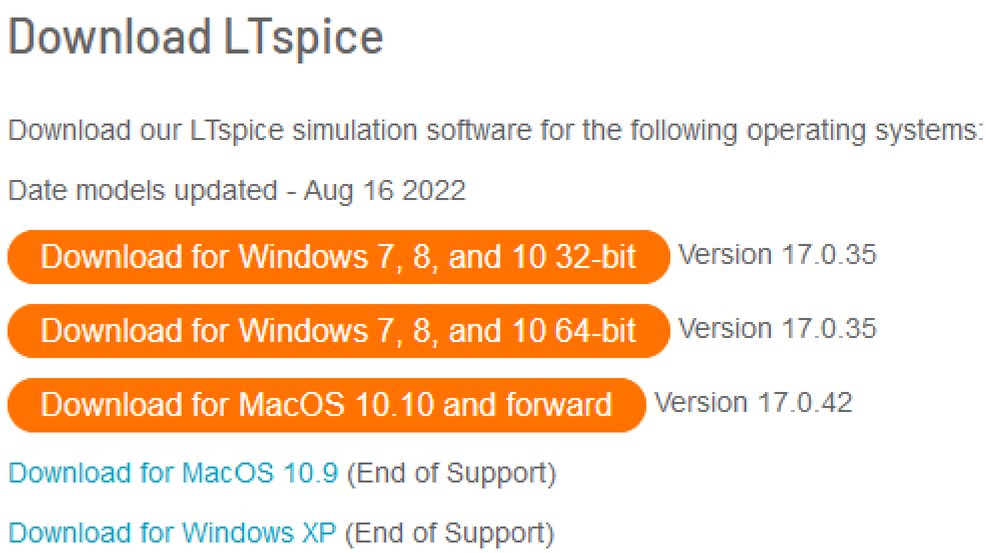
https://bit.ly/3R1UBXd

Installation guide

LTspice is a free software, it can be downloaded through this link.

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

Choose the appropriate version according to your computer's specifications:



Run the installation wizard and follow the steps.

List of Experiments

Experiment (1)
Introduction to LTspice

Objectives:

Students will learn how to use the LTspice circuit simulator, including schematic entry, selecting and running different simulation types, and how to produce simulation output for reports. Example circuits will be simulated to demonstrate the capabilities of LTspice.

Introduction:

LTspice is a fast, free circuit simulator. Linear Technology, Inc. originally designed it for engineers to simulate circuits. It competes with expensive commercial products like Electronic Workbench and PSpice. Advantages of LTspice are that circuit sizes are unlimited, new models can be added, and the user can modify the simulator's behavior.

In this lab exercise you will:

- Get familiar with LTspice
- Enter a simple schematic
- Explore the LTspice component library
- Run simulations for Transient analysis (time-domain)

Procedure:

1. Open LTspice by clicking on this icon.

LTspice XVII

When it starts up, LTspice may ask if you would like to download an update. Do not do that now.

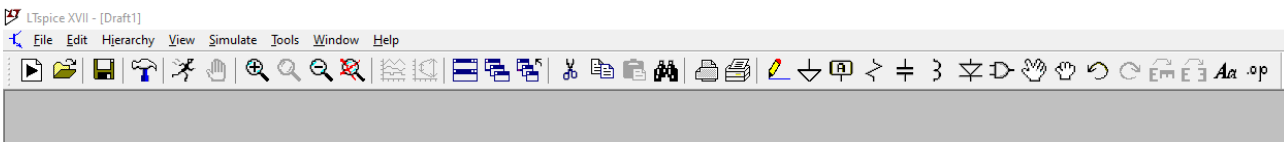


Fig. 1. LTspice startup window

2. Create new schematic by clicking on this icon

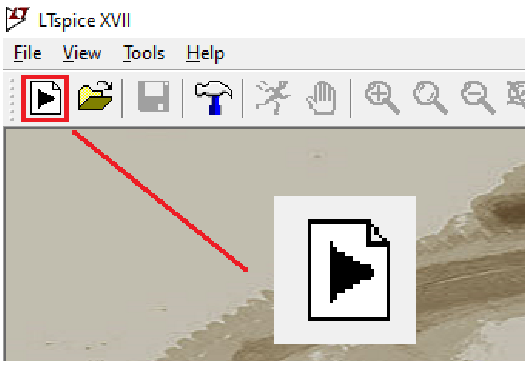


Fig. 2. Create new schematic

3. Left click on Tools > Control Panel > Drafting Options

Set **Pen thickness** to 5:

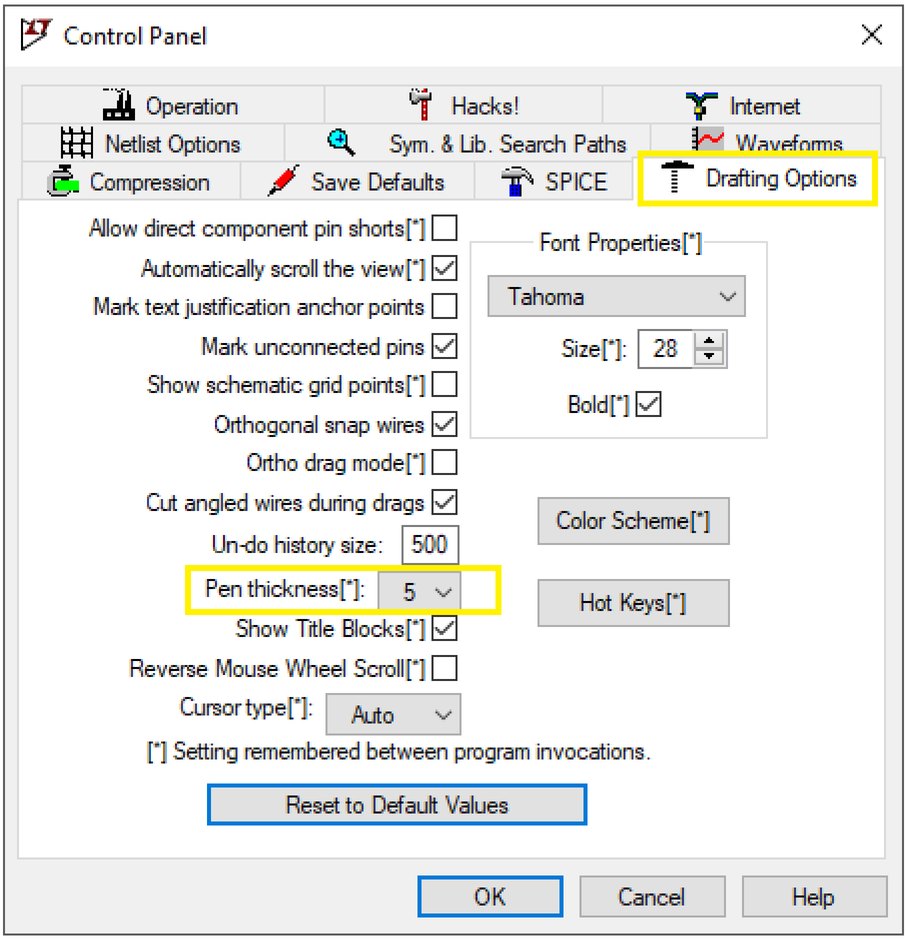


Fig. 3. Changing Pen thickness

Additionally, click on **Waveforms**, change **Data trace width** and **Cursor width** to 4, this will make schematics look better in reports.

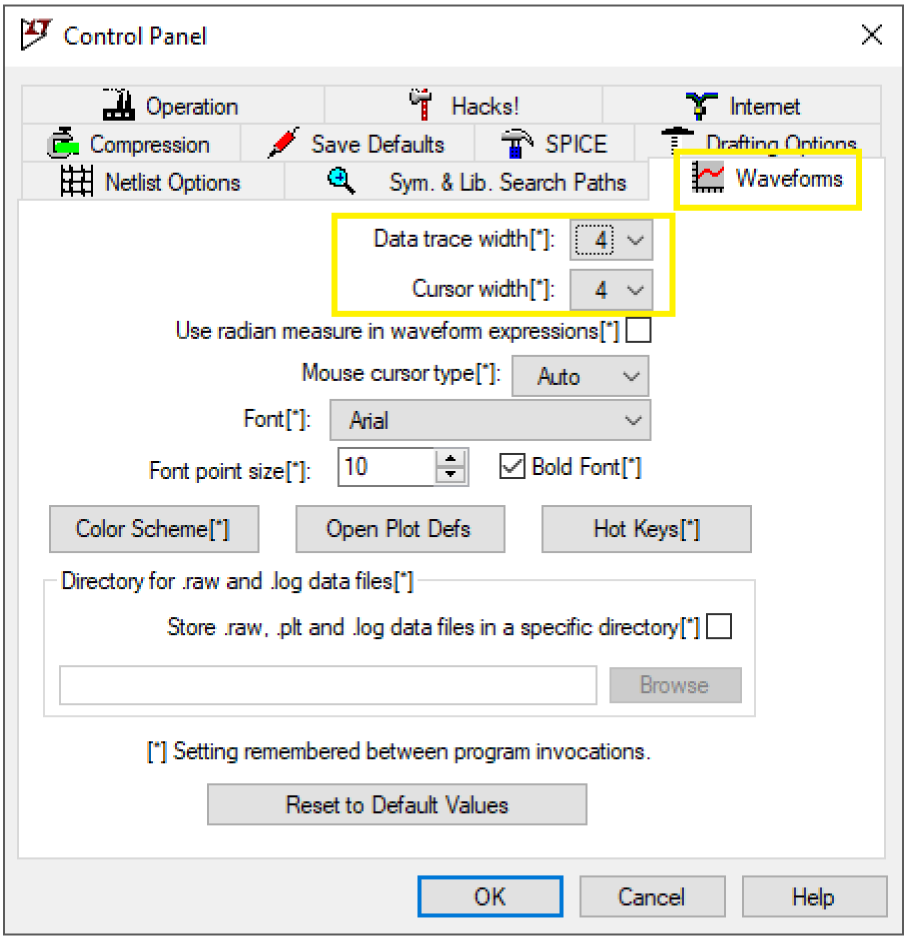


Fig. 4. Changing Data trace width and Cursor width

4. Add the element in Fig. 5., let us begin by adding an operational amplifier. Click on the AND symb to open the library of components. Insert "**OP27**" as shown in Fig. 6.

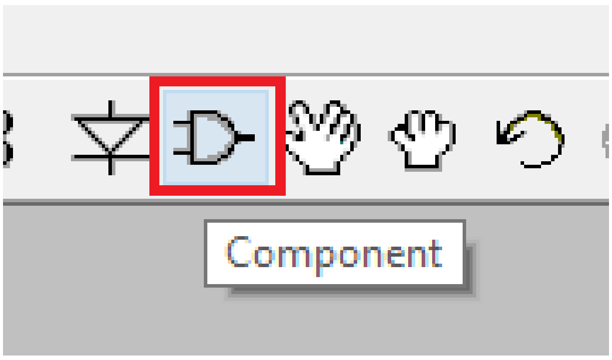


Fig. 5. Components library

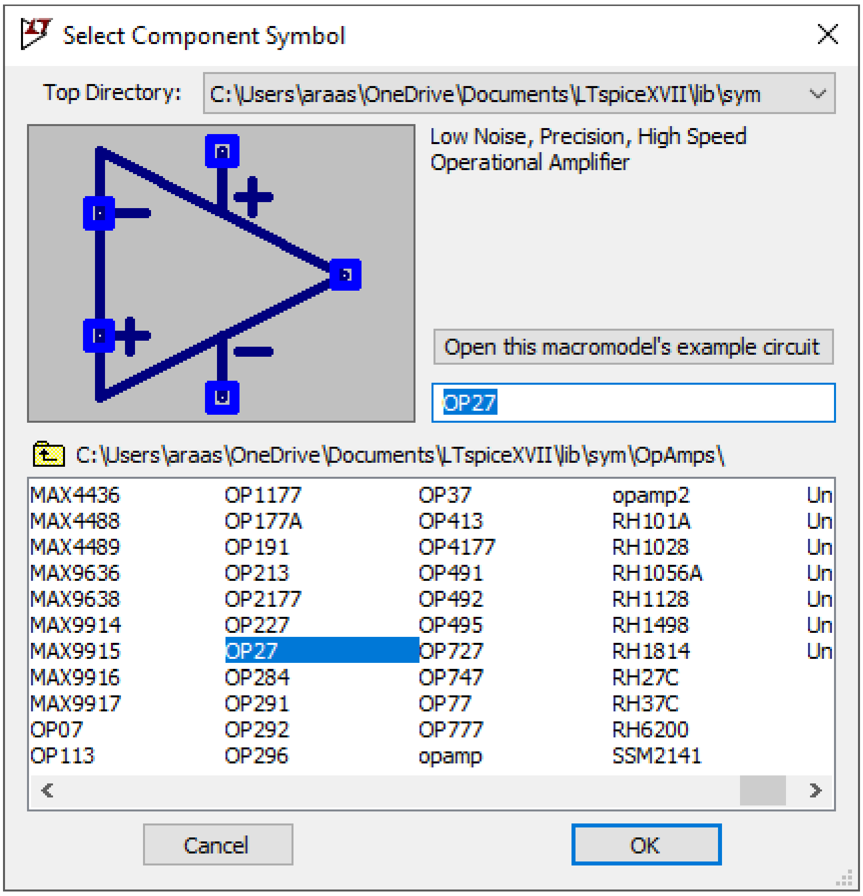


Fig. 6. Inserting an element

Add two resistors, click on resistor symbol as in Fig. 7.

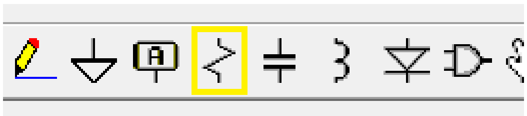


Fig. 7. Resistor symbol

Set the value to 1K, by right-clicking on the resistor & setting the resistance to 1K.

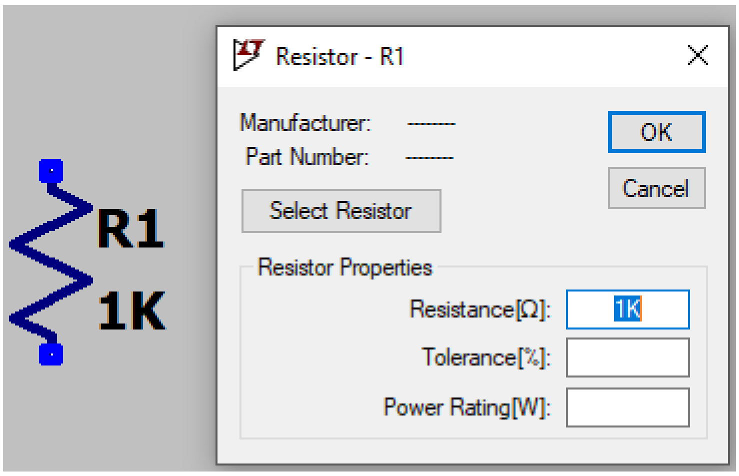
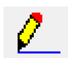


Fig. 8. Changing resistance value

Add ground  and connect the components using wire  as shown in Fig. 9.

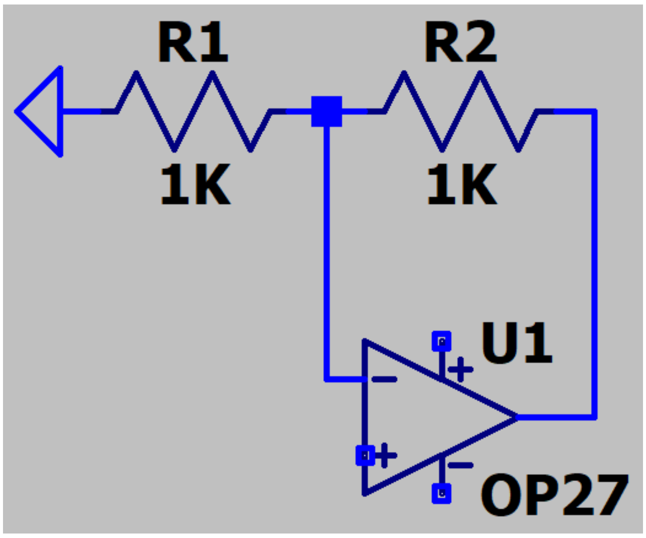


Fig. 9. Connecting the two resistors to negative pin of the op-amp

Insert a 10K resistor to the output of the amplifier as in Fig. 10.

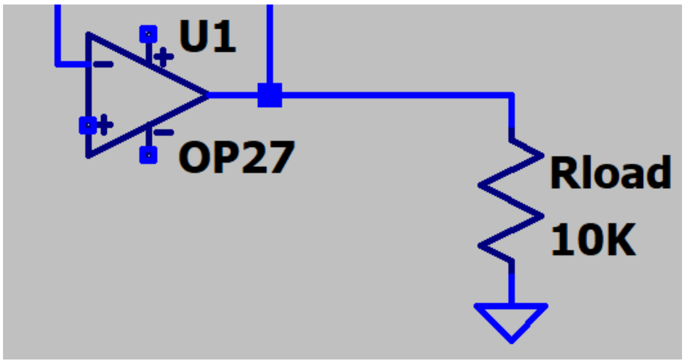


Fig. 10. Adding load

Right click on the wire, click on **Label Net,** name this wire **OUT** as in Fig. 11.

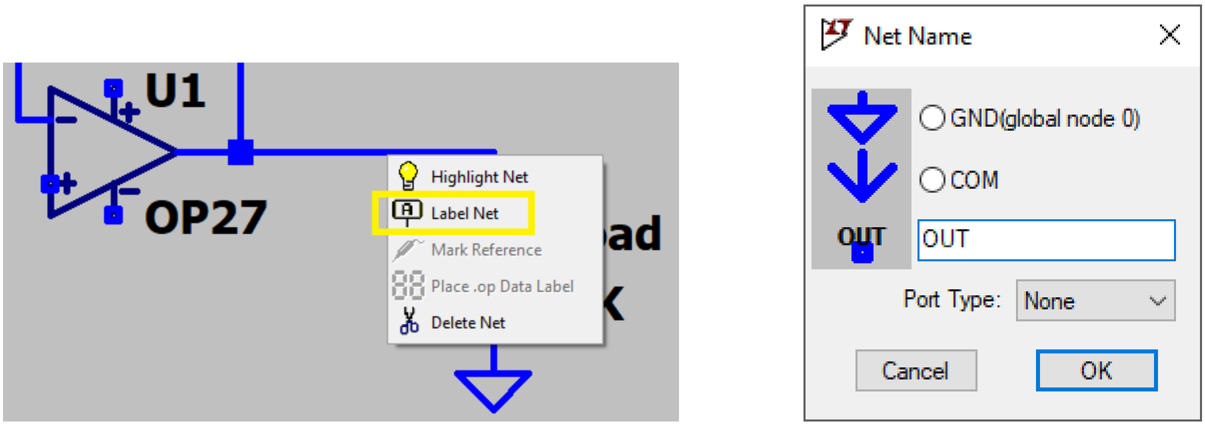


Fig. 11. Labeling a wire

Add three voltage sources, as shown in Fig. 12.

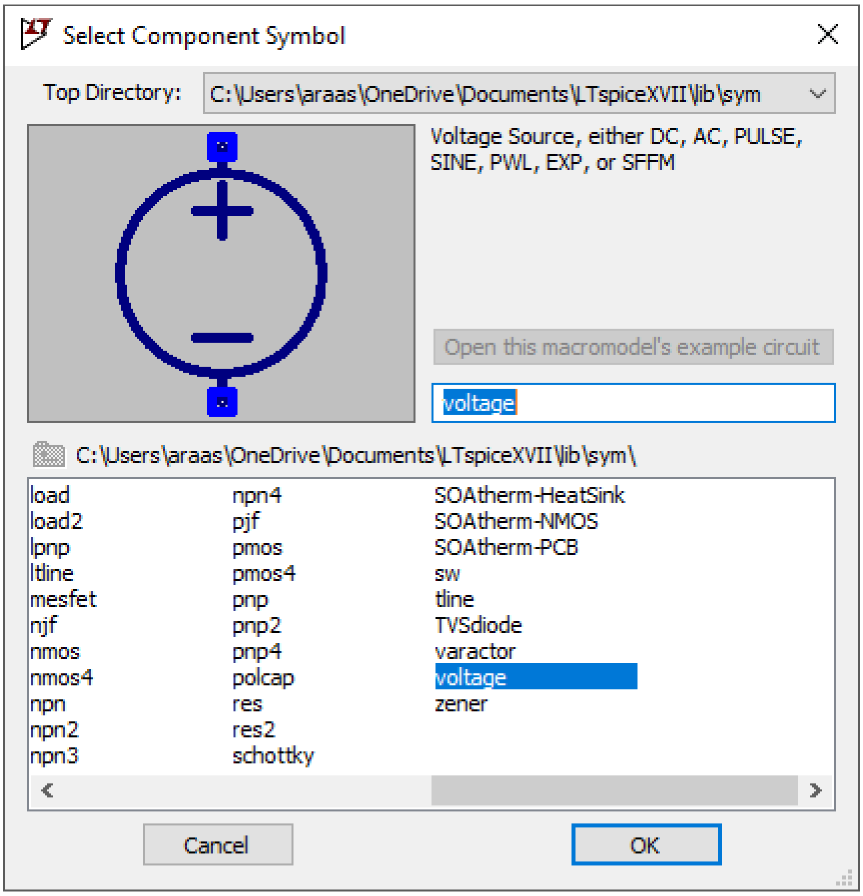


Fig. 12. Finding voltage source in the component library

Add wires and connect them to ground, as in Fig. 13. Right click on the voltage source to set their values, set V1 and V2 to 15 and -15. Click OK

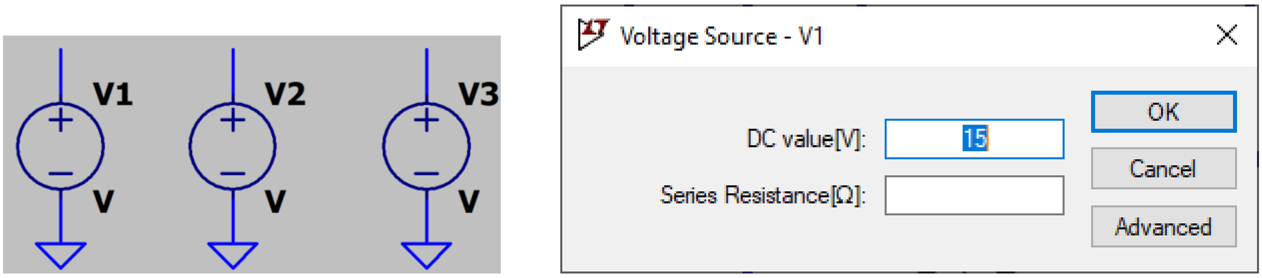


Fig. 12. Three voltage sources

V1 and V2 are the DC sources, V3 is an AC source (sine signal),

right click on V3 > Advanced, you will see the window in Fig. 13. Click on **SINE,** set the values of **DC offset, Amplitude** and **Frequency to** 0, 1 and 10K as in Fig. 13.

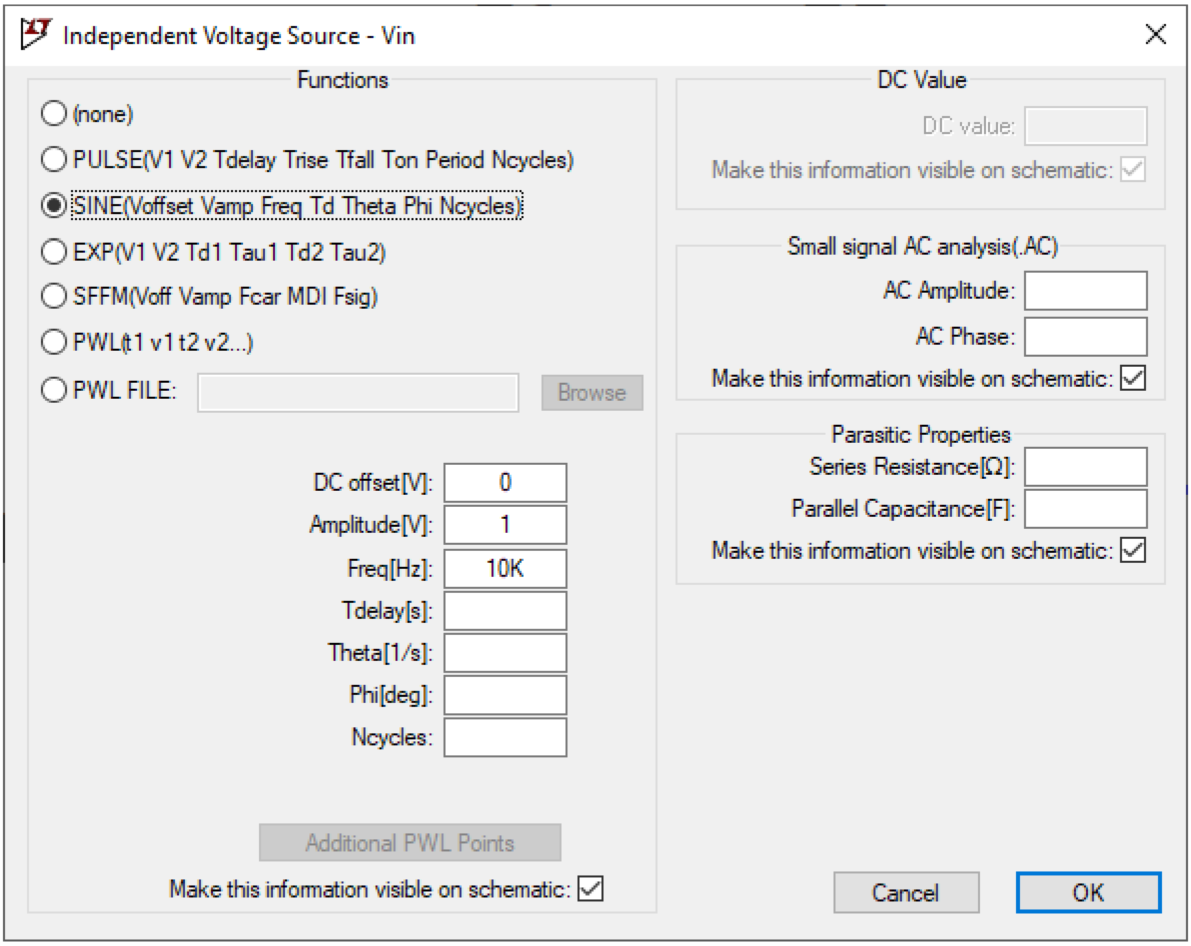


Fig. 13. Three voltage sources

Label the voltage sources V1 & V2 to **+V** and **-V**, label the input voltage "**IN**" and connect the full schematic in Fig. 14**. DON'T FORGET** to label the positive terminal of the op-amp to **+V** and the negative terminal of the op-amp to **-V.**

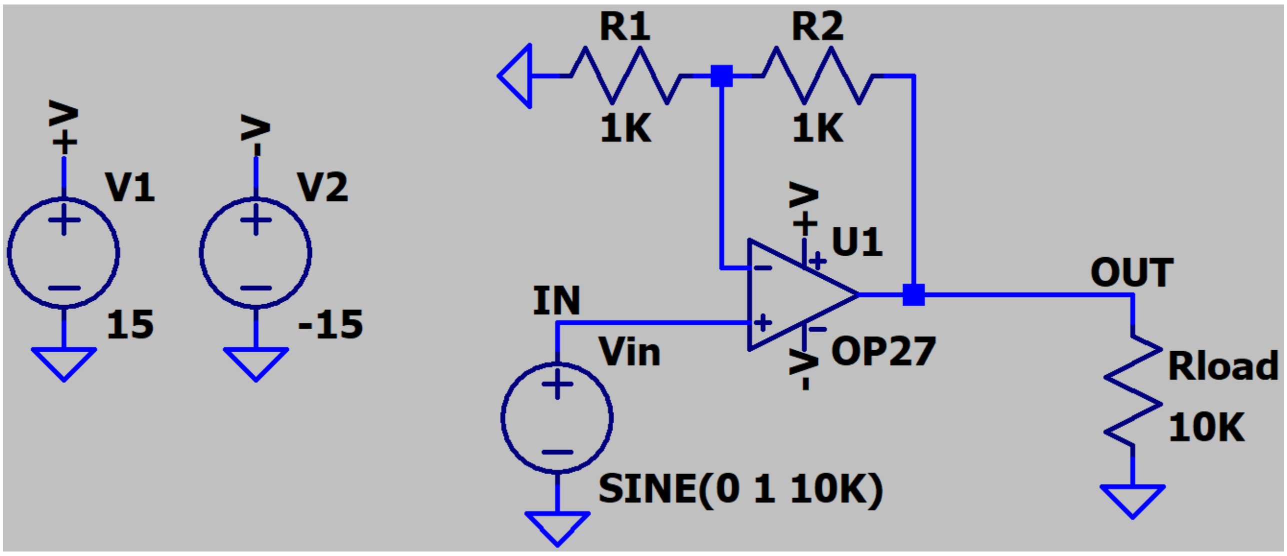


Fig. 14. Non-inverting amplifier's full schematic

After completing the circuit connection in Fig. 14, you may now start running the simulation, click on the run symbol  and set **Stop time** to 1 ms, then click OK.

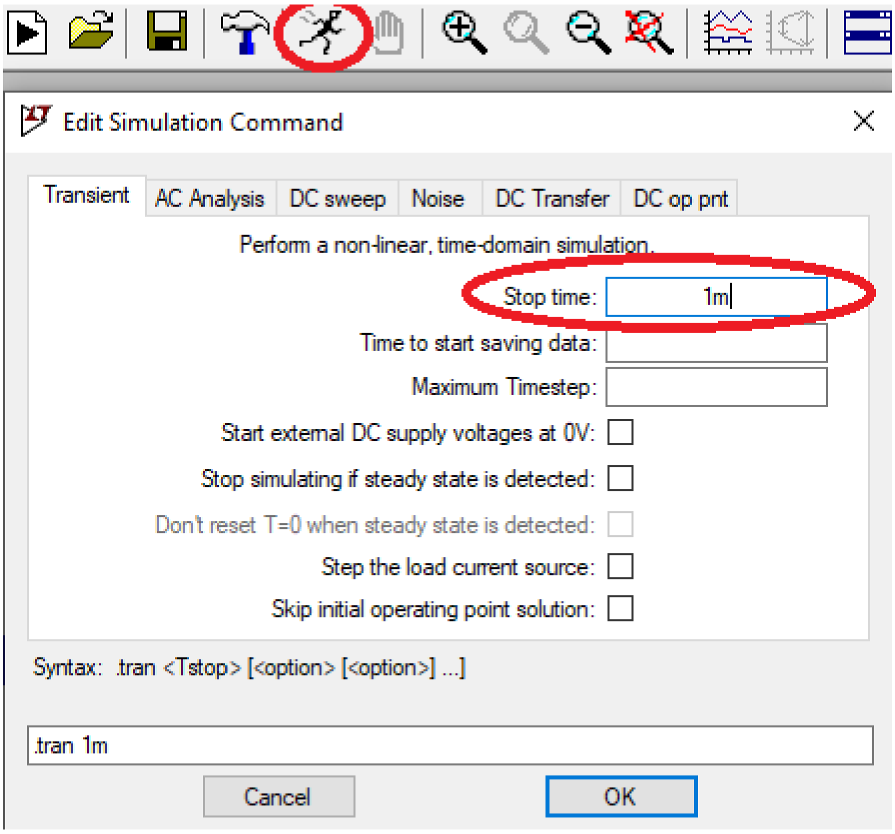


Fig. 15. Transient analysis specifications

You will see a blank (Draft screen) as shown in Fig. 16:

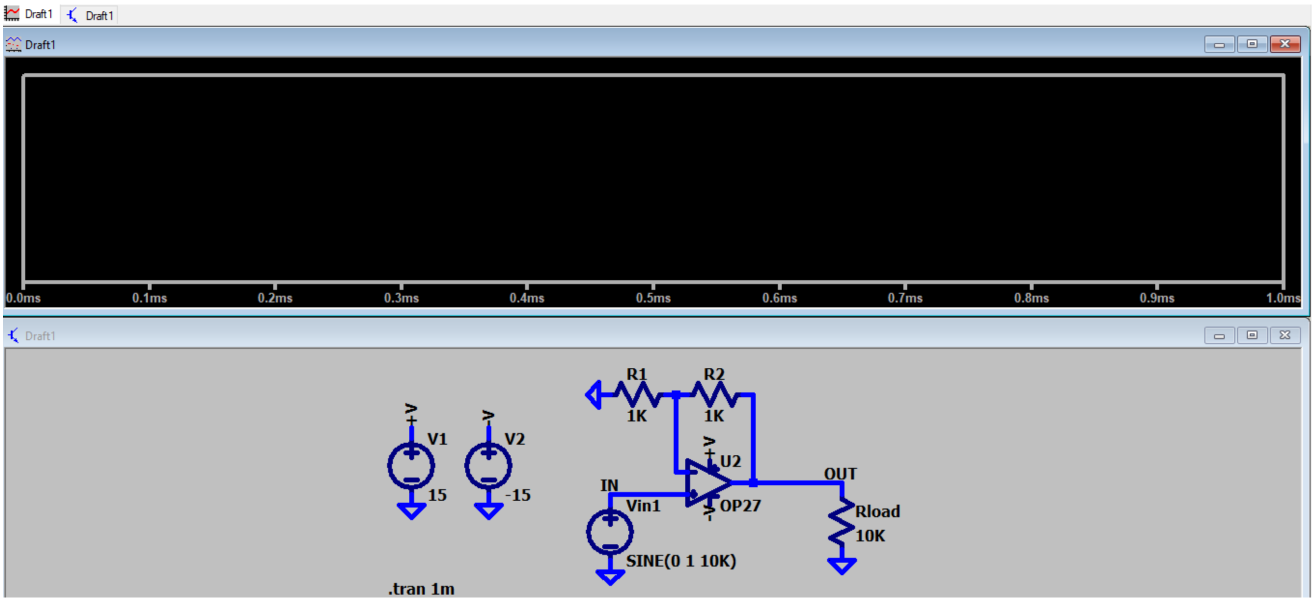


Fig. 16. Screen capture after running transient analysis

To obtain input and output waveforms, you need to click on the wires indicated in Fig. 17.

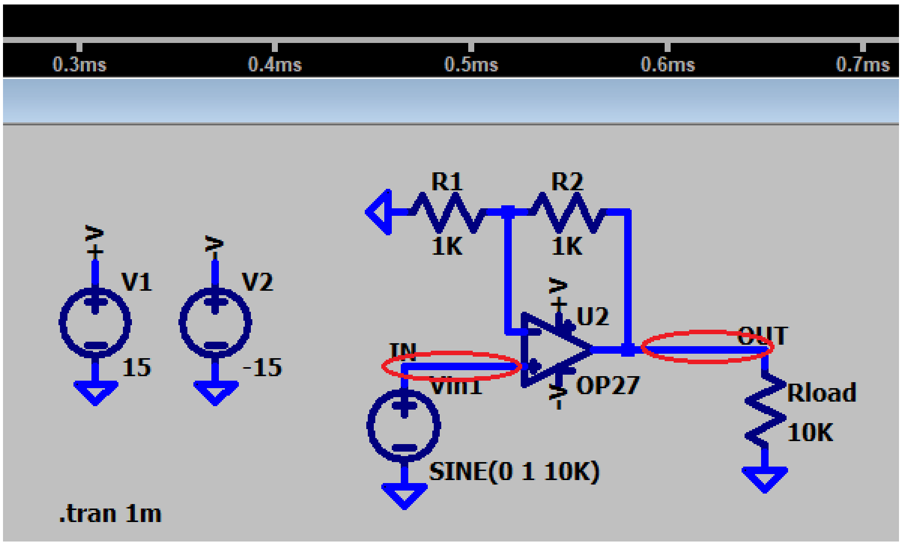


Fig. 17. Clickable wires

The results are provided in Fig. 18:

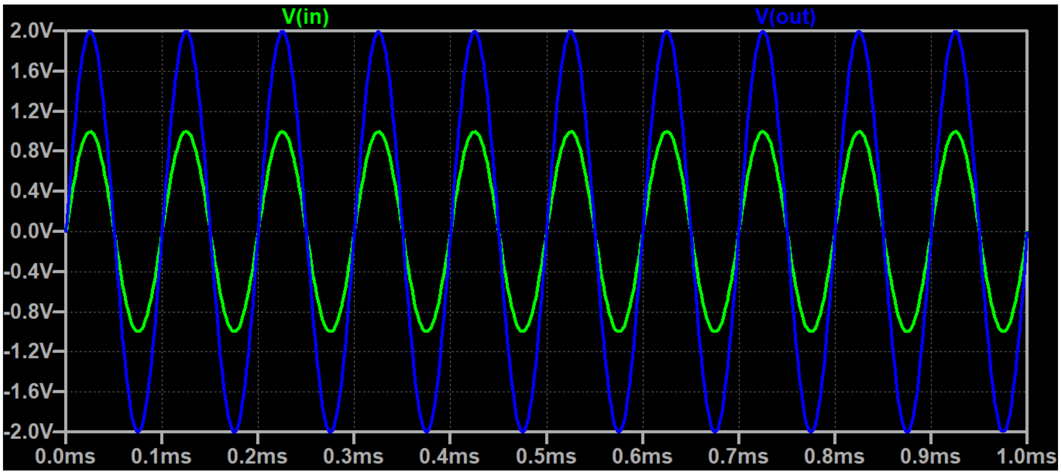
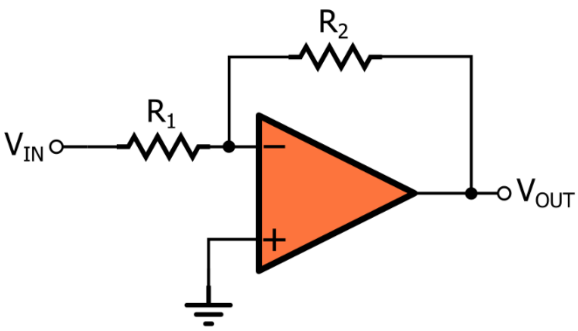


Fig. 18. Input Vs. Output waveforms in time-domain

<u>Report requirements:</u>

1. Modify the circuit in this experiment into an inverting op-amp, amplify this signal: $0.5\sin(2\pi.100000t)$, set gain as 10.



2. Choose the *Stop time* carefully so only 5 periods will be shown.
3. Add all the screenshots and write a conclusion of your work in the report.
4. Why do we use LTspice? What are the advantages?

Experiment (2)
AC and DC Analyses

Objectives:

Upon the completion of this experiment, students will be able to perform several types of SPICE simulation: DC Analysis, AC Analysis, Transient Analysis (covered in Experiment 1). This first part of this experiment is concerned with performing DC Analysis for a CMOS inverter, while the second part is on performing AC Analysis for an amplifier.

Introduction:

Nearly all SPICE-based simulators are capable of performing well known analysis functions.

Analysis Method	Function
Transient Analysis	Gives time domain waveforms which are plots of voltage or current versus time. (Oscilloscope)
AC analysis	Gives the voltage or current versus frequency in a linearized version of the circuit.
DC analysis	Gives DC voltage or current, usually versus a stepped voltage or current.
Fourier analysis	Plot the frequency content of any waveform using Fourier analysis
Noise analysis	Noise analysis at measurement points.
Monte Carlo	Simulations that reflect variation in circuit elements.
S-parameter	High-frequency characteristic analysis.

In this experiment you will run simulations for:

1. DC operating point.
2. AC small-signal frequency response

Procedure:

1. Open a new schematic file in LTspice.
2. Add a **pmos4** transistor and a **nmos4** transistor as shown in Fig. 1.
3. Place the PMOS transistor above the NMOS transistor as in Fig. 2.

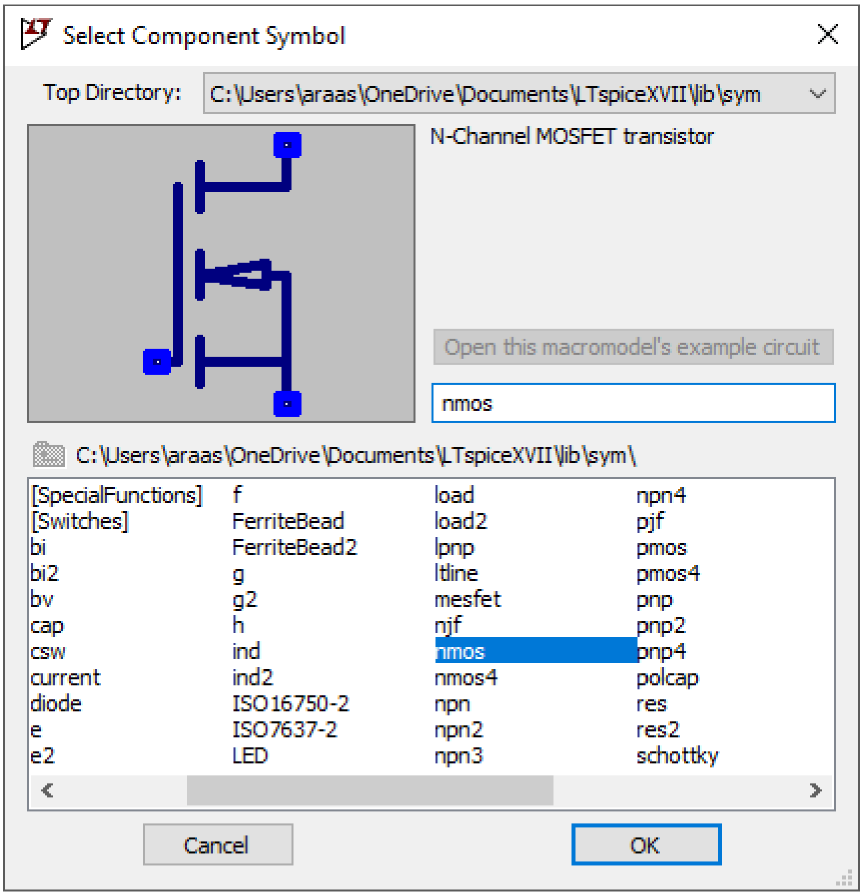


Fig. 1. nmos transistor in component library

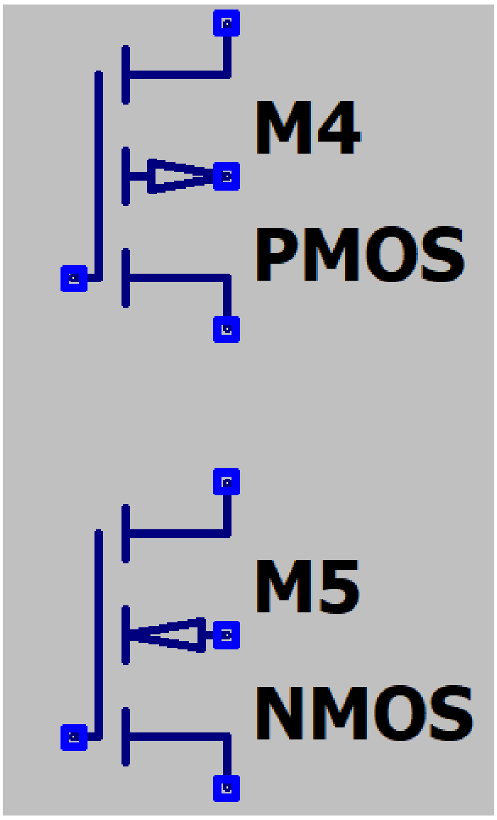


Fig. 2. PMOS and NMOS transistors

Right click on the PMOS transistor and **carefully** set the specifications as illustrated in Fig. 3:

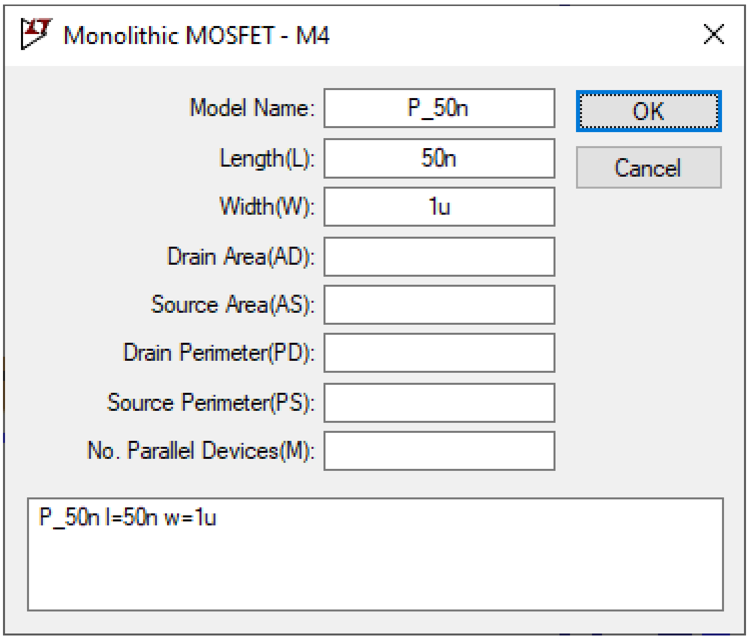


Fig. 3. PMOS specifications

Right click on the NMOS transistor and **carefully** set the specifications as illustrated in Fig. 4:

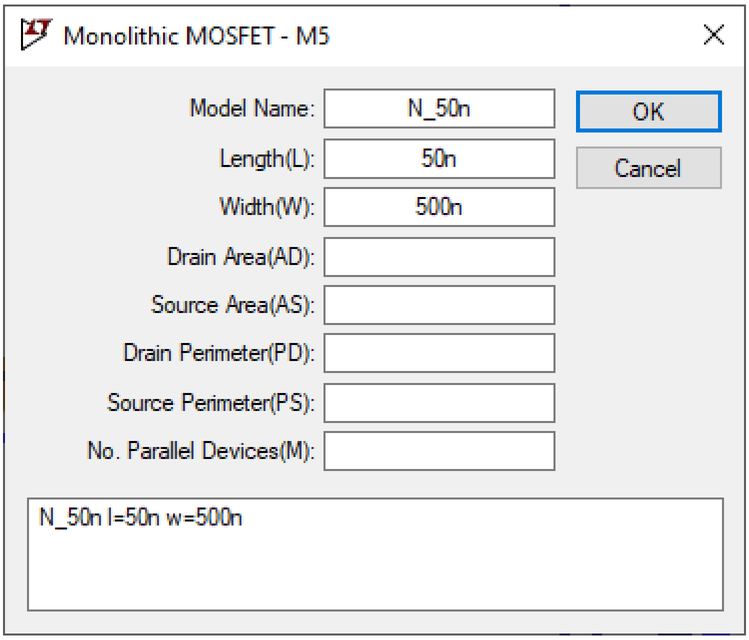


Fig. 4. NMOS specifications

Add two voltage sources (VDD = 1) and (Vin = 0), then connect the circuit in Fig. 5:

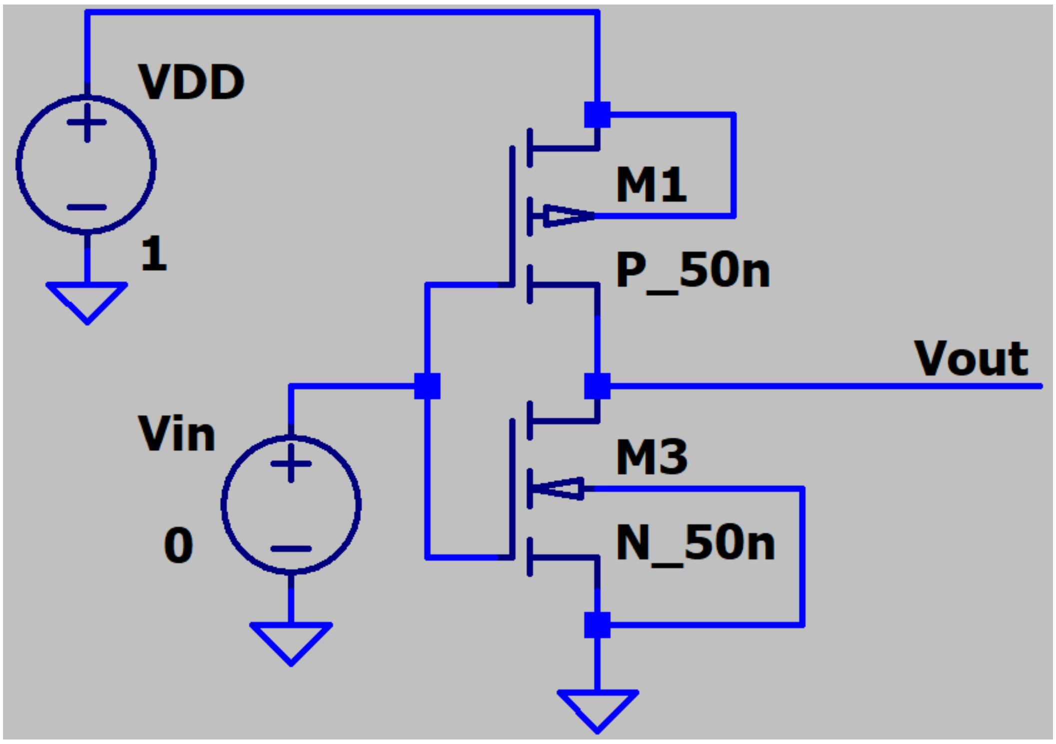


Fig. 5. CMOS inverter (NOT gate)

The transistor specifications have to be inserted through a text document (cmos.txt), the file is given to you by the lecturer, put the schematic file in the same folder with the **cmos.txt** file. Click on the Spice directive symbol, type: **.include cmos.txt** as shown in Fig. 6:

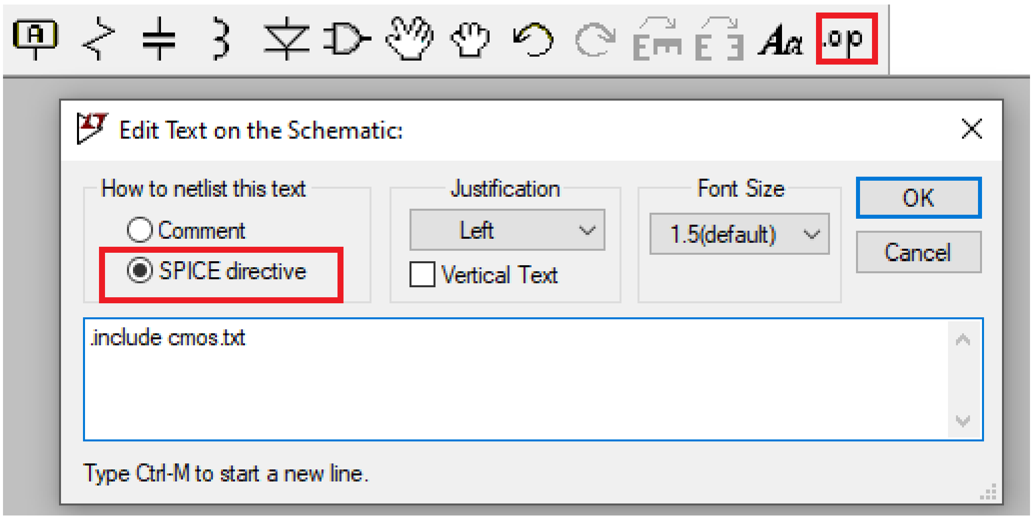


Fig. 6. Spice directive

Now we can run the DC analysis, click on Run  , choose DC sweep, set the specifications as in **Fig. 7:**

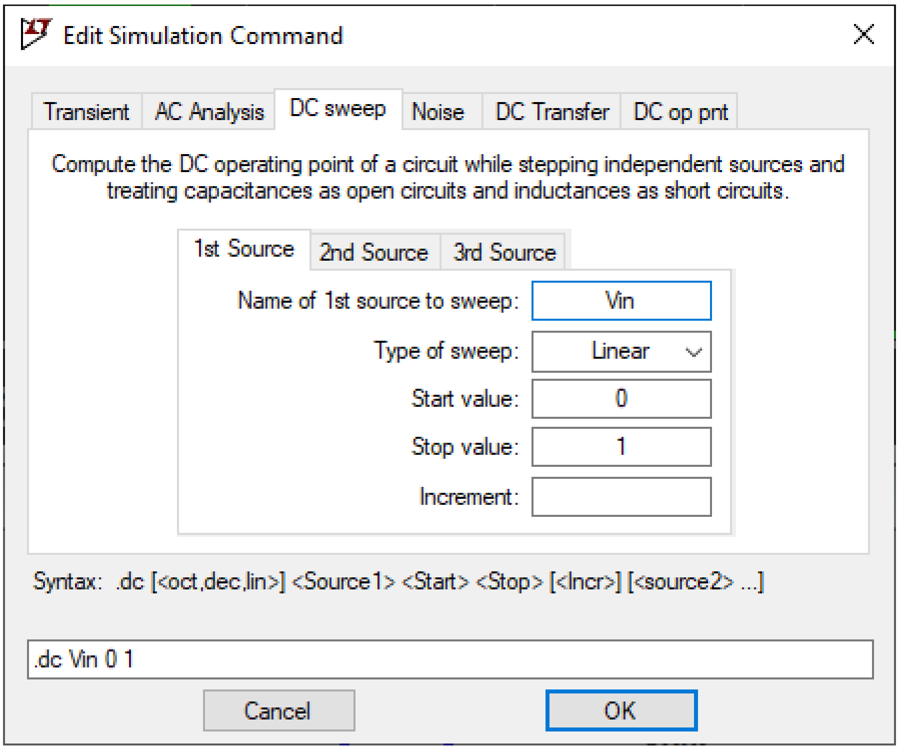


Fig. 7. DC operating point details

Click OK, then select Vout to plot, you should obtain a figure like the plot in Fig. 8.

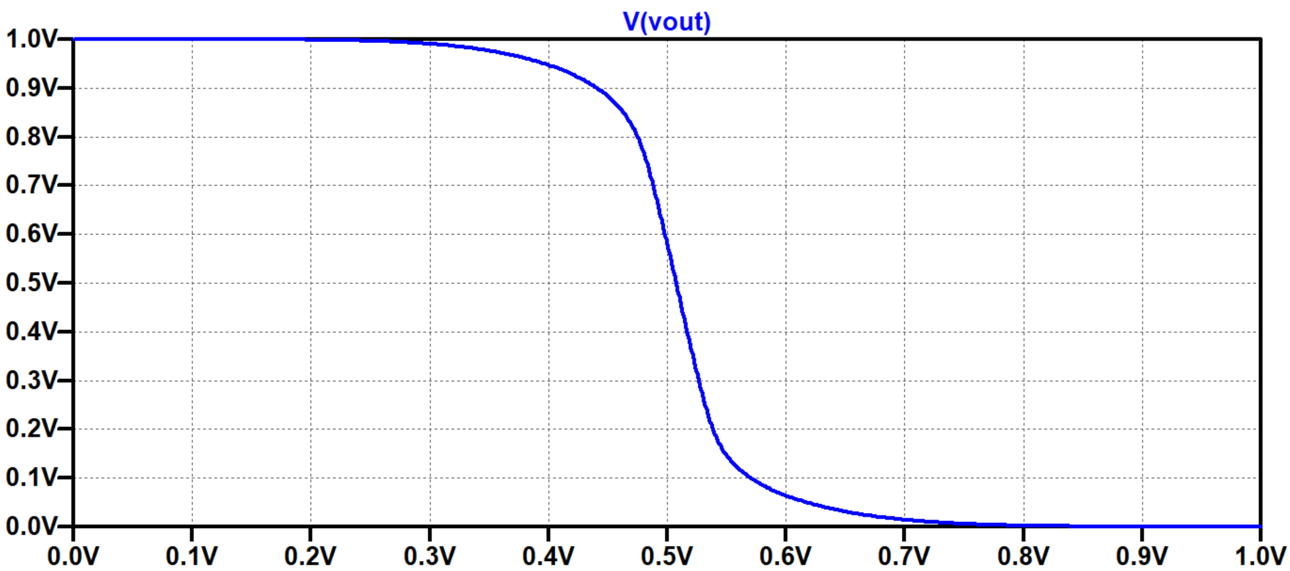


Fig. 8. DC analysis for the CMOS inverter

Once you completed the first part of this experiment, please save the file, for the second part of the experiment, you will need to create a new schematic, configure the circuit provided in Fig. 9:

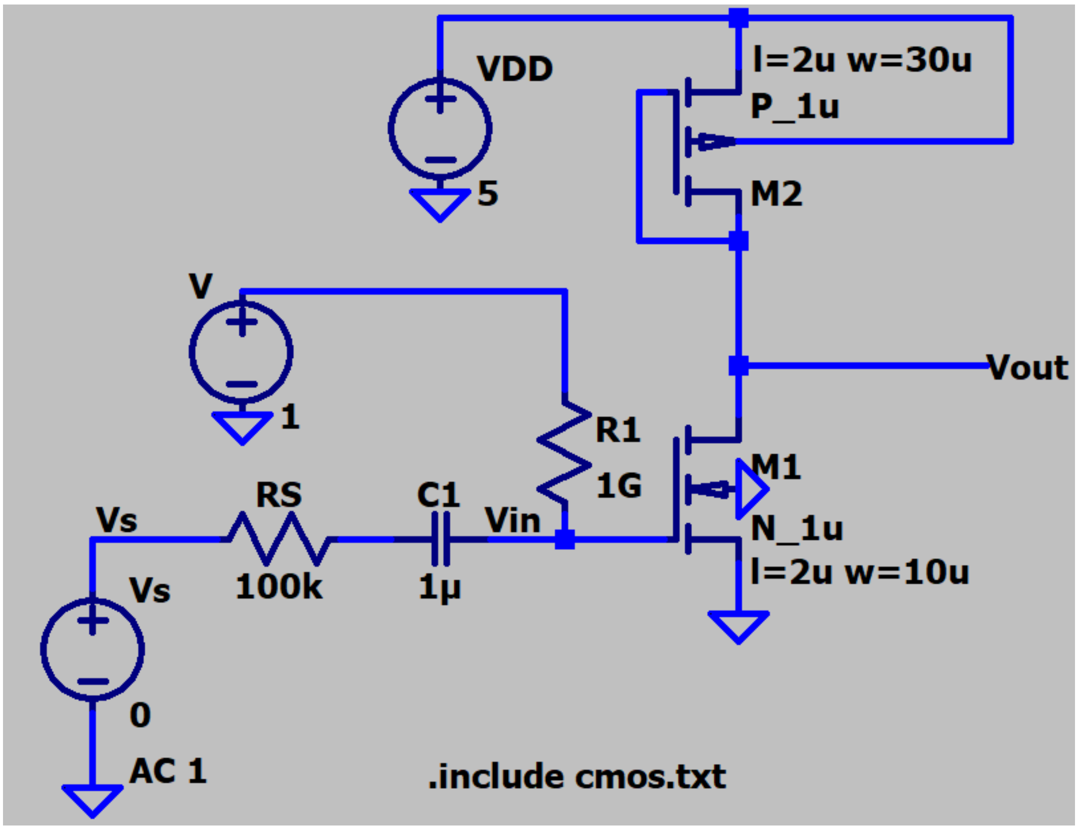


Fig. 9. Circuit diagram of the amplifier

run AC analysis, click on Run  , choose **AC Analysis**, set the specifications as in **Fig. 10:**

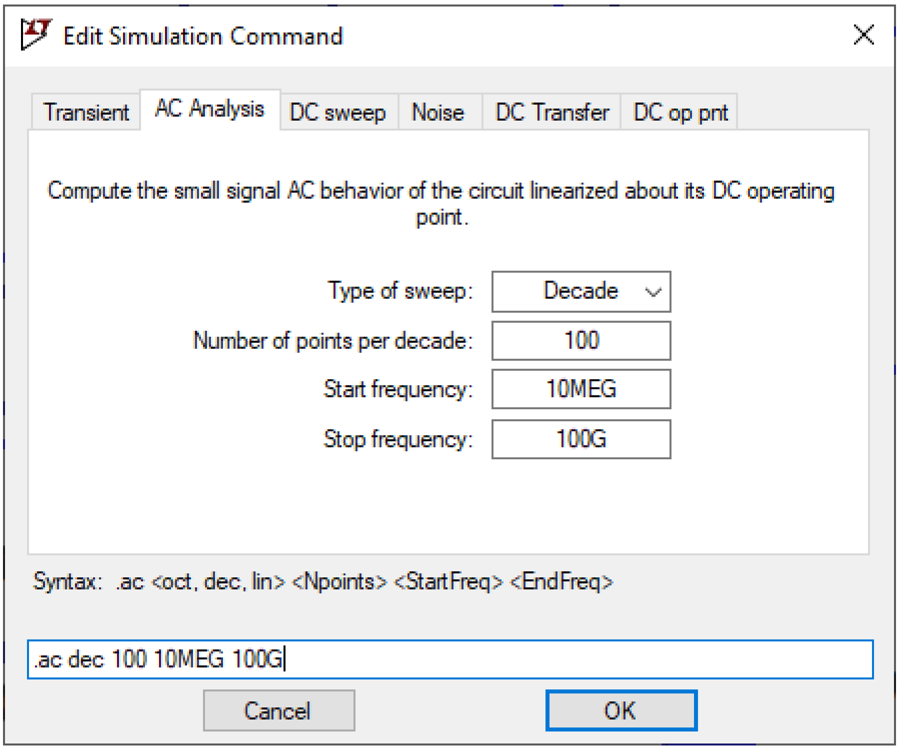


Fig. 10. Details of the AC Analysis

Press OK, then select Vout to plot, the result is the frequency response of the amplifier as depicted in **Fig. 11**:

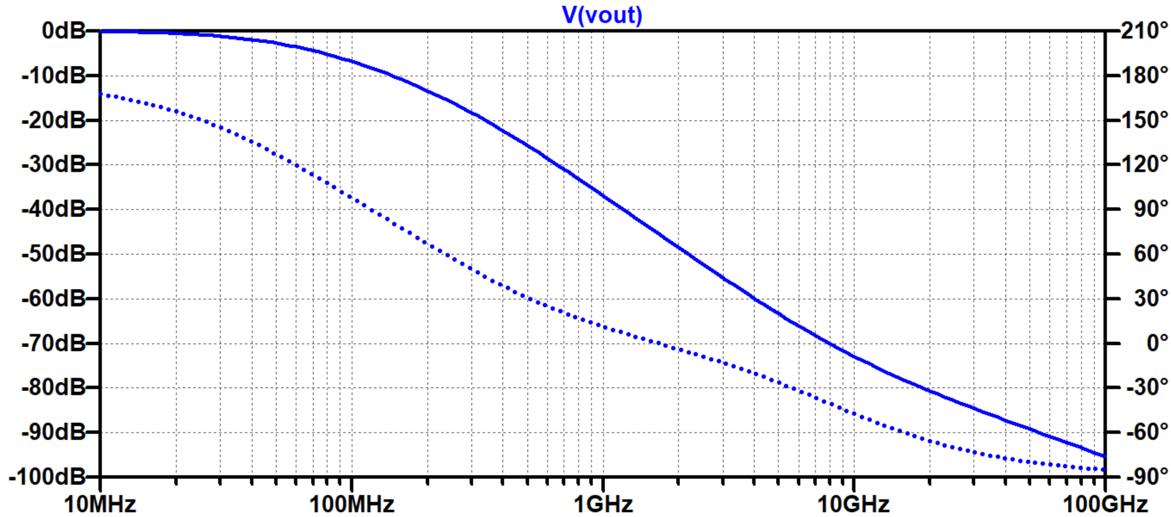
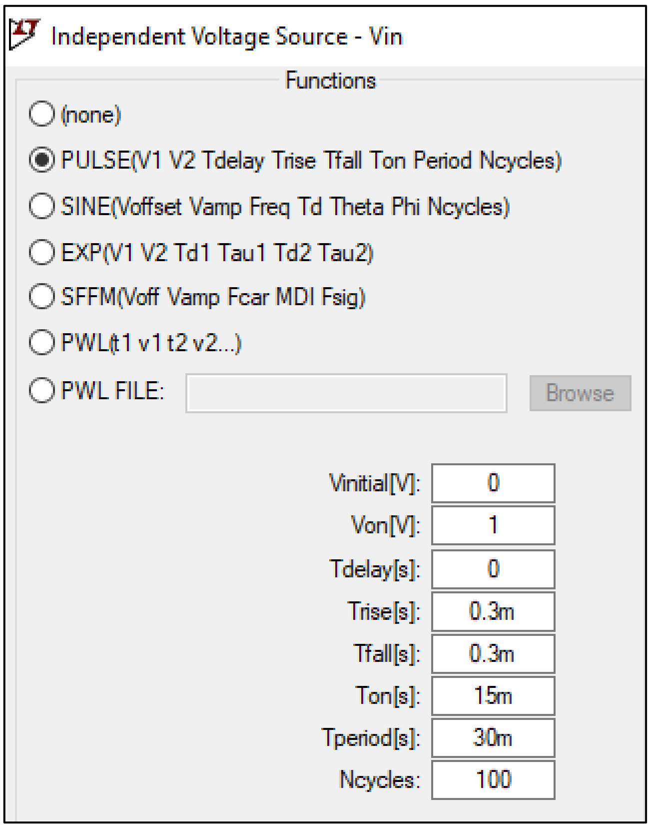


Fig. 11. Frequency response (Bode plot) of the amplifier

Report requirements:

1. For the CMOS inverter, replace the DC voltage source (Vin) to Pulse signal with these specifications, what does the output look like in time-domain? Does the DC operating point change? Add screenshots with your answers.



2. How does an ideal amplifier's phase response look like? Show it on Fig. 11.

Experiment (3)
Junction Field-Effect Transistor Characteristics

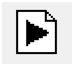
Objectives:

The students will learn how to plot transistor IV characteristics using DC sweep analysis in LTspice.

Introduction:

Junction field effect transistors are widely used devices in electronic circuits (amplifiers, switching circuits, oscillators, etc.), JFETs use a small voltage to control current flow, JFETs are unipolar devices which means the current that passes through the channel consists of either electrons (in an N-type JFET) or holes (in a P-type JFET).

Procedure:

1. Create a new schematic 
2. Search for **njf** (N-channel JFET) as in Fig. 1.

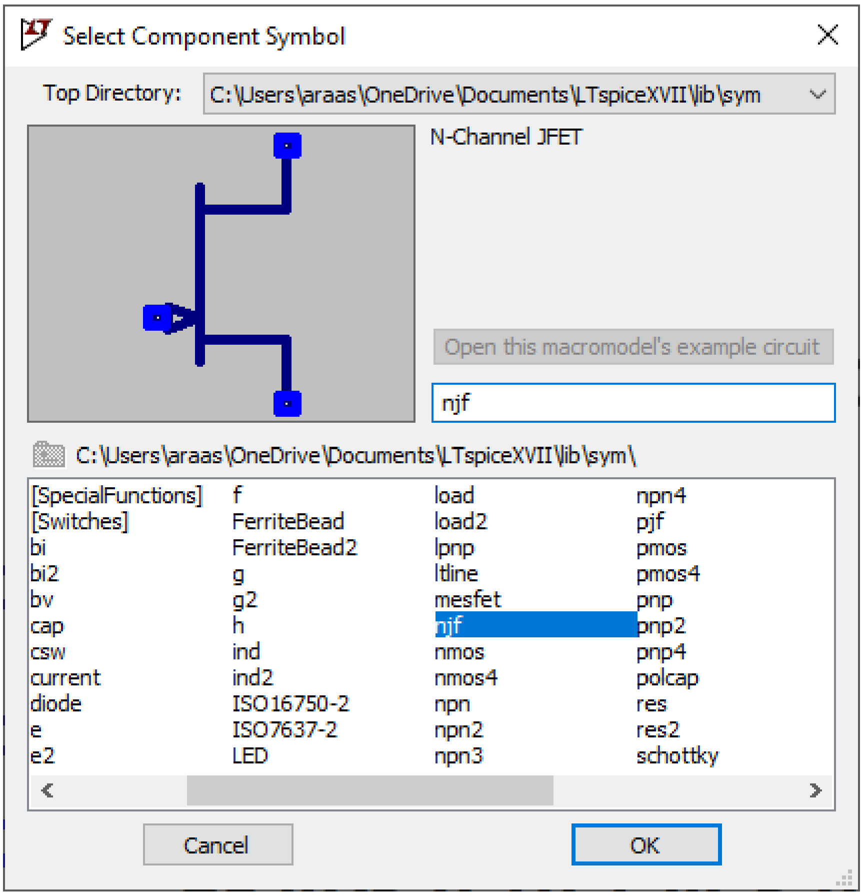


Fig. 1. N-Channel JFET

3. Right click on the transistor symbol then choose **(Pick New JFET)** as in Fig. 2.

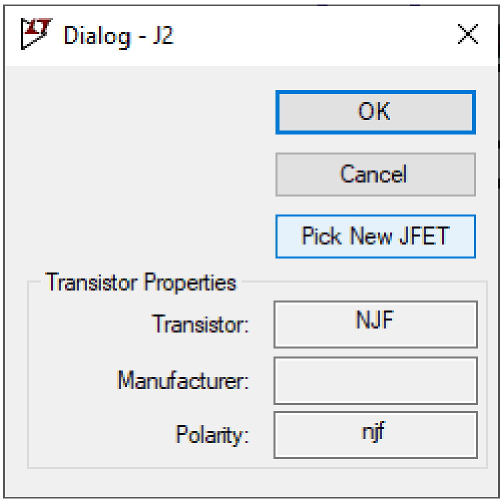


Fig. 2. N-Channel JFET

Choose 2N5432 transistor in the list.

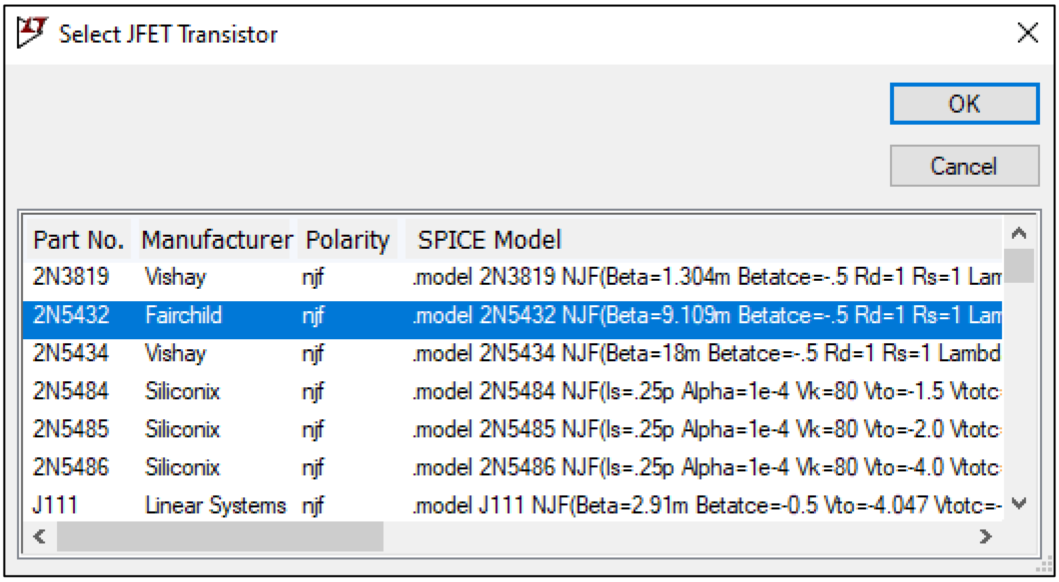


Fig. 3. 2N5432 by Fairchild

Connect the circuit in Fig. 4:

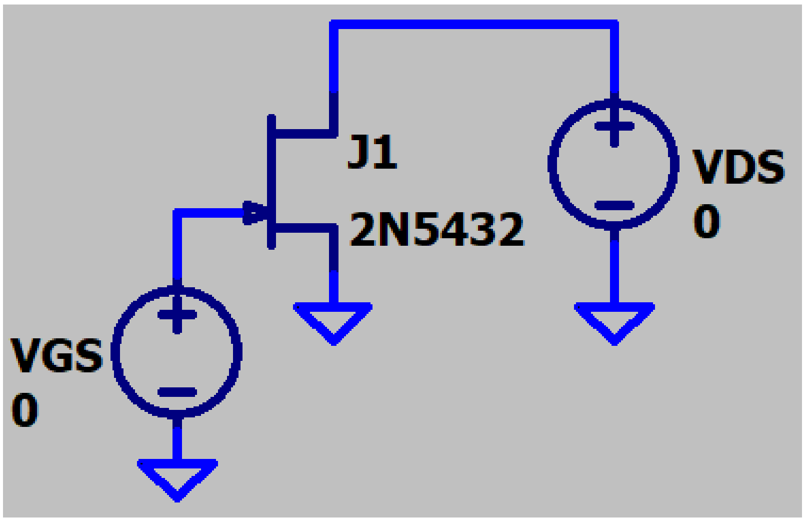


Fig. 4. NJFET testbench

4. Run the system , choose **DC sweep,** pay attention that in this experiment there are two voltage sources. Set their specifications as shown in Fig. 5 and Fig. 6:

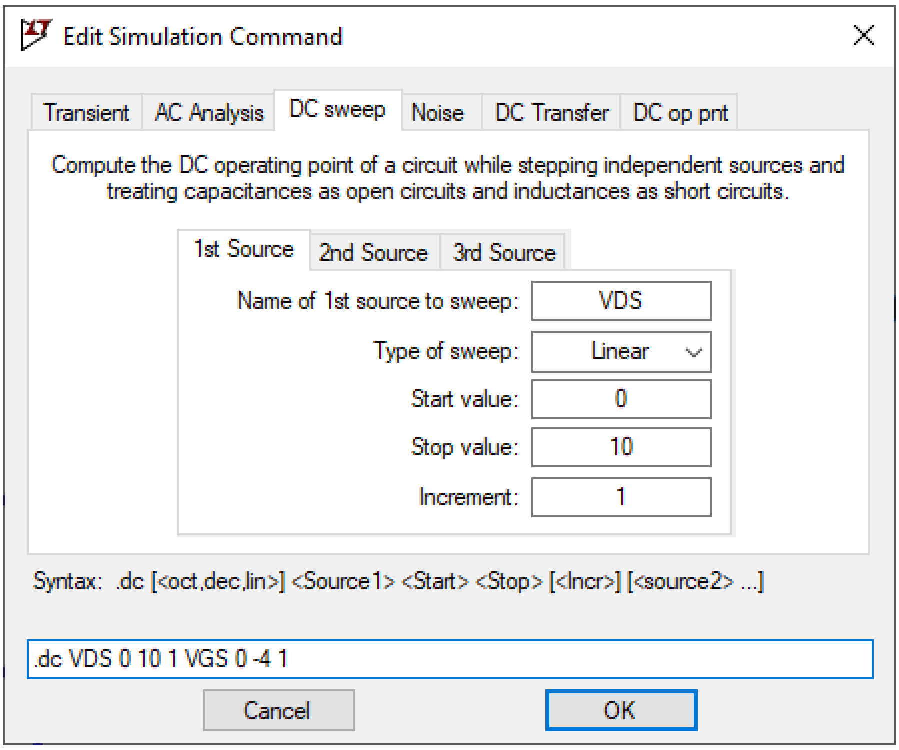


Fig. 5. DC analysis specifications for 1^{st} source

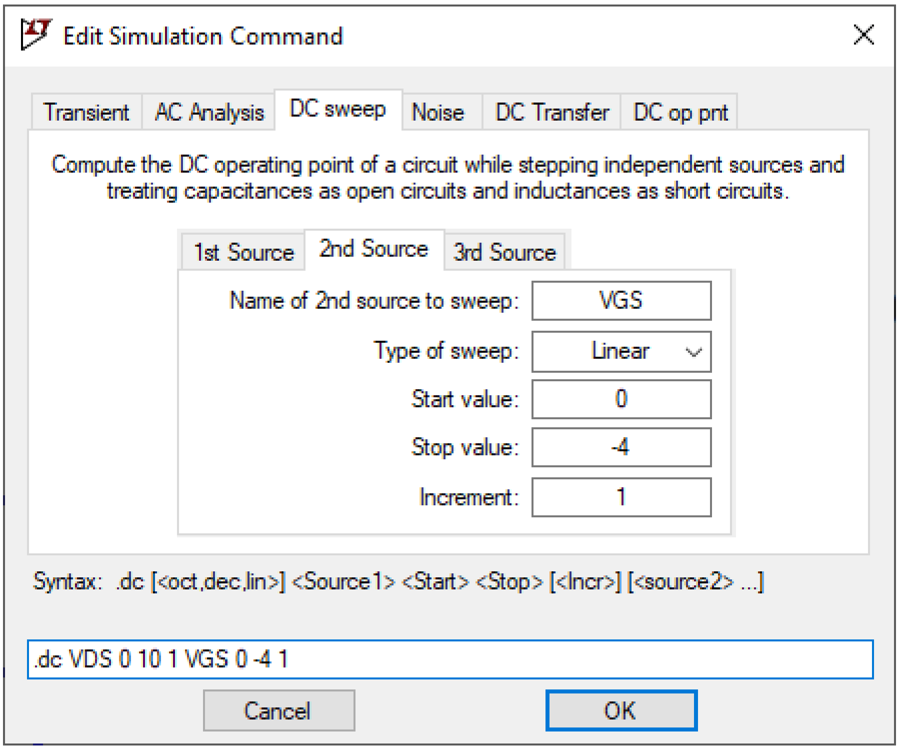


Fig. 6. DC analysis specifications for 2^{nd} source

Place the mouse cursor on drain terminal of the transistor, and you will get the output characteristics of the NJFET, as in Fig. 7:

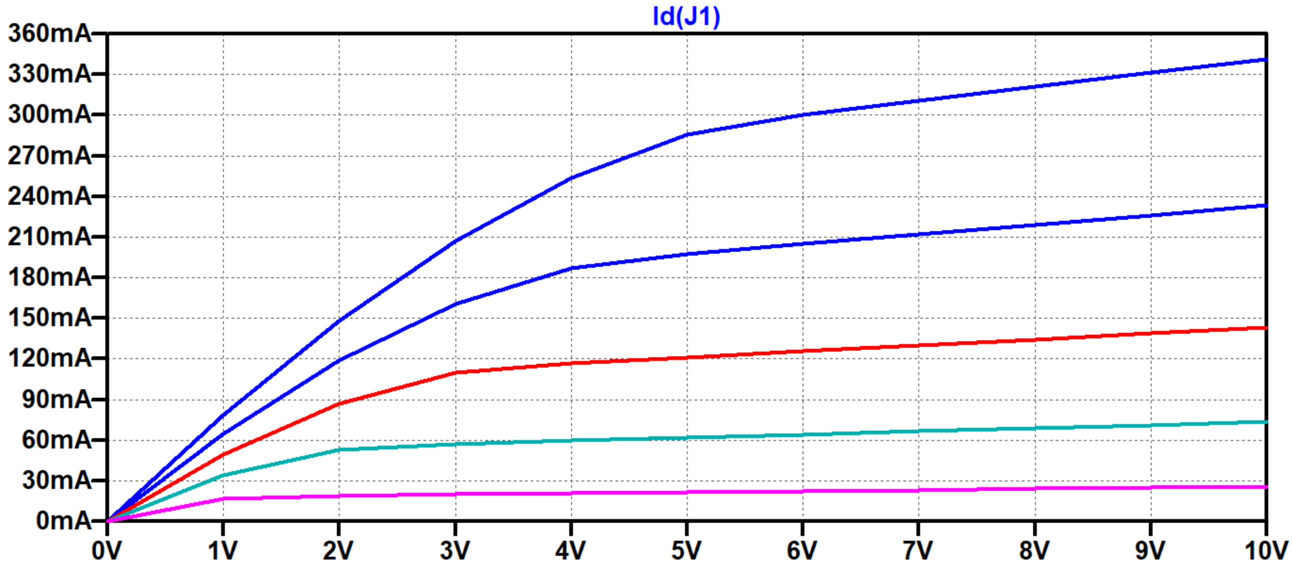


Fig. 7. N-Channel JFET output characteristics

Now you may proceed to the second part of this experiment, repeat the steps but this time for a P-channel JFET. The steps are repeated in Fig. 8 and beyond.

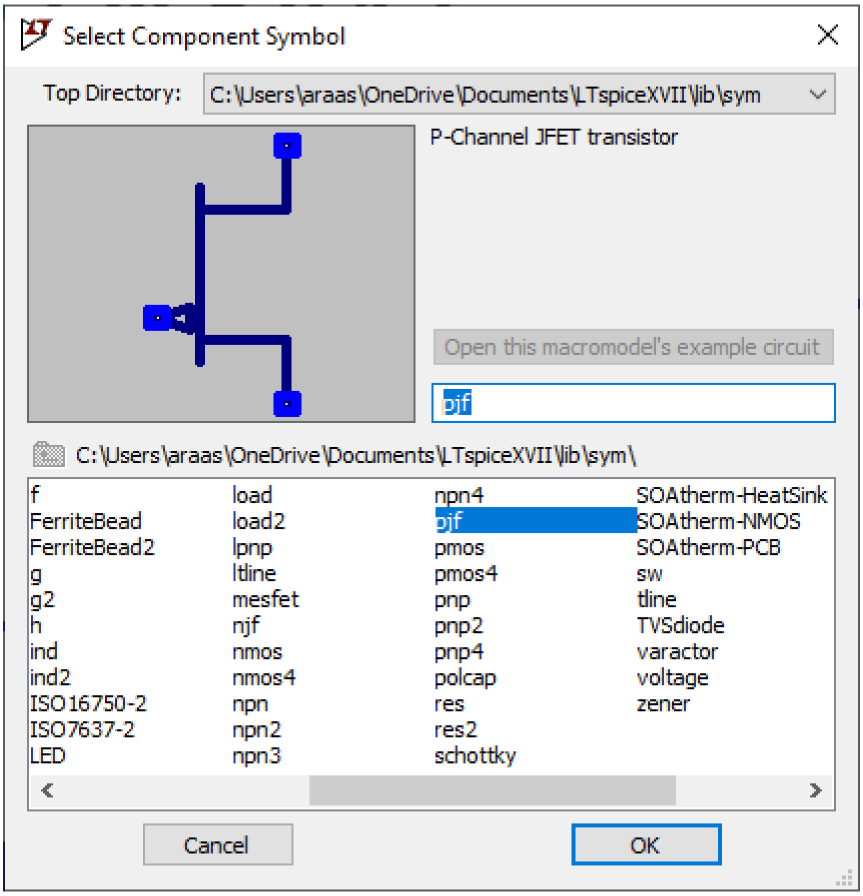


Fig. 8. P-Channel JFET

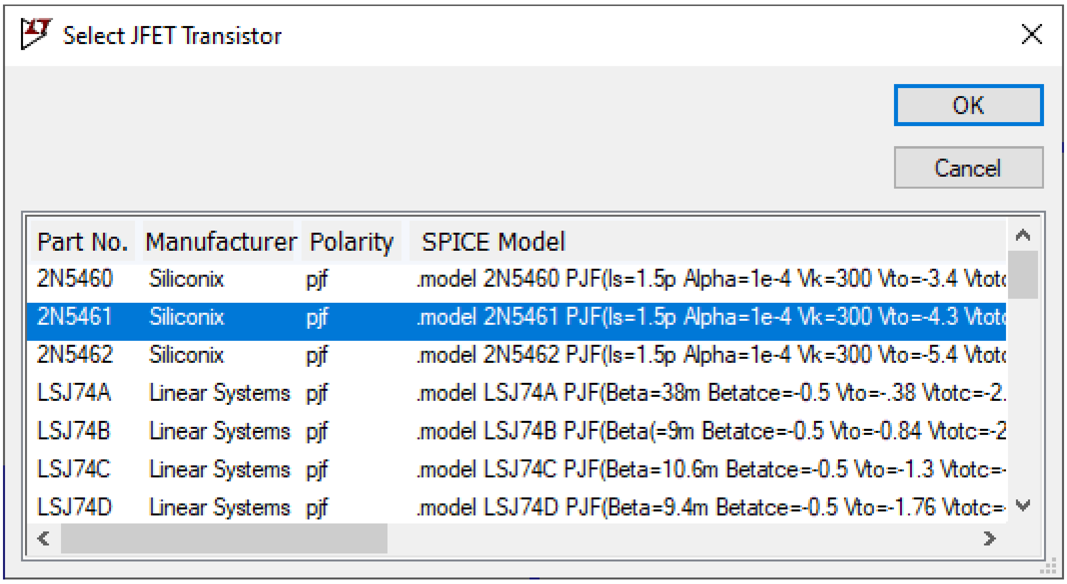


Fig. 9. 2N5461 by Siliconix

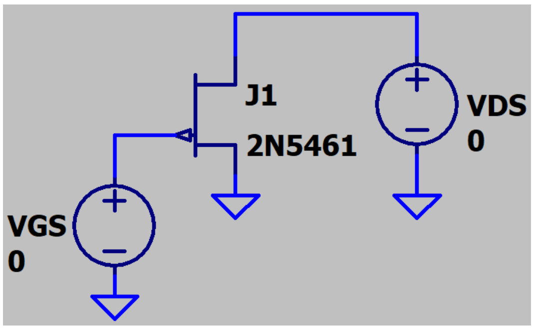


Fig. 10. PJFET testbench

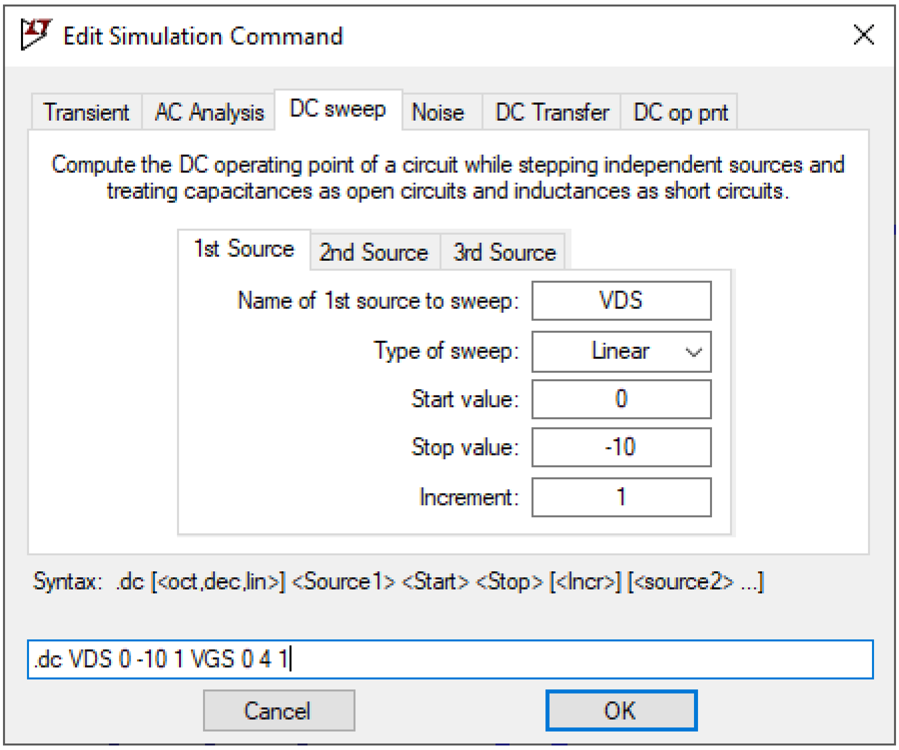


Fig. 11. DC analysis specifications for 1st source

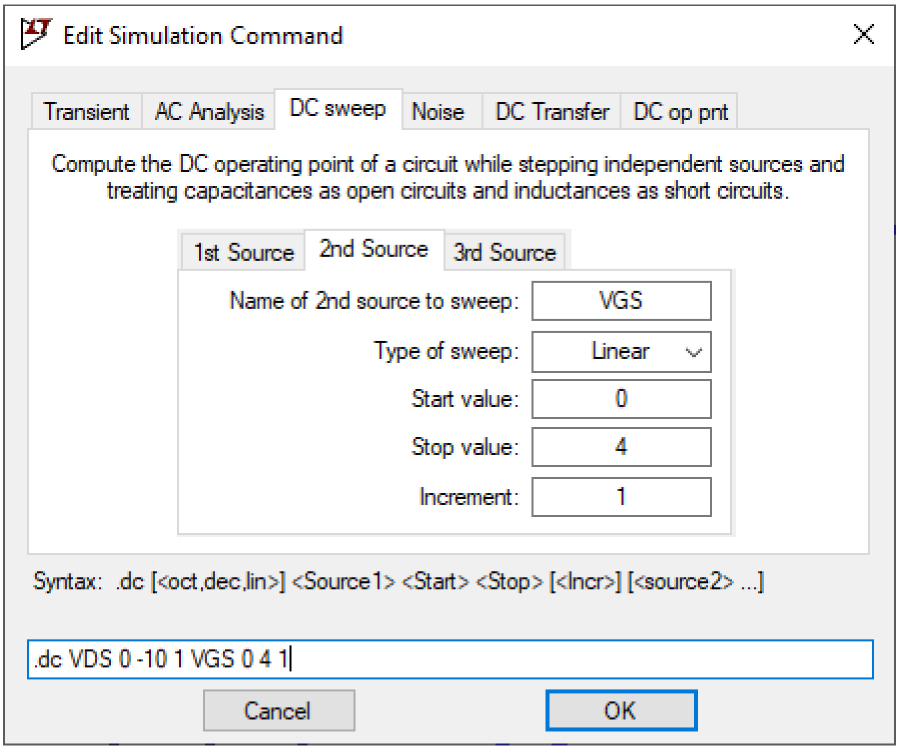


Fig. 12. DC analysis specifications for 2nd source

Note how the output characteristics of the P-channel JFET differs from the N-channel JFET, due to the different polarities.

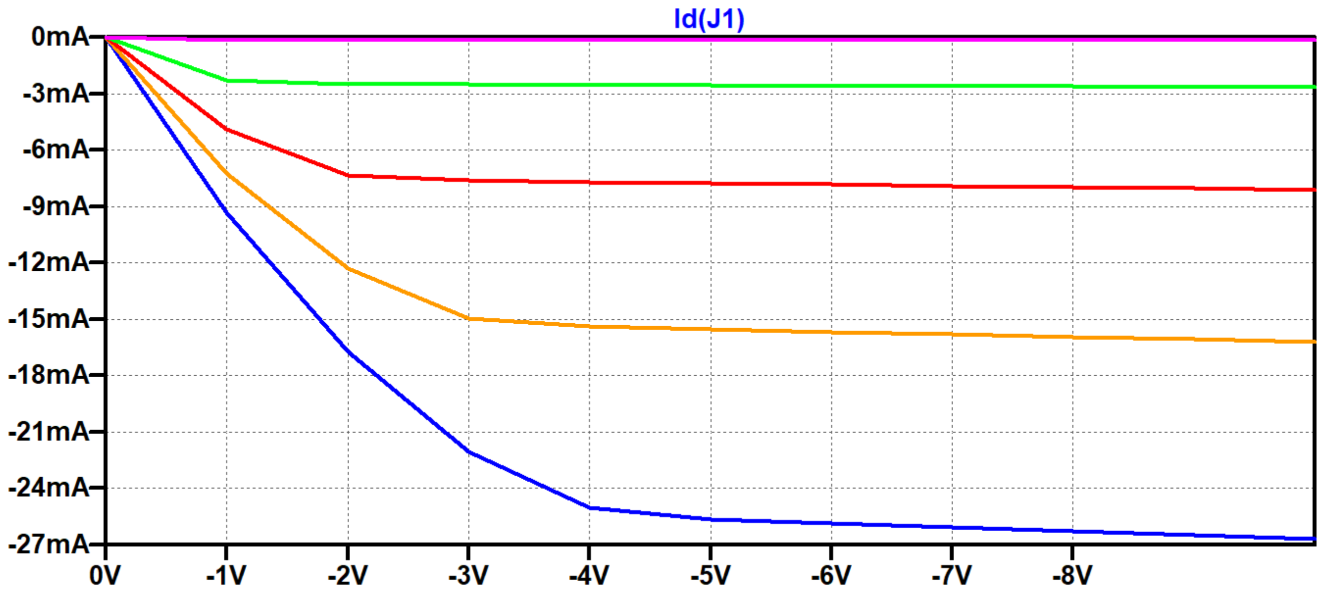


Fig. 13. P-Channel JFET output characteristics

Report requirements:

1. Describe the operation of JFET, add graphs and equations to support the statements.

2. Connect the circuit in Fig. 14, run transient analysis (set **Stop time** as **0.12m**).

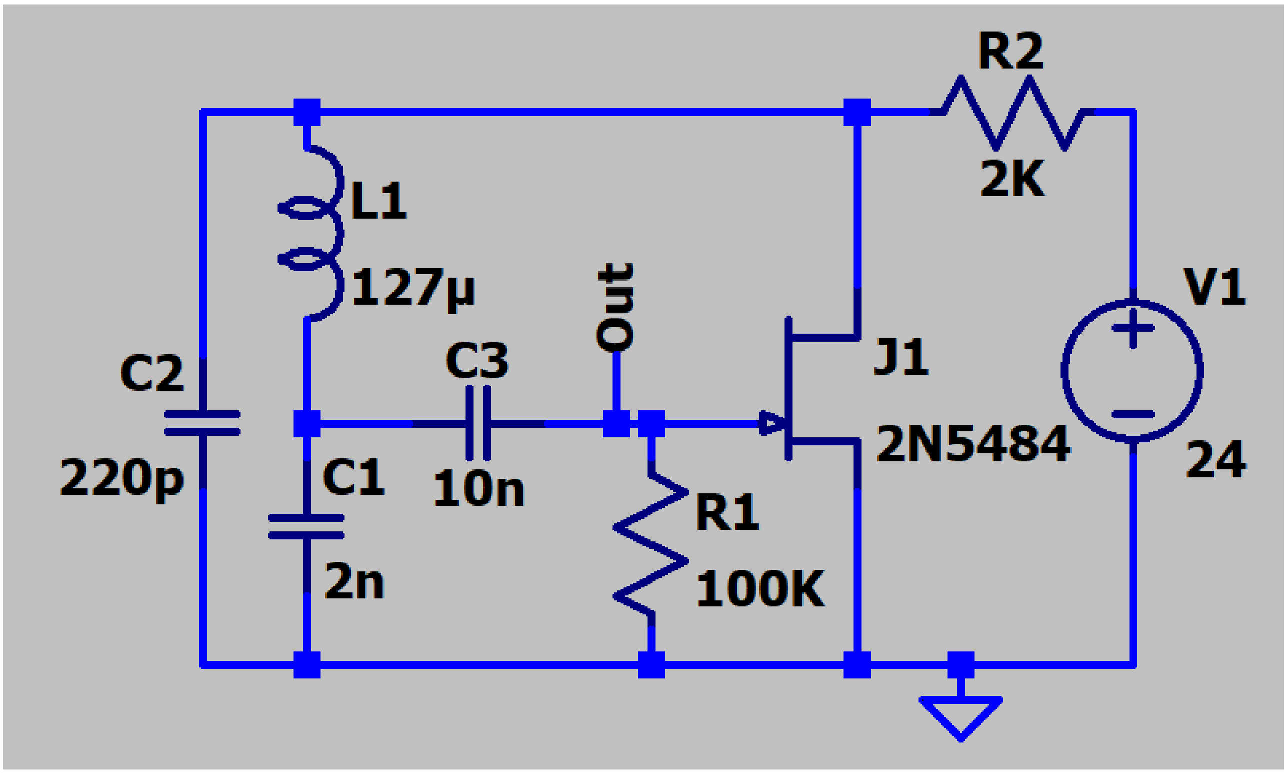


Fig. 14.

Provide a screenshot of the output signal (**Out**), then answer the following questions:

1. Which category does this circuit belong to?
2. Mention some applications of these circuits.
3. Zoom the output signal to see **one period,** then calculate the frequency of the signal (approximately).

Experiment (4)
MOSFET Characteristics

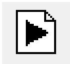
Objectives:

The students will analyze MOSFET's IV characteristics using DC sweep analysis in LTspice.

Introduction:

MOSFET (Metal Oxide Semiconductor Field Effect Transistor) is a type of transistor, which is widely used for switching circuits and in amplifier circuits. MOSFETs are commonly used in integrated circuits, because they can be fabricated in a single chip due to their exceedingly small dimensions/sizes.

Procedure:

1. Create a new schematic 
2. Search for **nmos4** (N-channel MOSFET) as in Fig. 1.

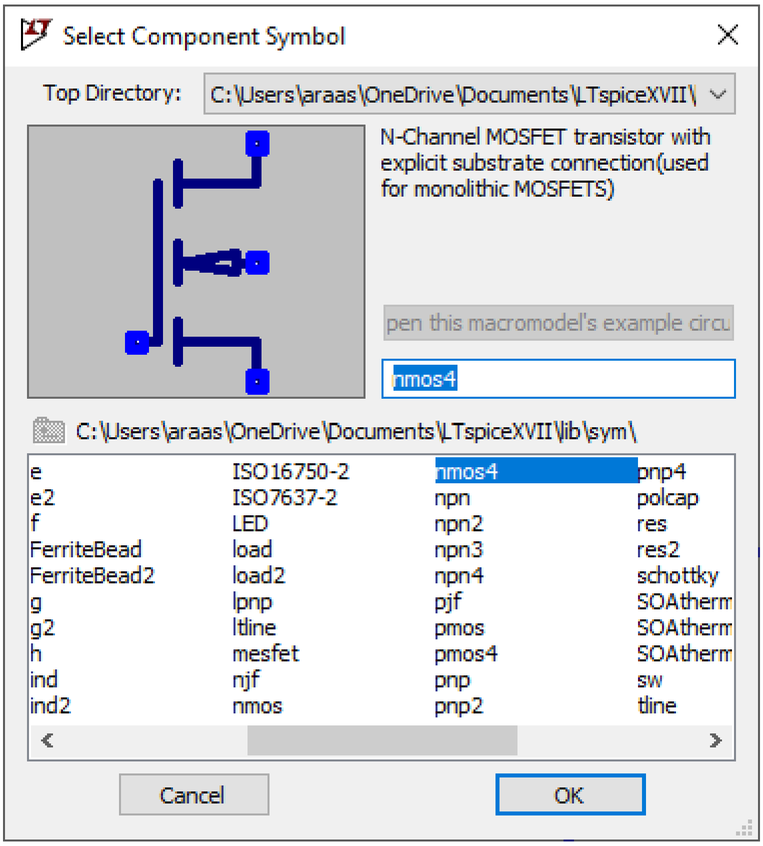


Fig. 1. N-MOSFET in components library

Press OK, right click on the transistor and set the Model Name, Length and Width as in Fig. 2:

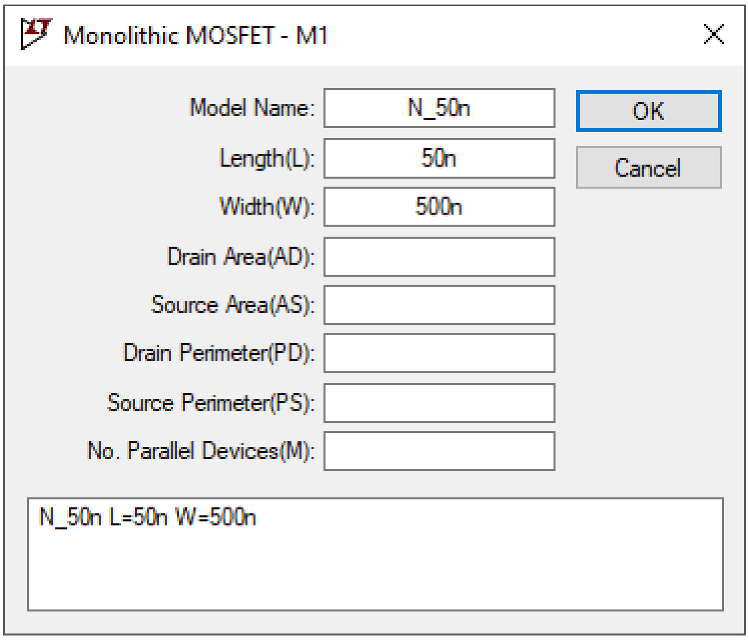


Fig. 2. Transistor Model Name and Sizes

Connect this circuit in Fig. 3 (this will be your testbench):

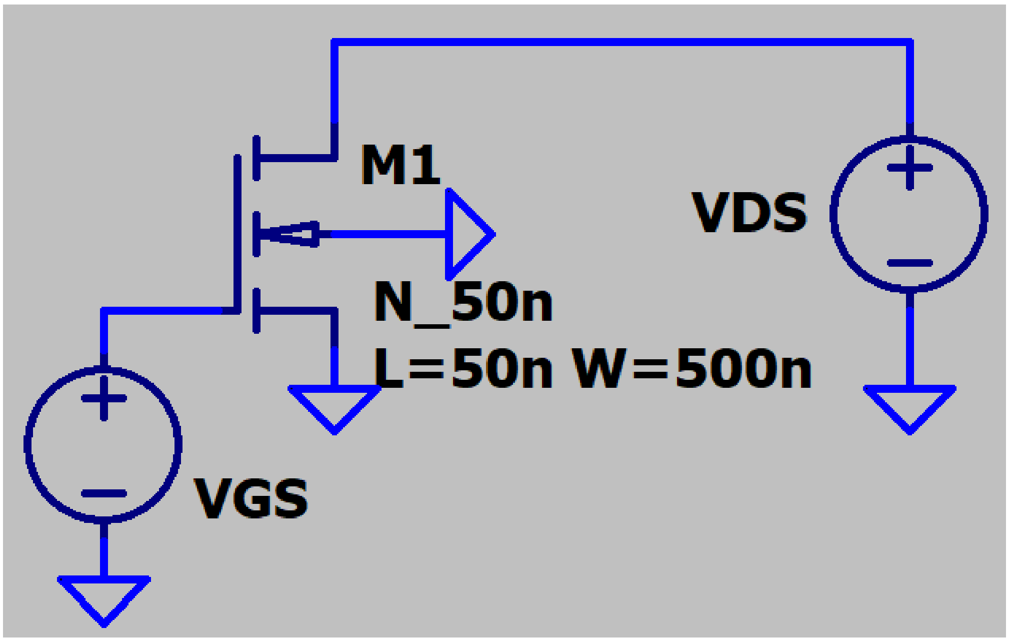


Fig. 3. N-MOSFET testbench

Click on SPICE directive .op, insert: **.include cmos.txt** as in Fig. 4:

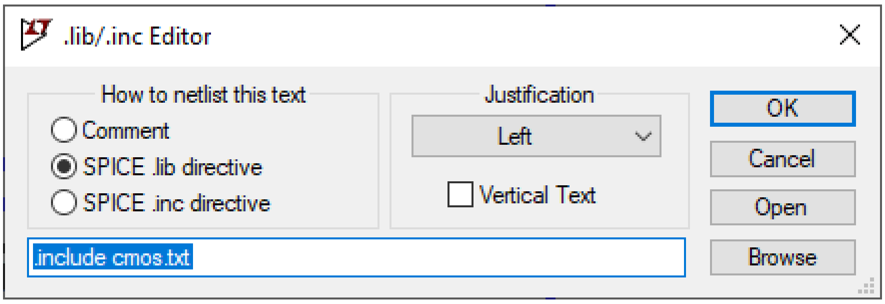


Fig. 4. Including cmos.txt

Run the circuit, set the DC sweep parameters in accordance with Fig. 5 and Fig. 6:

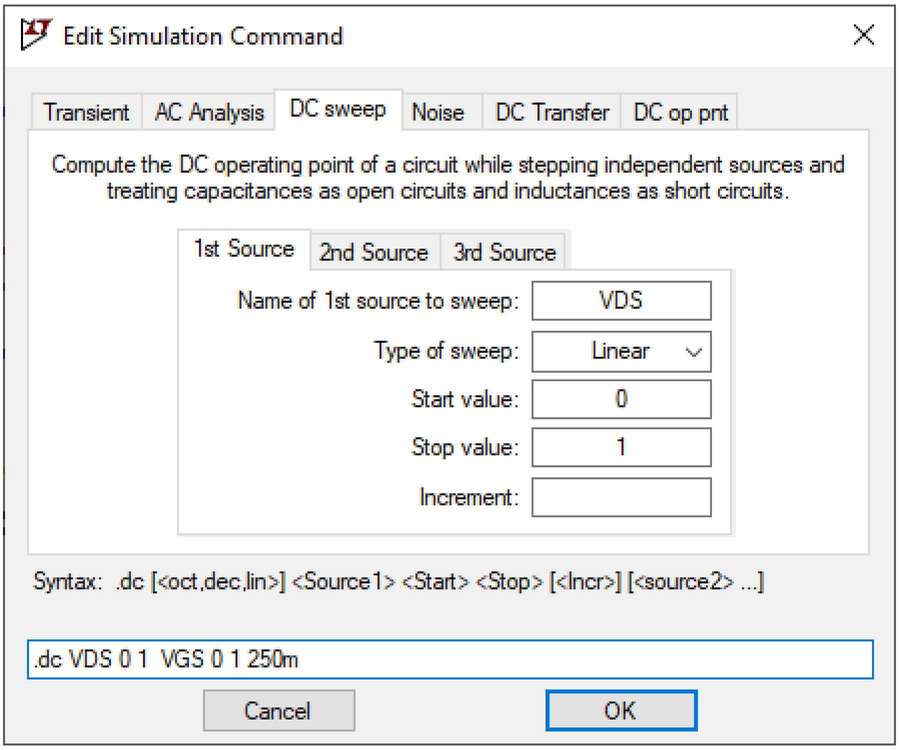


Fig. 5. DC Analysis (1^{st} Source)

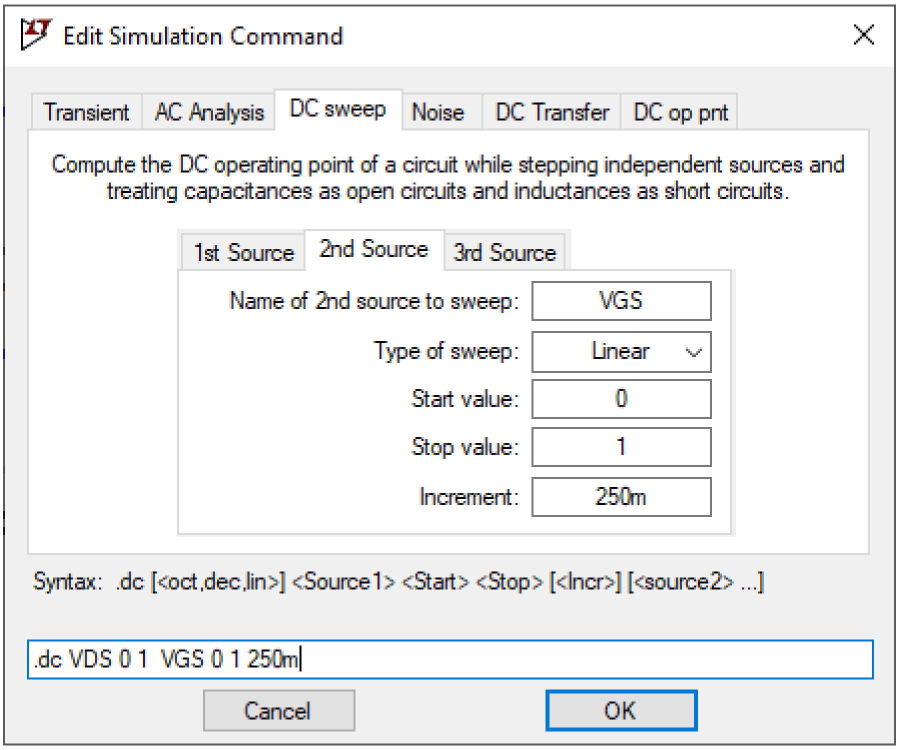


Fig. 6. DC Analysis (2^{nd} source)

Plot the **drain current (Id)**, your results must look exactly like the curves in Fig. 7.

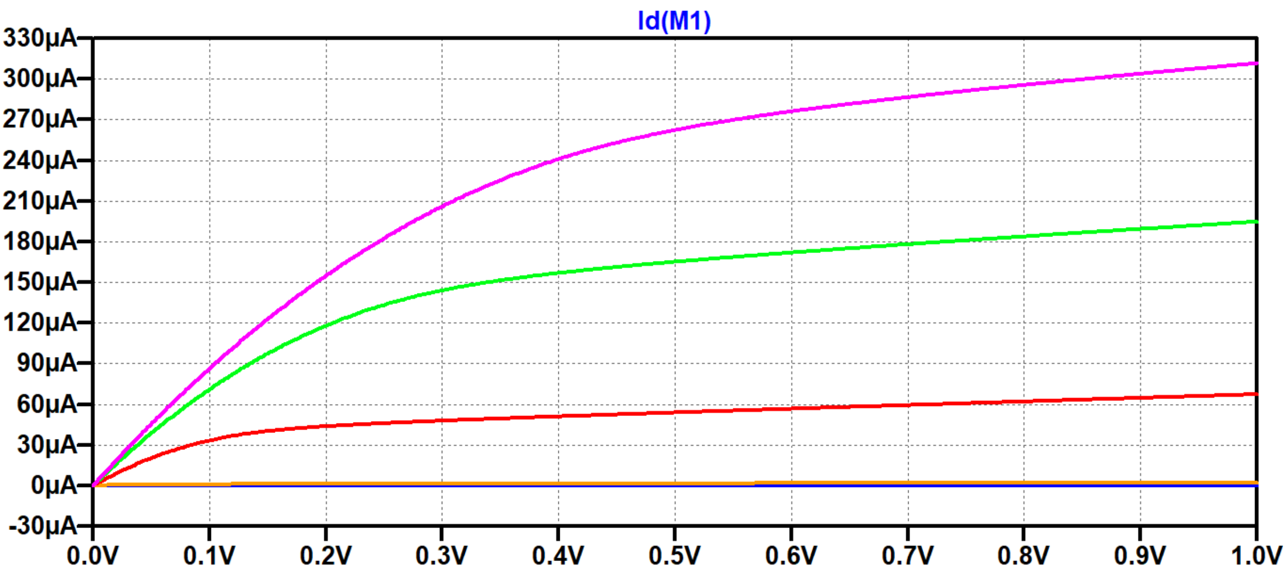


Fig. 7. N-MOSFET IV characteristics

Now we will repeat the previous procedure for a P-MOSFET transistor, follow these steps carefully:

Search for **pmos4** (p-channel MOSFET) as in Fig. 8.

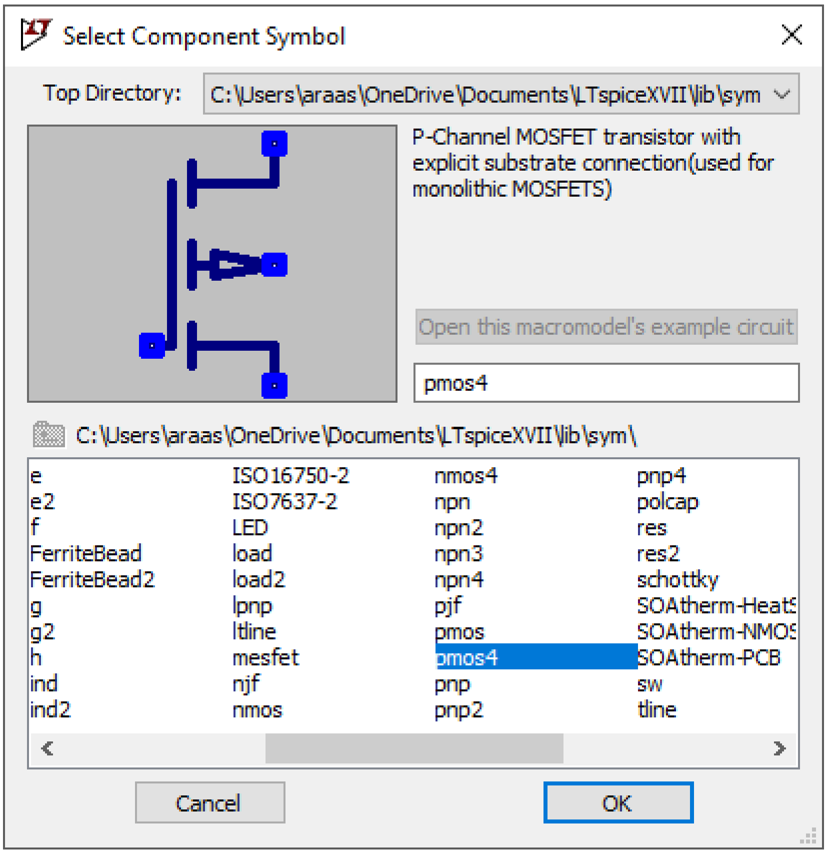


Fig. 8. P-MOSFET in components library

Press OK, right click on the transistor and set the Model Name, Length and Width as in Fig. 9:

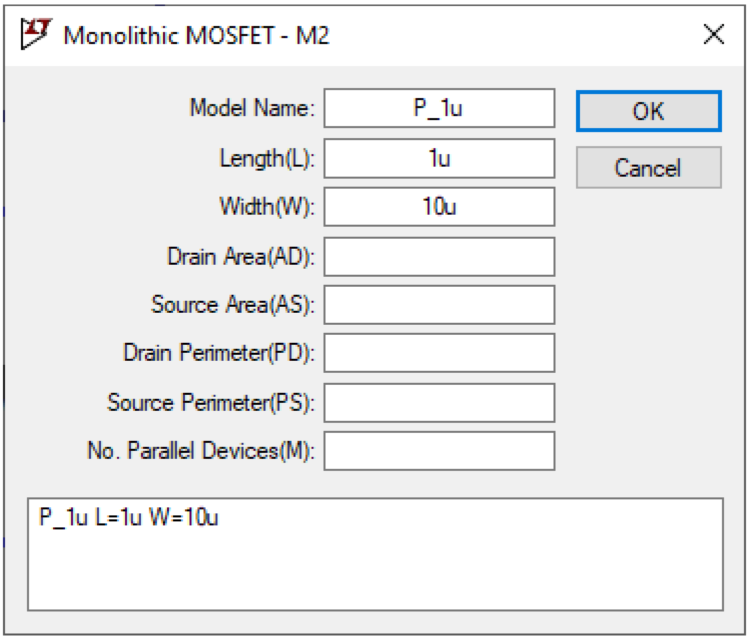


Fig. 9. Transistor Model Name and Sizes

Connect this circuit in Fig. 10 (this will be your testbench):

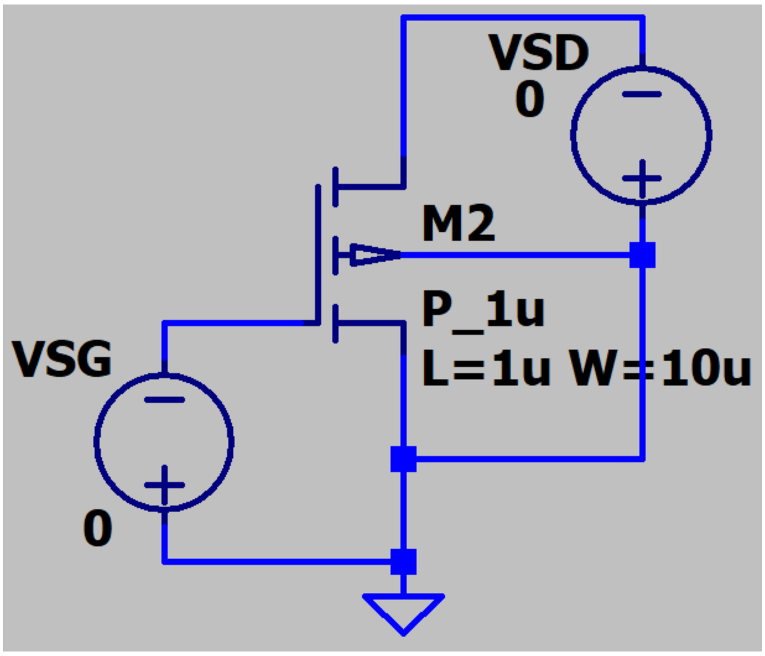


Fig. 10. P-MOSFET testbench

Click on SPICE directive .op, insert: **.include cmos.txt** as in Fig. 11:

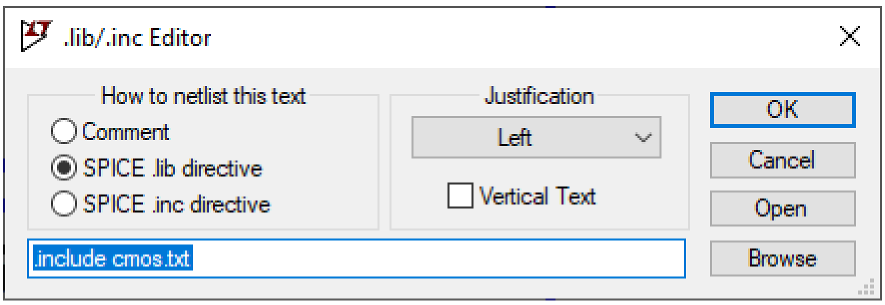


Fig. 11. Including cmos.txt

Run the circuit, set the DC sweep parameters in accordance with Fig. 12 and Fig. 13:

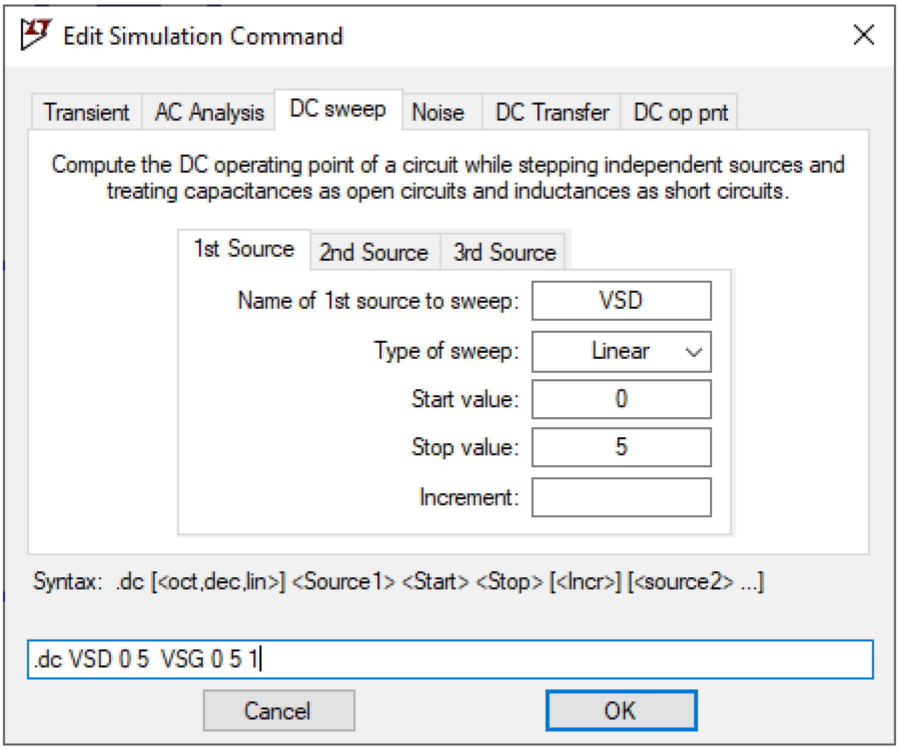


Fig. 12. DC Analysis (1st Source)

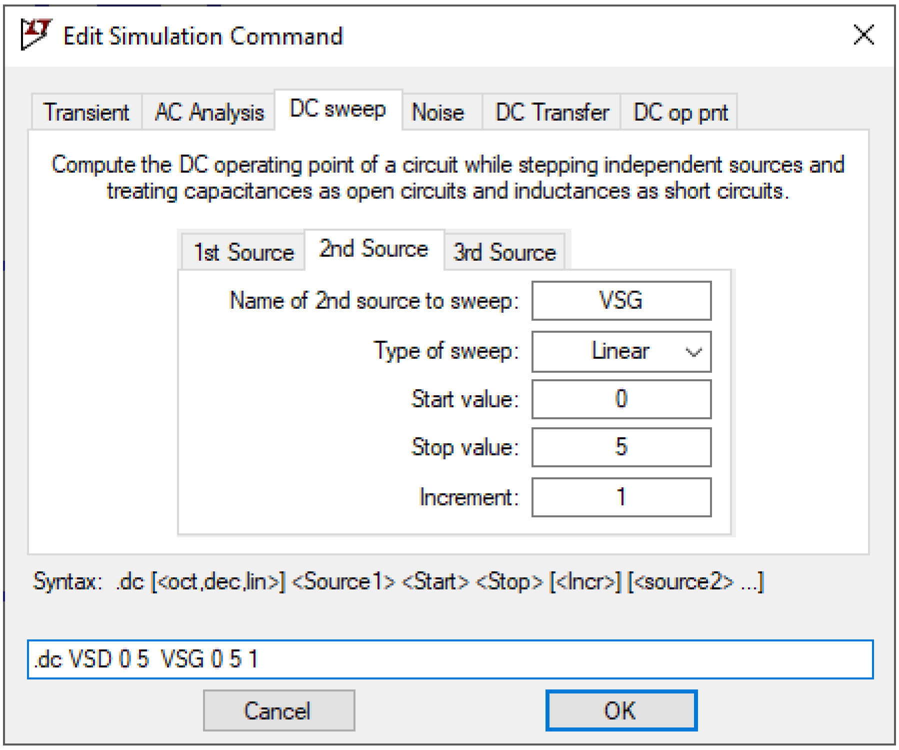


Fig. 13. DC Analysis (2nd source)

Plot the **source current (Is)**, your results must look exactly like the curves in Fig. 14.

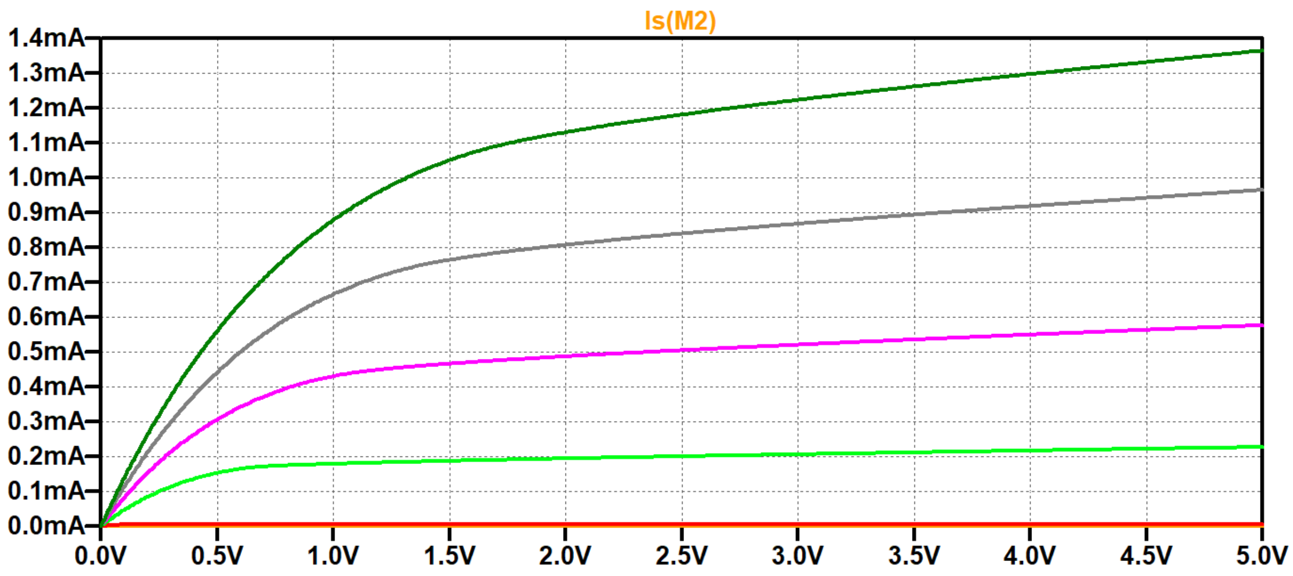


Fig. 14. P-MOSFET IV characteristics

Report requirements:

1. Describe the operation of MOSFET, include graphs and equations.

2. Plot drain current (Id) for the circuit in Fig. 10, how does it differ from the graph in Fig. 14?

3. How does the drain current (Id) behave in the body-effect bias circuit in Fig. 15:

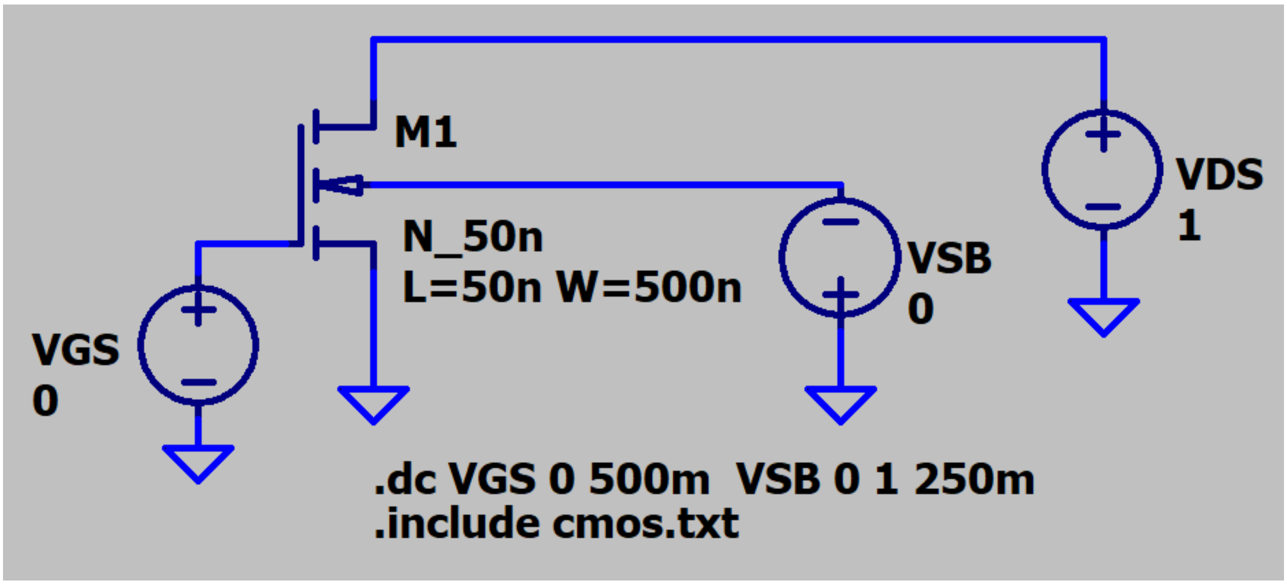


Fig. 15. Body-effect bias

Experiment (5)
Common Source Amplifier

Objectives:

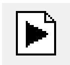
To understand and experiment the performance of common source amplifier (utilizing a voltage-divider bias).

Introduction:

Common source amplifiers are the most widely used amplifiers due to their high-performance characteristics, compared to other amplifier configurations like common drain (source follower) and common gate. They provide a remarkably high power gain, medium input and output resistance, medium current and voltage gain.

They are called common source because the source terminal is common between the input and the output (the input signal is applied at the gate terminal of the transistor and the output is taken from the drain terminal). The output of common source amplifier is 180^{o} out of phase with the input signal.

Procedure:

1. Create a new schematic 
2. Search for **njf** (N-channel JFET) as in Fig. 1.

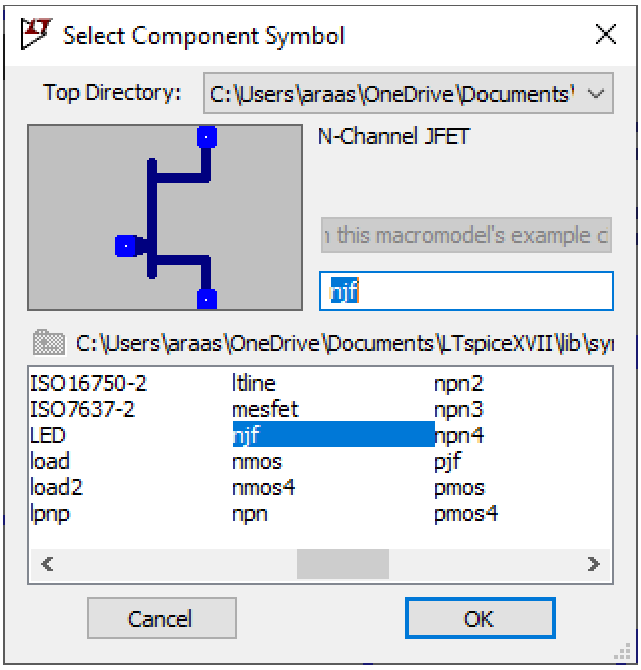


Fig. 1. njf in component library

3. Right click on the transistor, pick **Pick New JFET,** then choose the following transistor model 2N5434 as illustrated in Fig. 2.

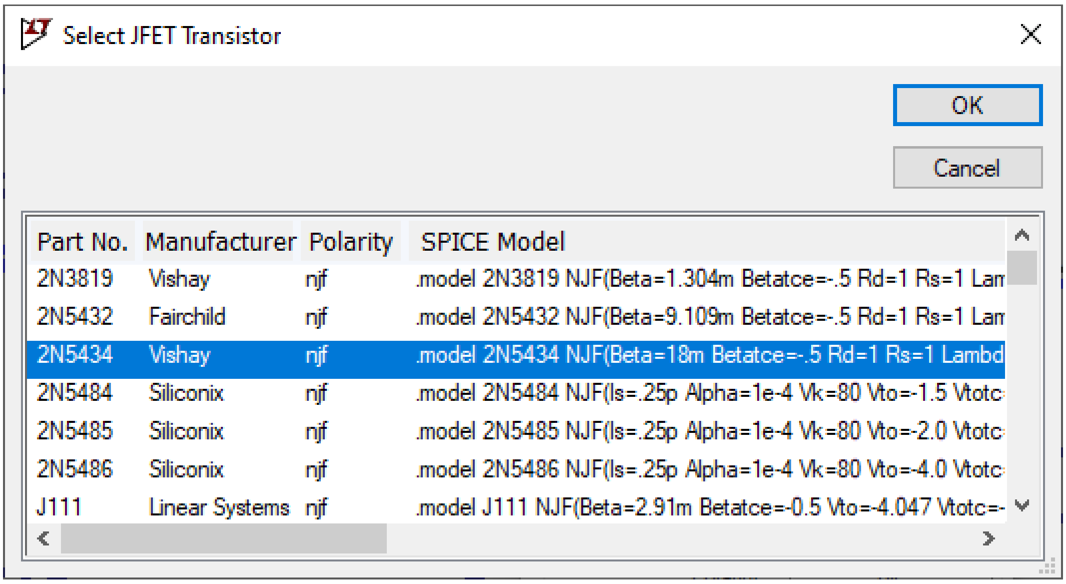


Fig. 2. JFET transistor model

4. Assemble the amplifier circuit in Fig. 3.

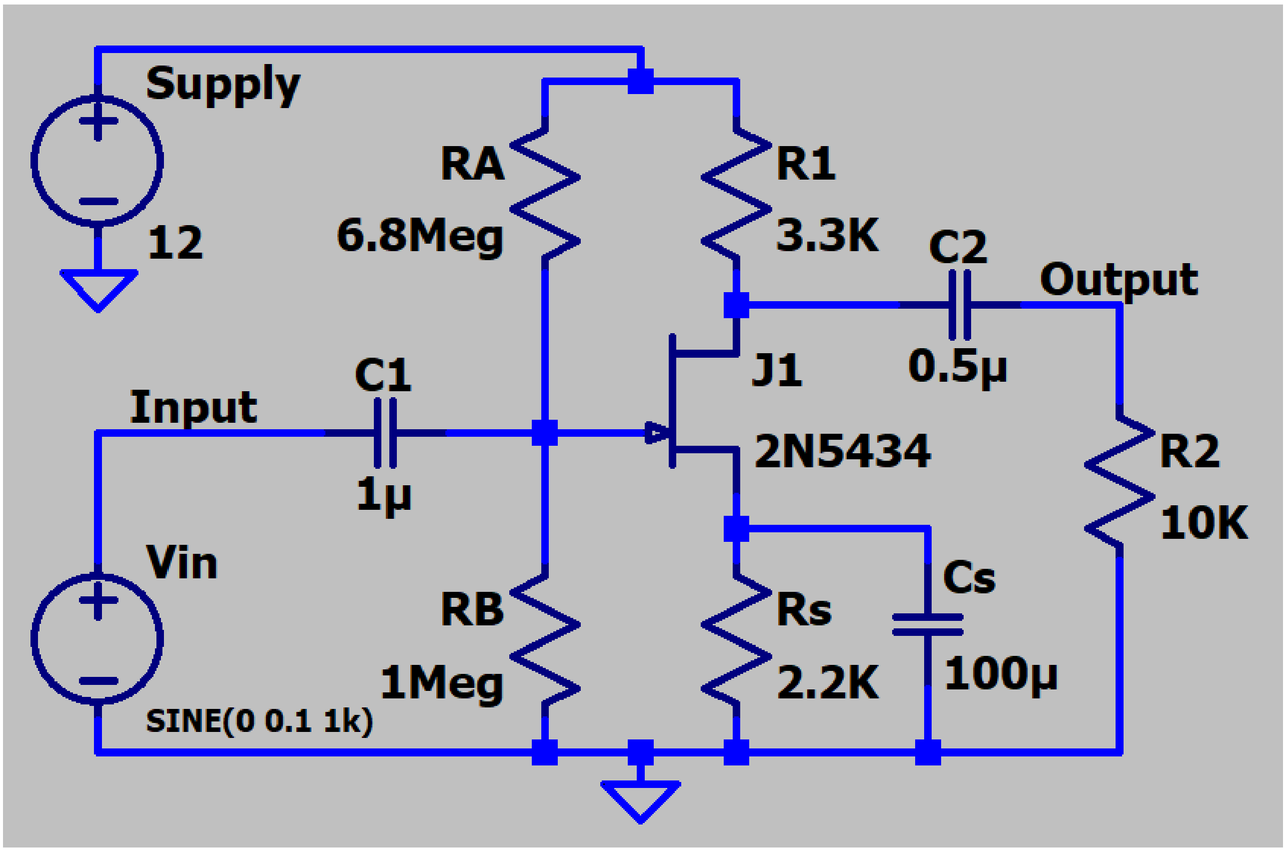


Fig. 3. JFET Common Source Amplifier

5. Set the input voltage signal (Vin) parameters in accordance with the specifications mentioned in Fig. 4:

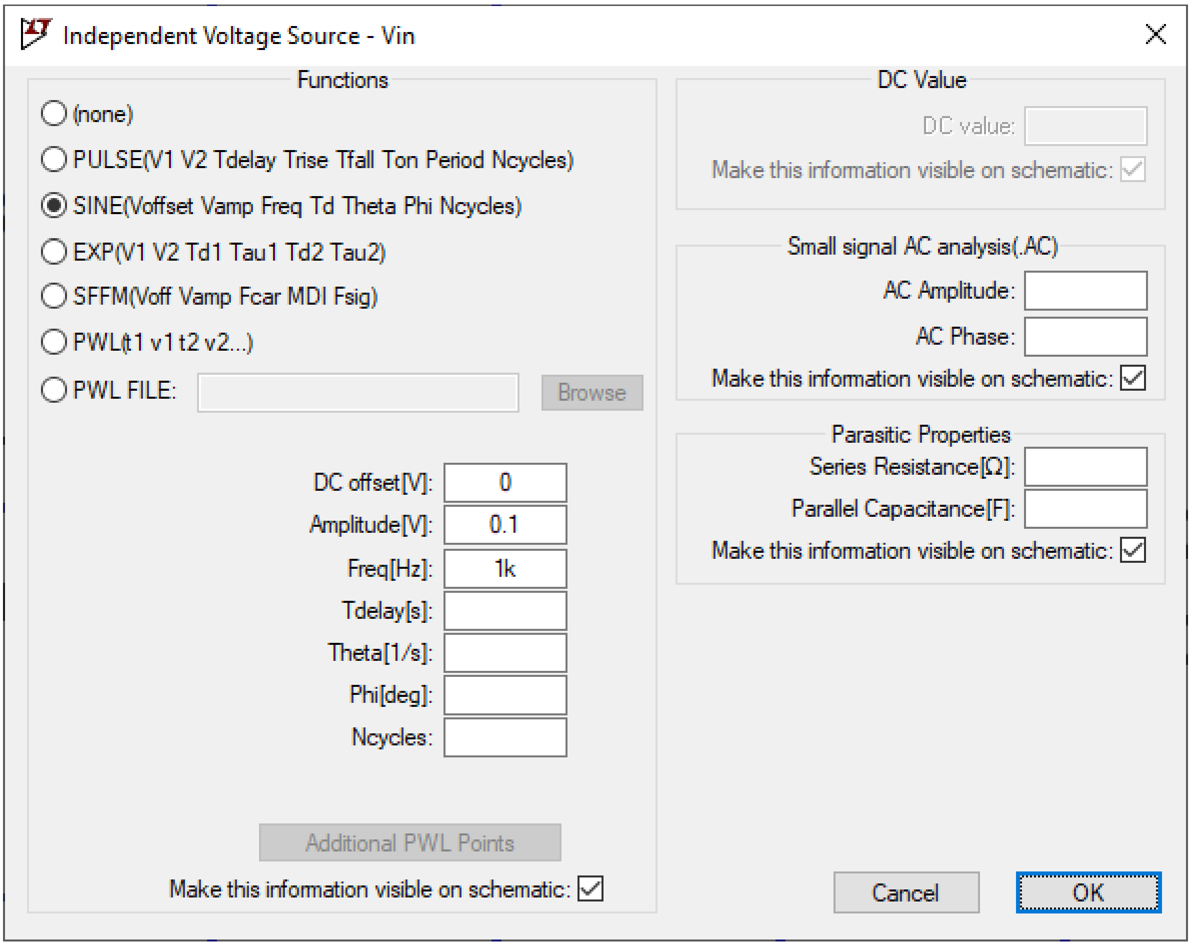


Fig. 4. JFET Common Source Amplifier

6. Run the circuit by clicking on , choose **Transient,** set Stop time: 10m, click OK, then plot the input and output voltage signals.

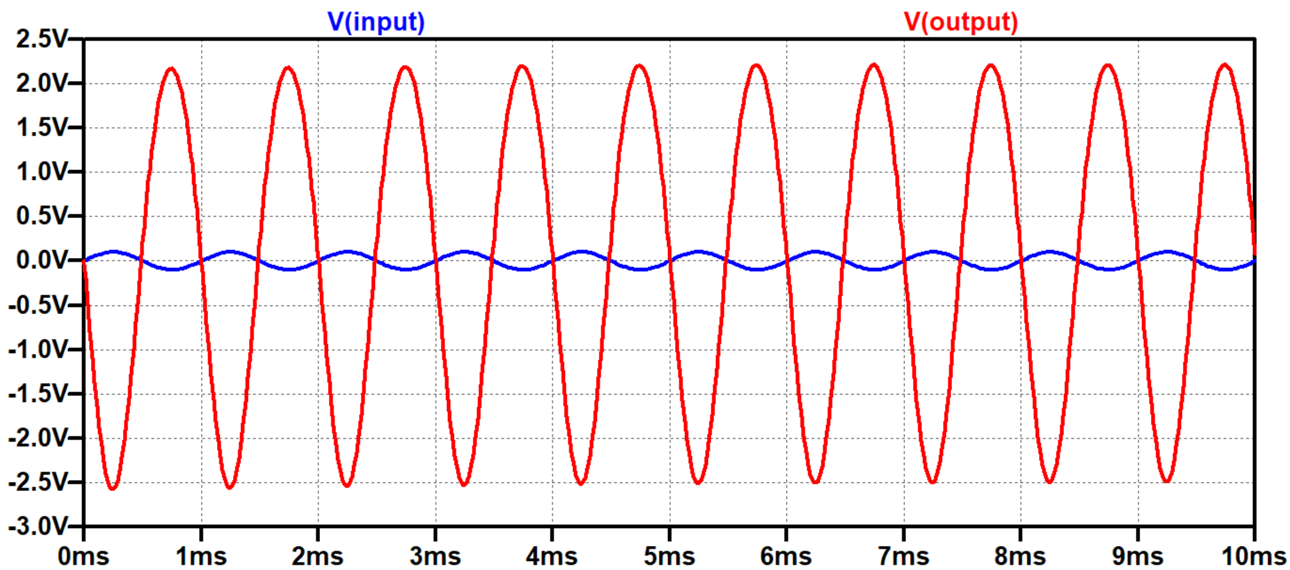


Fig. 5. Input & output voltage signals of the JFET common source amplifier

Report requirements:

1. What are the advantages of common source amplifiers over common drain & common gate amplifiers?

2. Where are common source amplifiers used? Mention 5 of their applications.

3. Plot the frequency response of the amplifier in this experiment (refer to the two figures below), do not forget to set the AC Amplitude as 1.

Include the plot in your report, how much is the gain & phase shift at **10 kHz**?

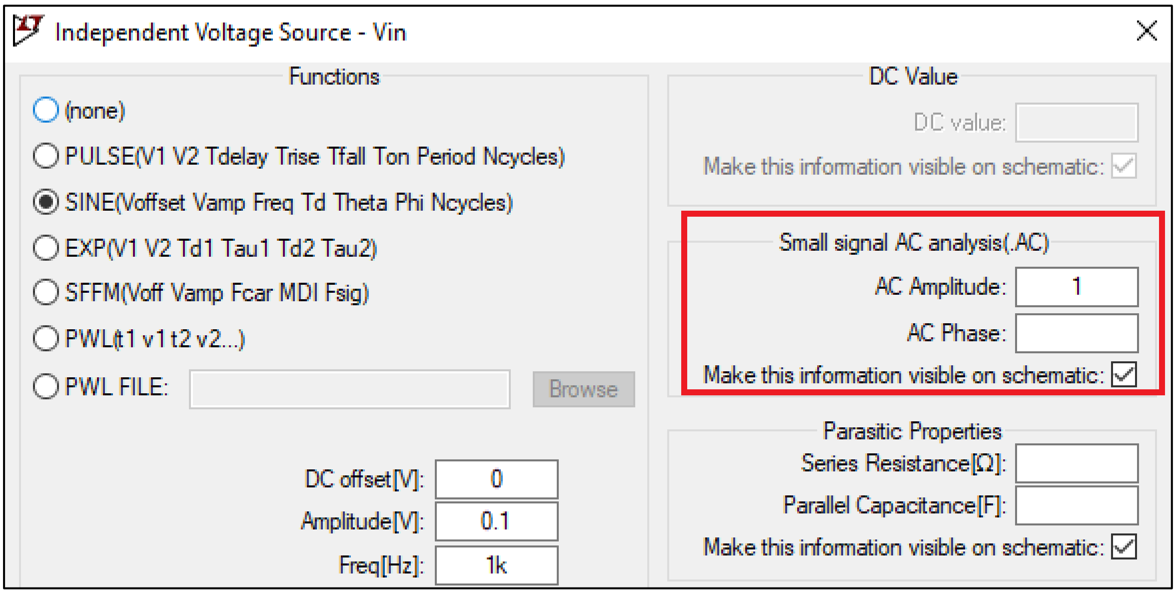


Fig. 6. AC Amplitude

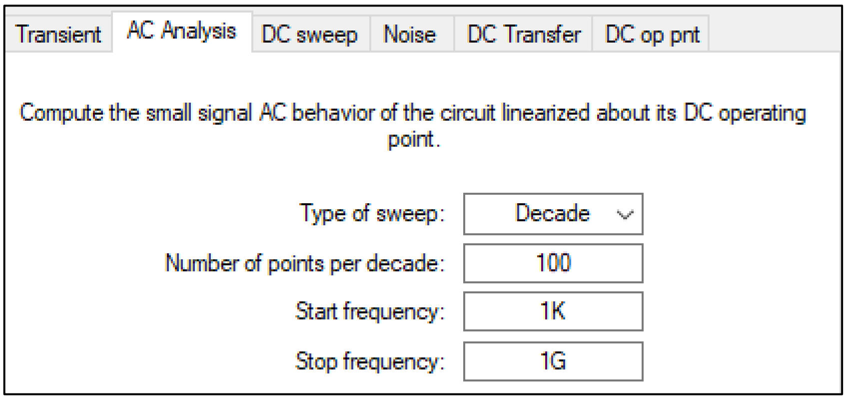


Fig. 7. AC Analysis specifications

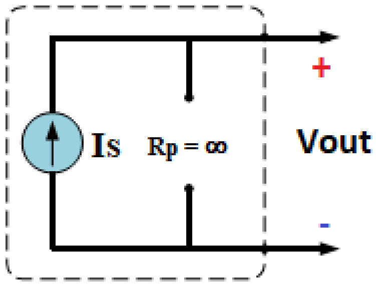
Experiment (6)
Current Sources

Objectives:

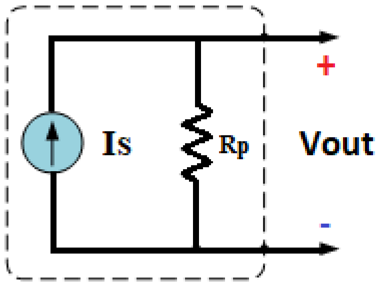
To understand the working principle of current sources and implement a practical current source circuit.

Introduction:

A current source is a circuit that supplies a constant current flow regardless of the impedance that it is driving, the difference between an ideal and a practical current source is illustrated below:



Ideal Current Source



Practical Current Source

Procedure:

1. Search for **nmos** (N-channel MOSFET) as in Fig. 1.

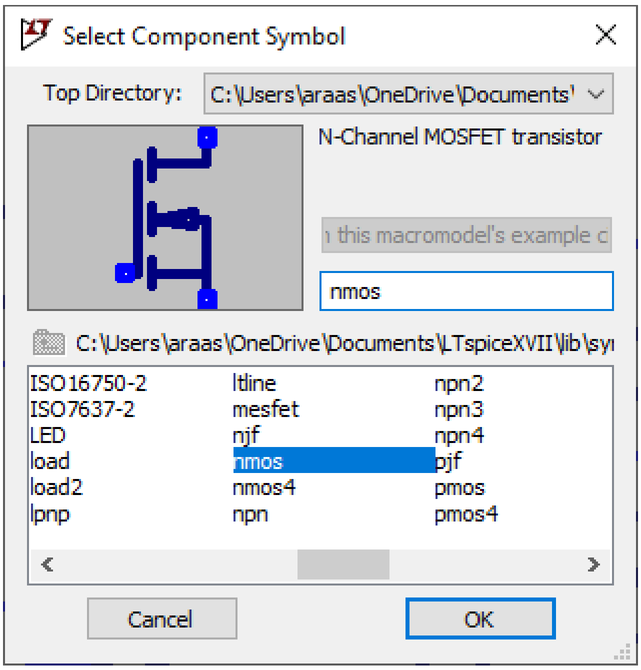


Fig. 1. NMOS transistor

2. Assemble this circuit in Fig. 2, run transient analysis (set Stop time to 1).

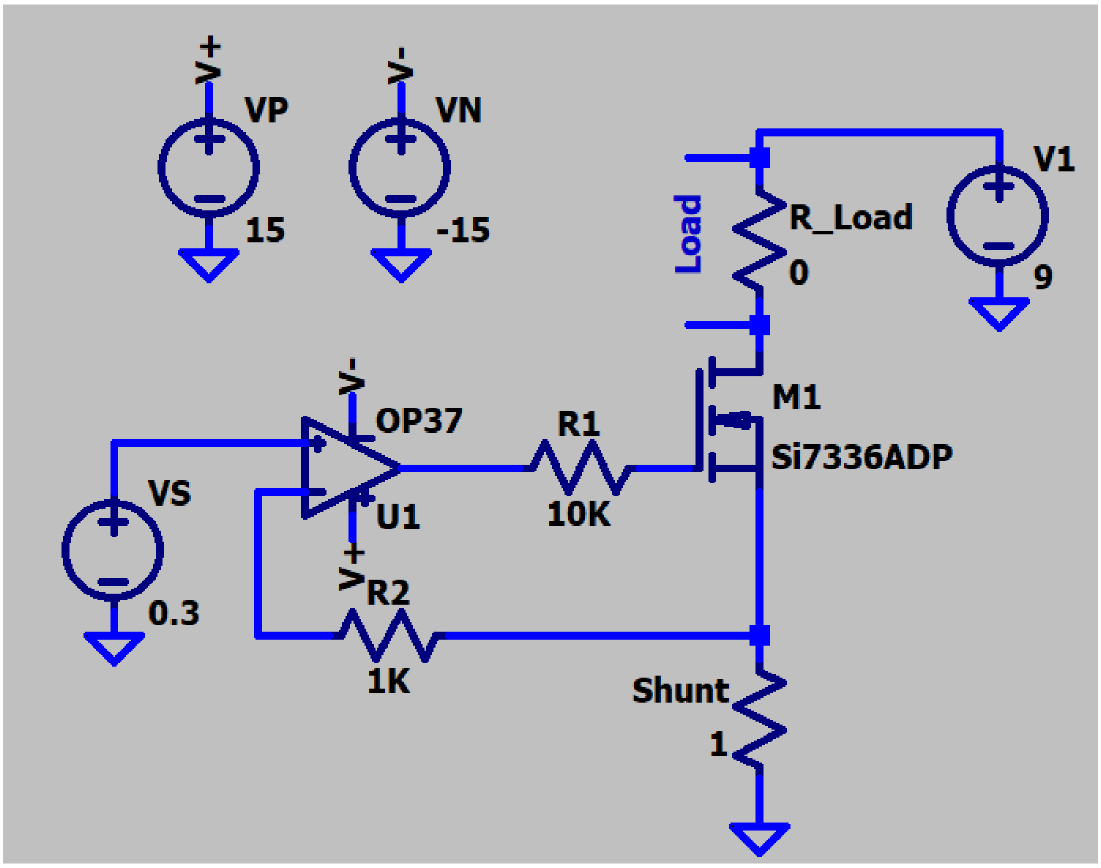


Fig. 2. Voltage–Controlled Current Source

Change the input VS source's voltage according to this table and record the current across the load resistor (R_load).

VS	I_{R_load} (mA)	What can you conclude?
0		
0.1		
0.25		
0.5		
0.75		
1		

Report Questions:

1. Write some applications of current sources.
2. Can current sources be connected in series? Or in parallel? Prove your answer using some circuit diagrams.
3. Does the current of a **practical** current source change as load impedance increases? Prove your answer with graphs/equations.

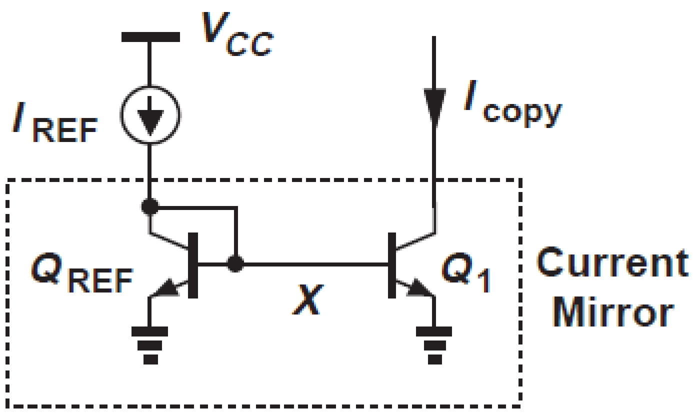
Experiment (7)
Current Mirrors

Objectives:

Students will learn how to design basic current mirrors for various applications.

Introduction:

Current mirror is a circuit that copies the current in a device and controls the current in another device by maintaining the output current stable. Current mirrors function as a more practical current source, they are widely used to bias currents to circuits. They come in different forms and complexities, the simplest current mirror circuit uses only two transistors (BJT or FET), as shown below:



Procedure:

1. Add two **nmos4** transistors, mirror the transistor on the left as in Fig. 1.

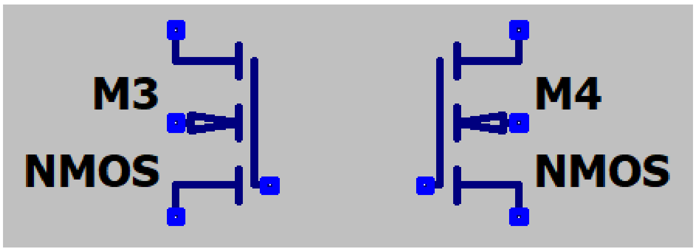


Fig. 1. NMOS4 transistors

2. Click on SPICE directive .op, write: **.include cmos.txt** as in Fig. 2:

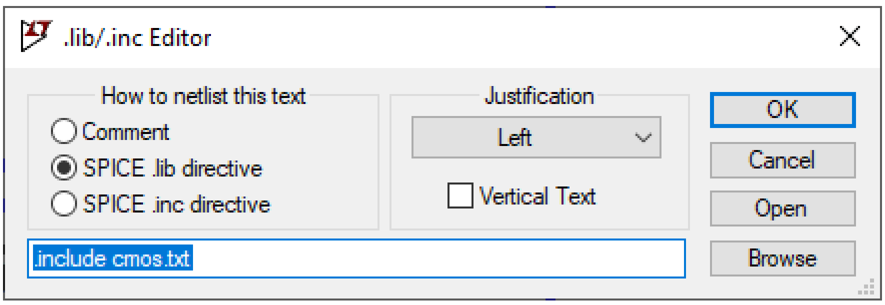


Fig. 2. Including cmos.txt

3. Right click on each of the transistors, set their parameters as illustrated in Fig. 3.

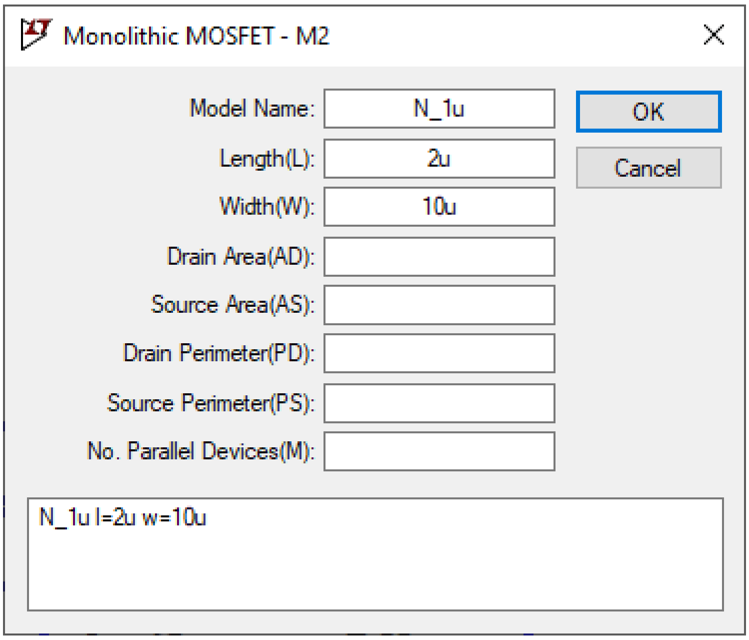


Fig. 3.

4. Connect the current mirror circuit in Fig. 4:

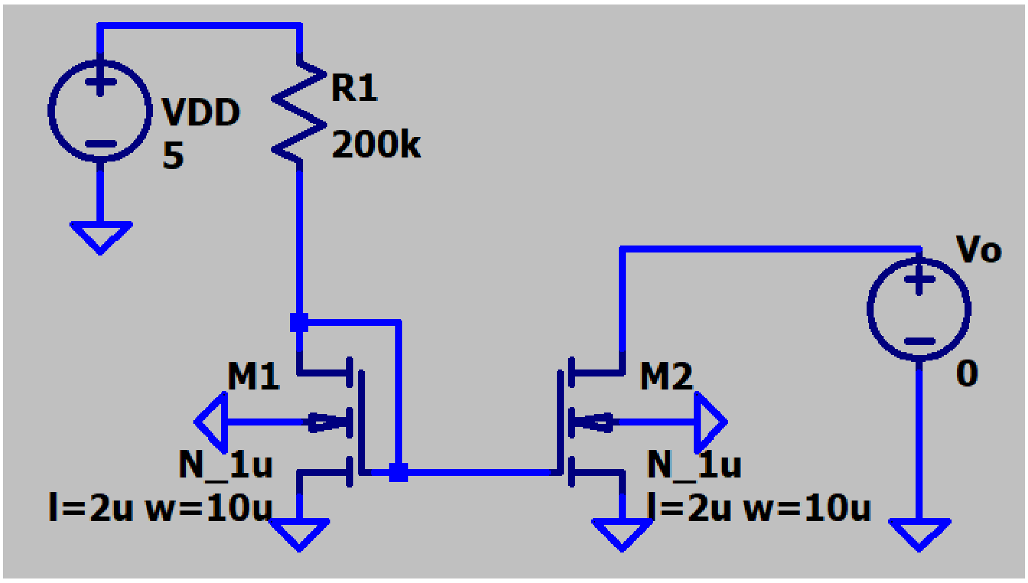


Fig. 4. A simple MOS current mirror

5. Run DC sweep analysis with the specifications in Fig. 5:

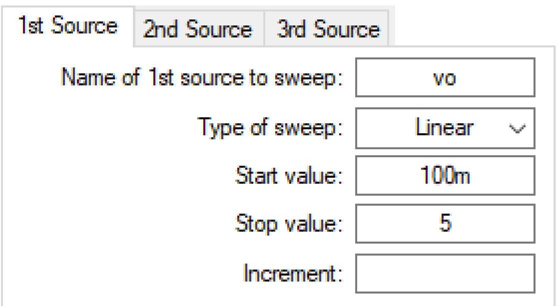


Fig. 5. DC analysis

Plot drain currents of both transistors:

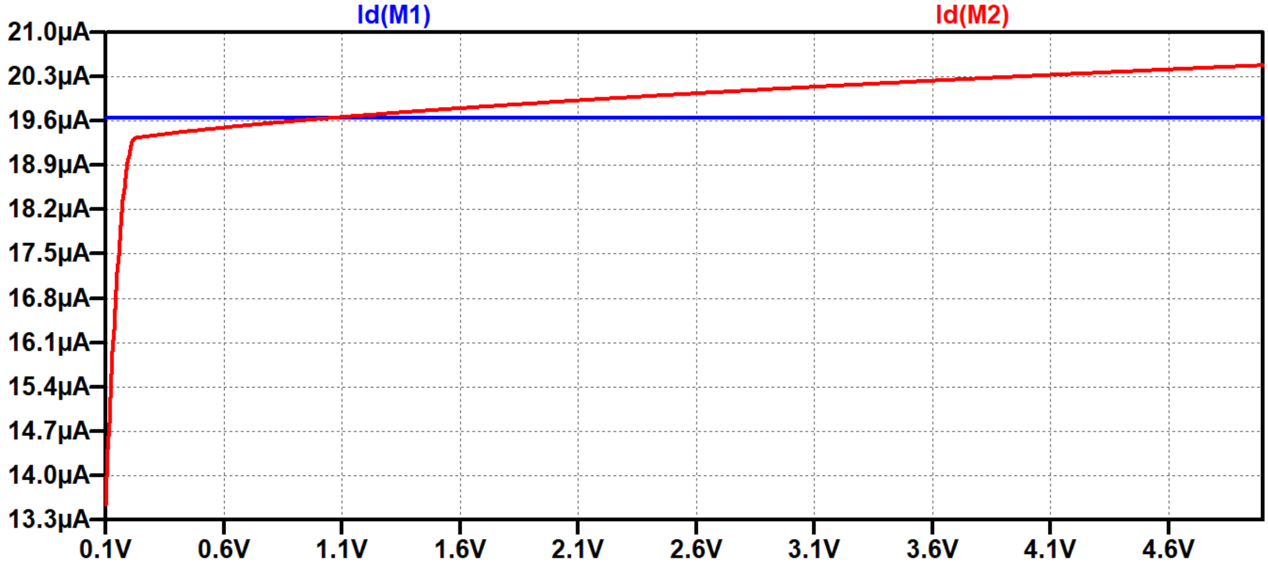


Fig. 6. Drain currents of Transistors 1 & 2

The second part of this experiment is about Wilson Current Mirror which is an enhanced circuit configuration designed to provide a more constant current source. Refer to Fig. 7, connect the circuit using the same transistors and voltage sources (you may create copies of them).

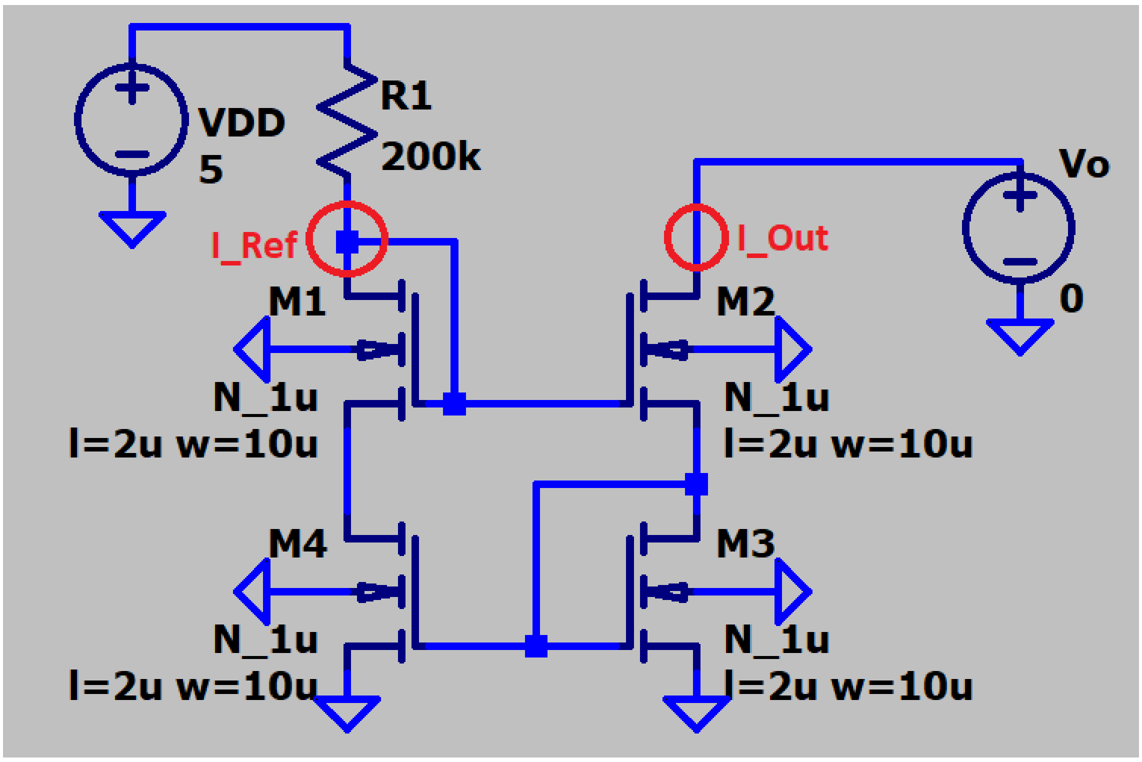


Fig. 7. Wilson Current Mirror

Run this circuit with the exact same DC sweep parameters as in the first circuit, Fig. 8 depicts how the drain currents overlay each other (unlike in the previous current mirror).

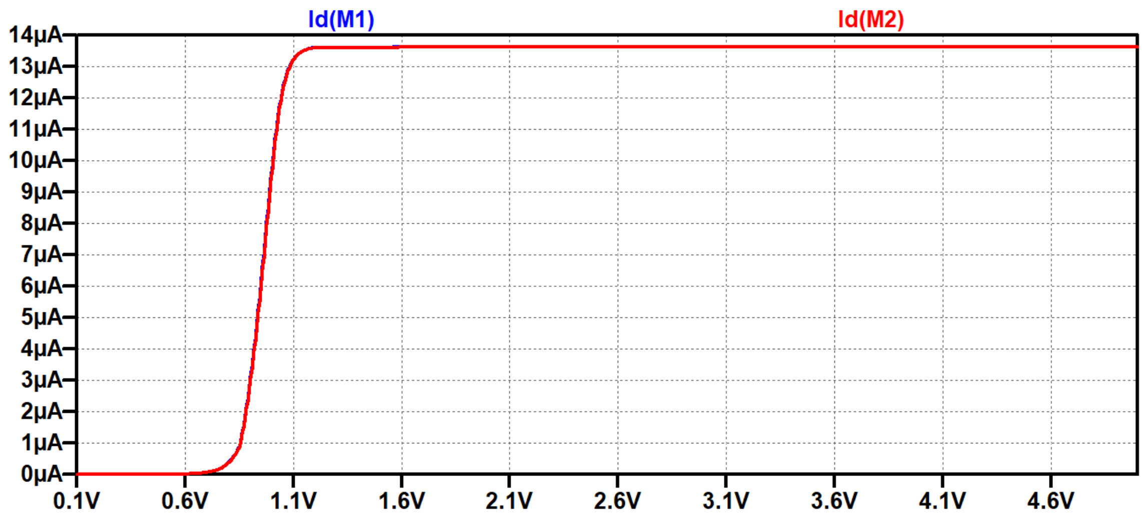


Fig. 8. Drain currents for Wilson Current Mirror

Report Questions:

1. What are some applications of current mirrors?
2. Why does Wilson current mirror perform better than a simple current mirror?
3. Plot collector currents (IC1 & IC2) of both BJT transistors in the current mirror in Fig. 9, use DC sweep analysis (**Vo** source) from **100m to 5.**

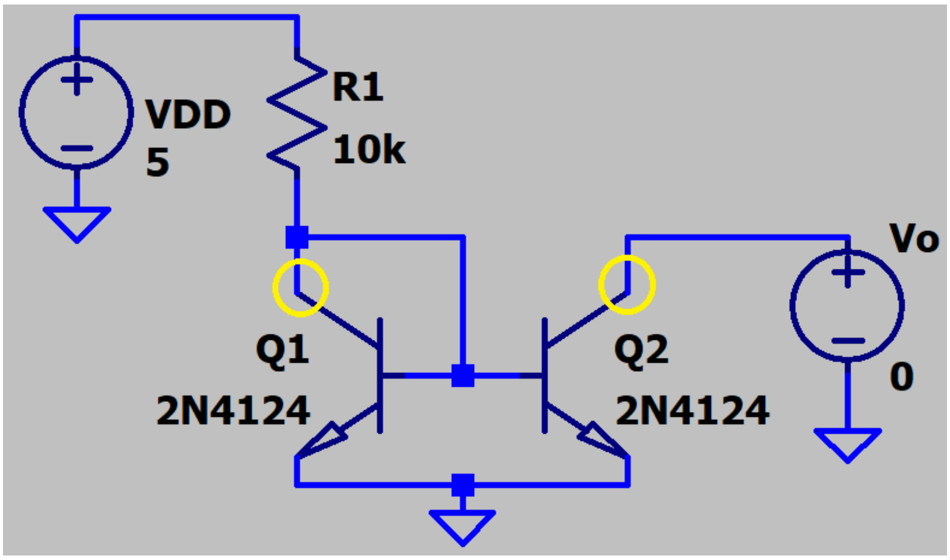
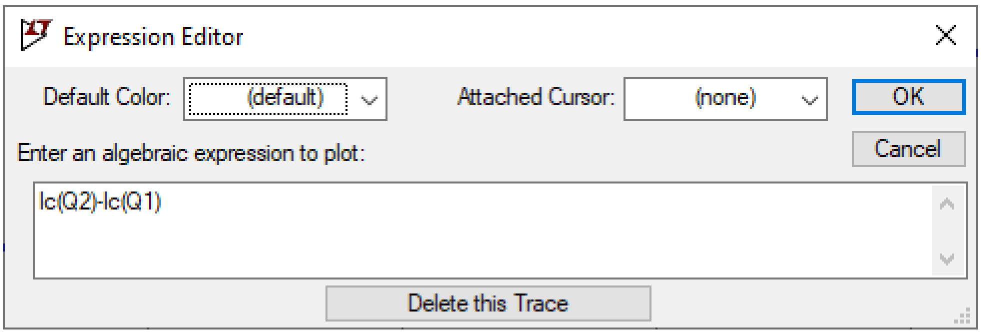


Fig. 9. BJT current mirror

4. For the circuit in Fig. 9, plot the algebraic expression: Ic(Q2) – Ic(Q1)



Experiment (8)
Differential Amplifiers

Objectives:

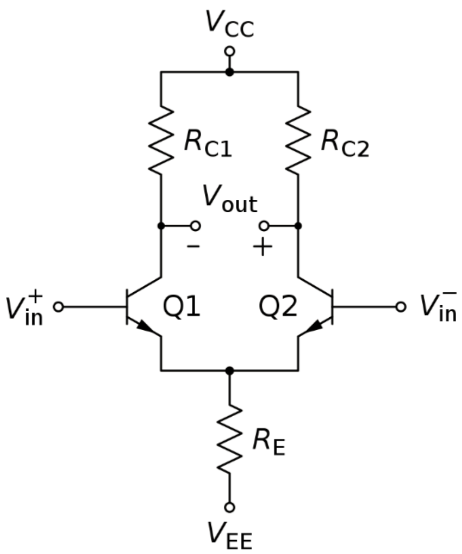
To analyze and design simple differential amplifiers.

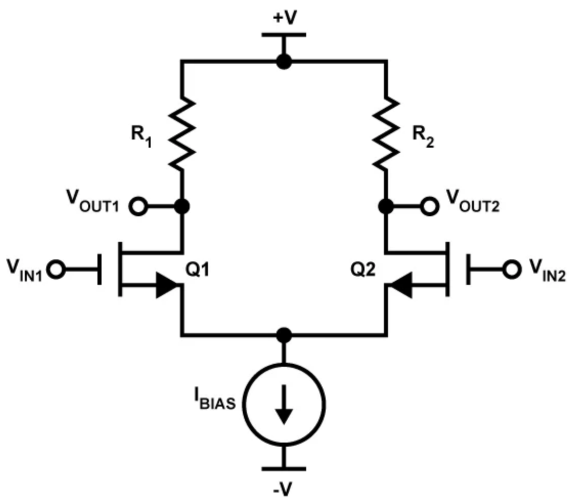
Introduction:

A differential amplifier (also known as a difference amplifier) is a type of amplifier that has two inputs, it amplifies the difference between two input voltages and rejects any common voltage value to the two inputs. The output voltage of a simple differential amplifier can be expressed as:

$V_{out} = A\,(V_1 - V_2)$ Where A is the gain of the amplifier.

Differential amplifier is one of the most widely used building blocks in analog integrated-circuit design. It is the input stage of every Operational Amplifier. Two simple differential amplifiers are provided below:





Procedure:

1. Add 2 **pmos4** transistors and 4 **nmos4** transistors.
2. Click on SPICE directive , write: **.include cmos.txt** as in Fig. 1:

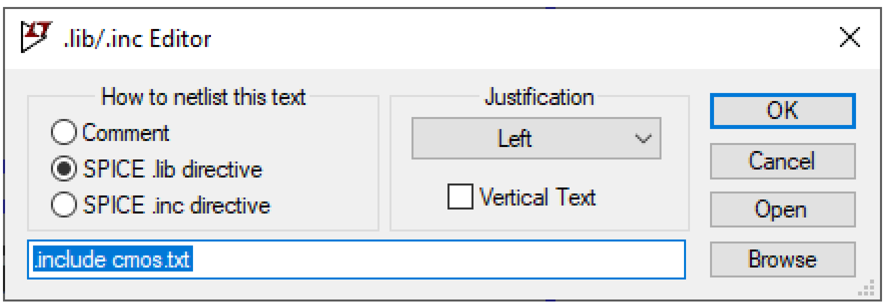


Fig. 1. Including cmos.txt

3. Connect the circuit in Fig. 2:

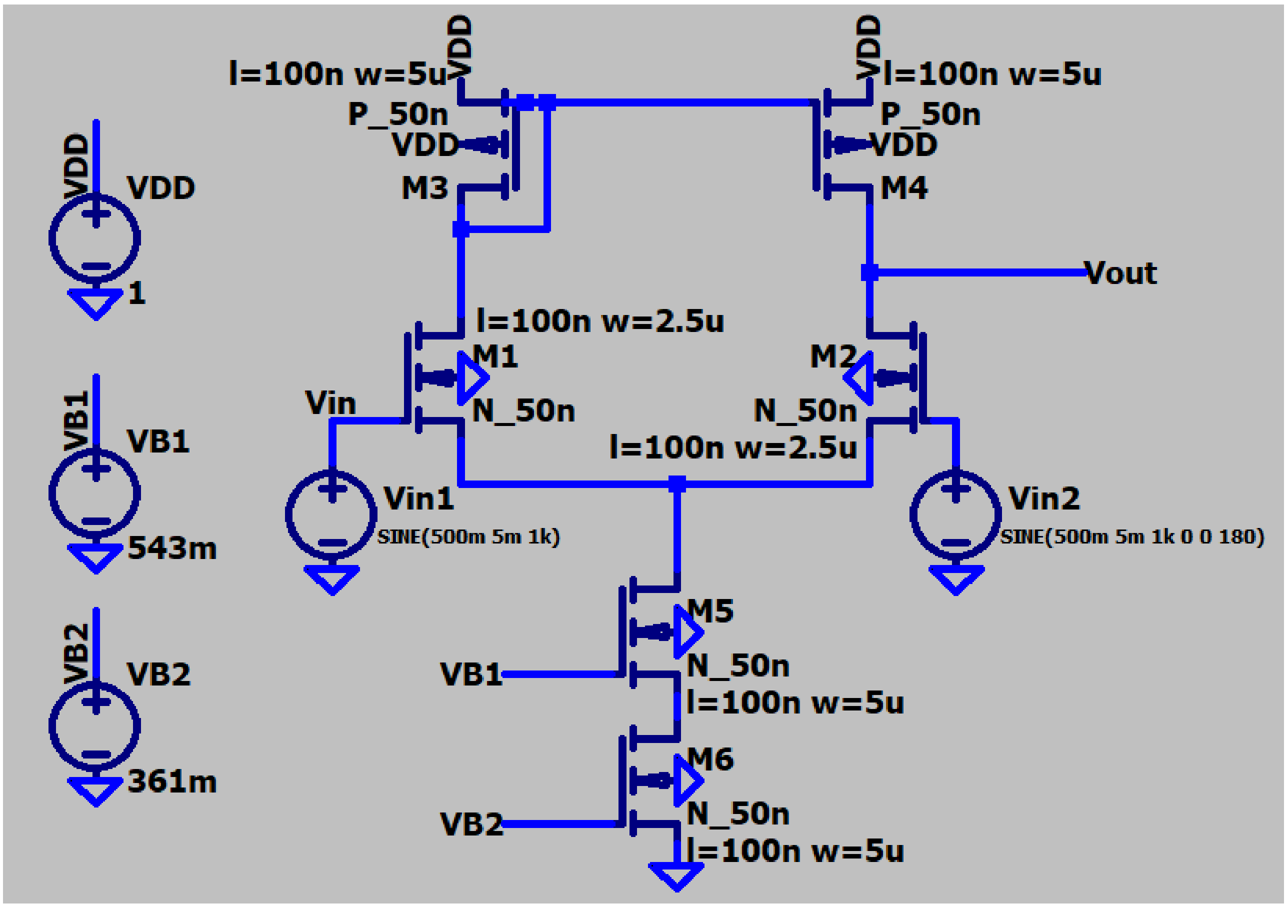


Fig. 2. CMOS Differential Amplifier

Set the transistor parameters as in Fig.3:

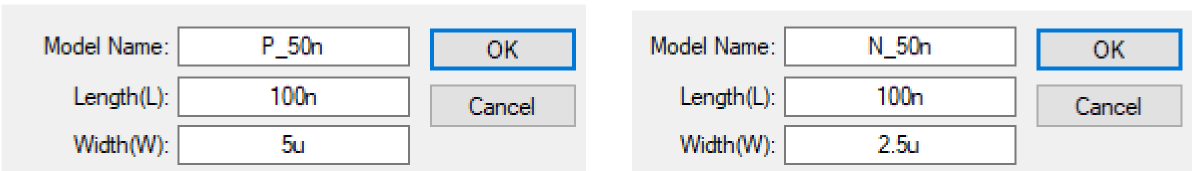


Fig. 3. PMOS and NMOS specifications

The lower **two transistors (M5 & M6)** have a width of 5u instead of 2.5u:

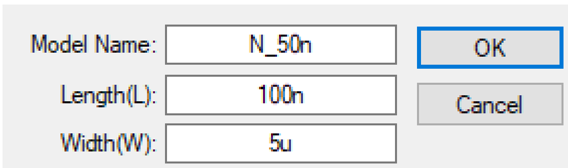


Fig. 4. M5 & M6 transistors' specifications

Right click on the voltage sources Vin1 & Vin2, click (Advanced), set their parameters as in Fig. 5 & Fig. 6:

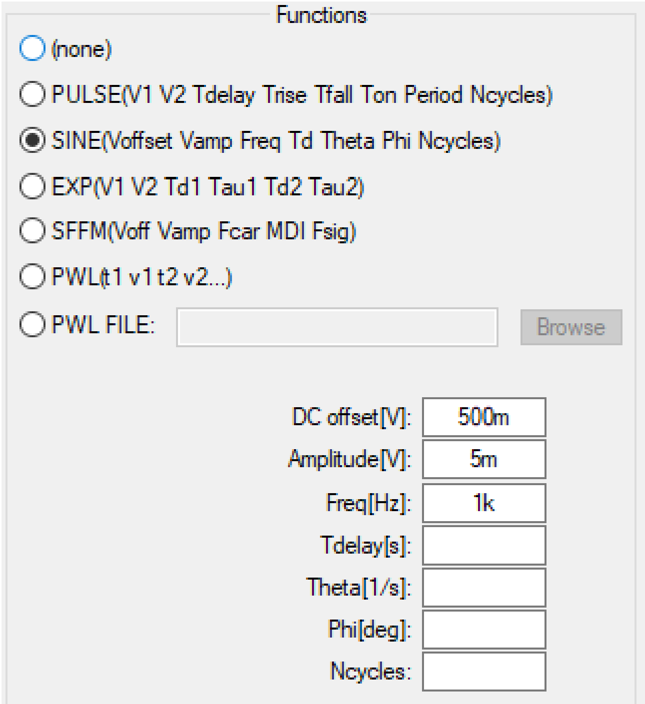


Fig. 5. Input voltage (Vin1)

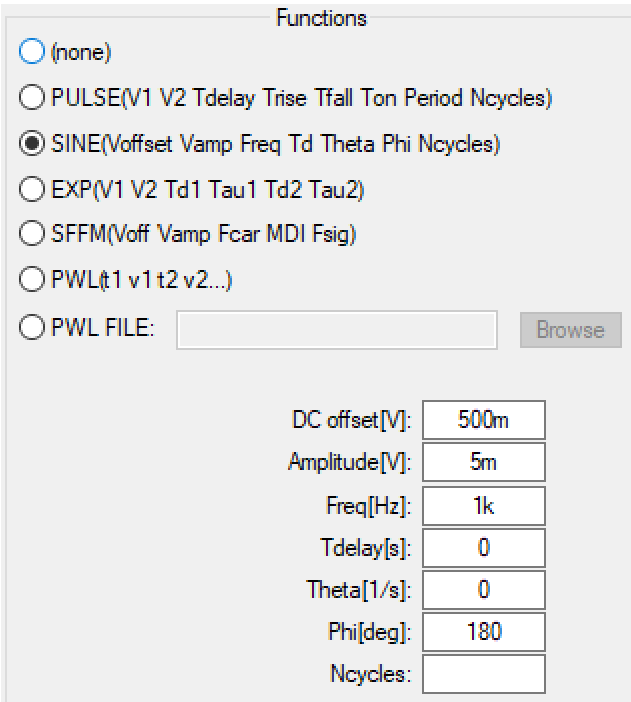


Fig. 6. Input voltage (Vin2)

Note that the signals are out of phase by 180 degrees.

Run transient analysis, set Stop time (10m), plot **Vin1**, **Vin2** & **Vout** as illustrated in Fig. 7:

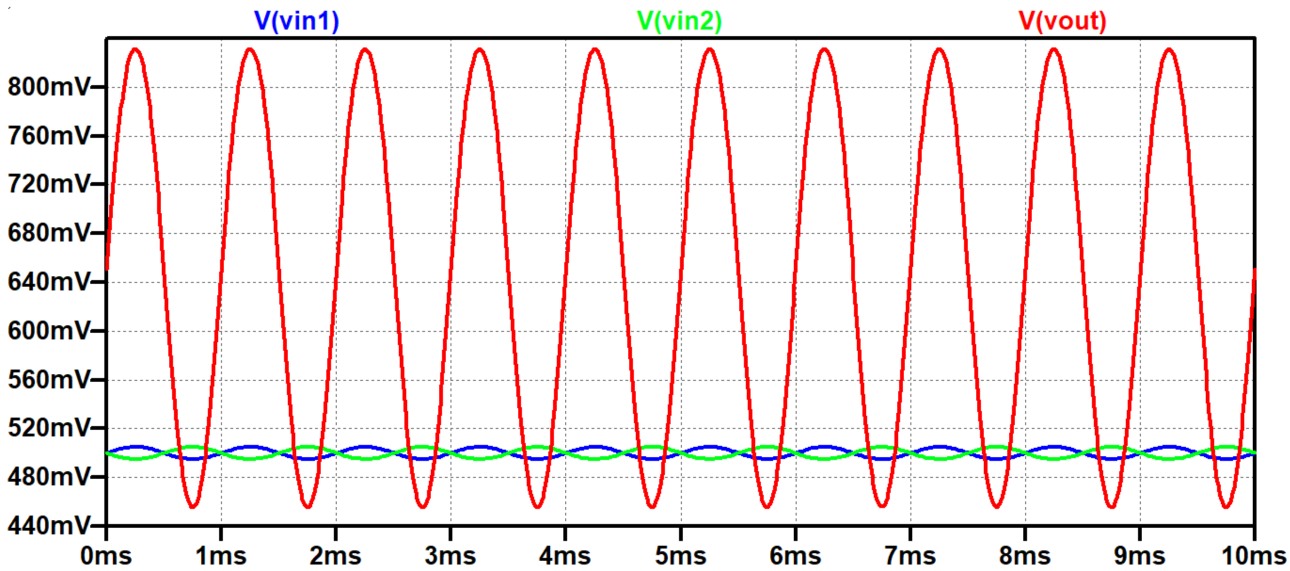


Fig. 7. Output and input voltages of the differential amplifier

Report Questions:

1. Where are differential amplifiers used?
2. Modify the input voltages, set their phases to zero, what does the output voltage look like? Include its plot in the report.
3. Run DC sweep analysis (refer to Fig. 8) for the output voltage (Vout), include the plot in the report.

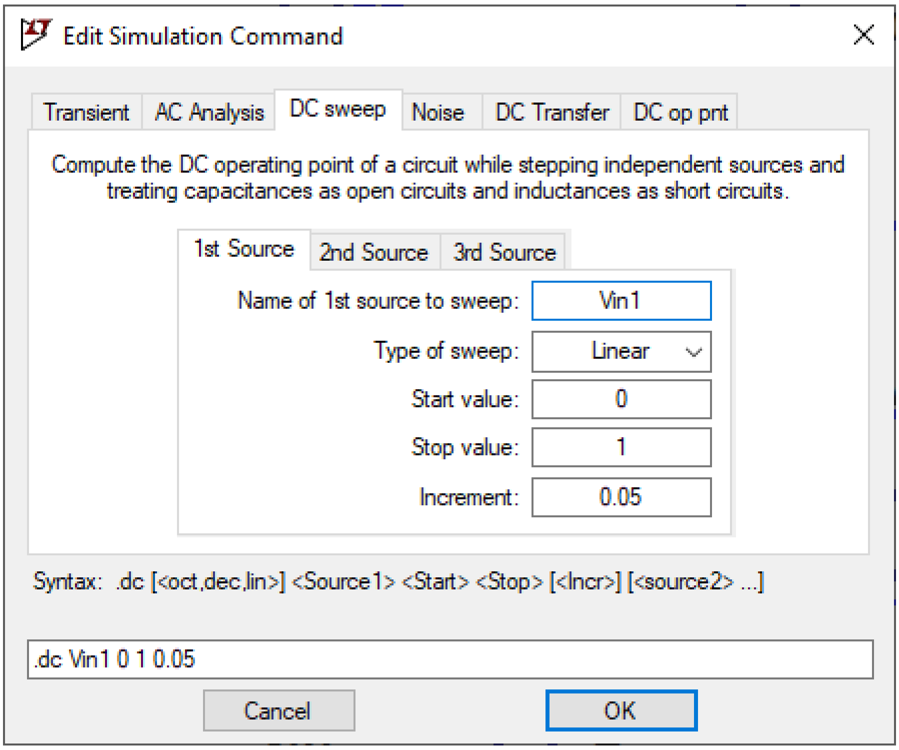


Fig. 8. DC sweep parameters

Experiment (9)
Power Amplifiers

Objectives:

To study the major aspects of class C power amplifier and learn how to plot FFT of signals in LTspice.

Introduction:

A power amplifier is an electronic amplifier designed to increase the magnitude of power of a given input signal. The power of the input signal is increased to a level high enough to drive loads of output devices like speakers, headphones, RF transmitters etc. Unlike voltage and current amplifiers, a power amplifier is designed to drive loads directly and is used as a final block in an amplifier chain.

Procedure:

1. Insert one BJT (npn) transistor, as in Fig. 1.

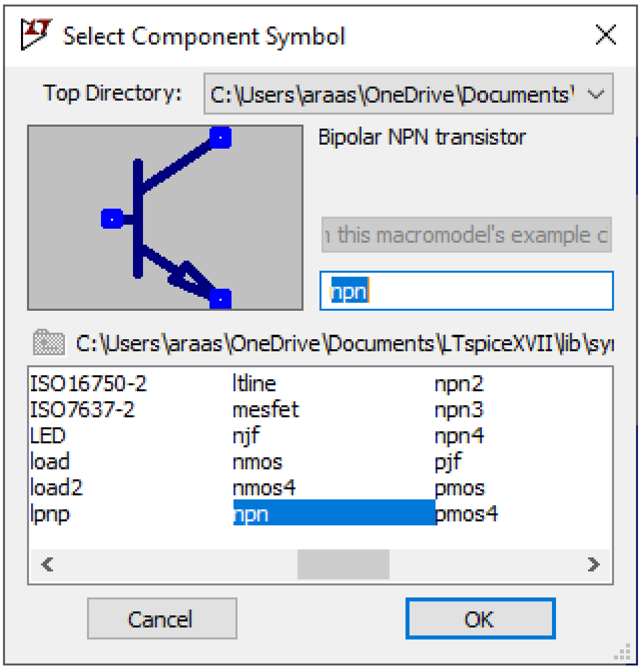


Fig. 1. npn transistor

2. Right click on the transistor, choose (2N3055) model as shown in Fig. 2:

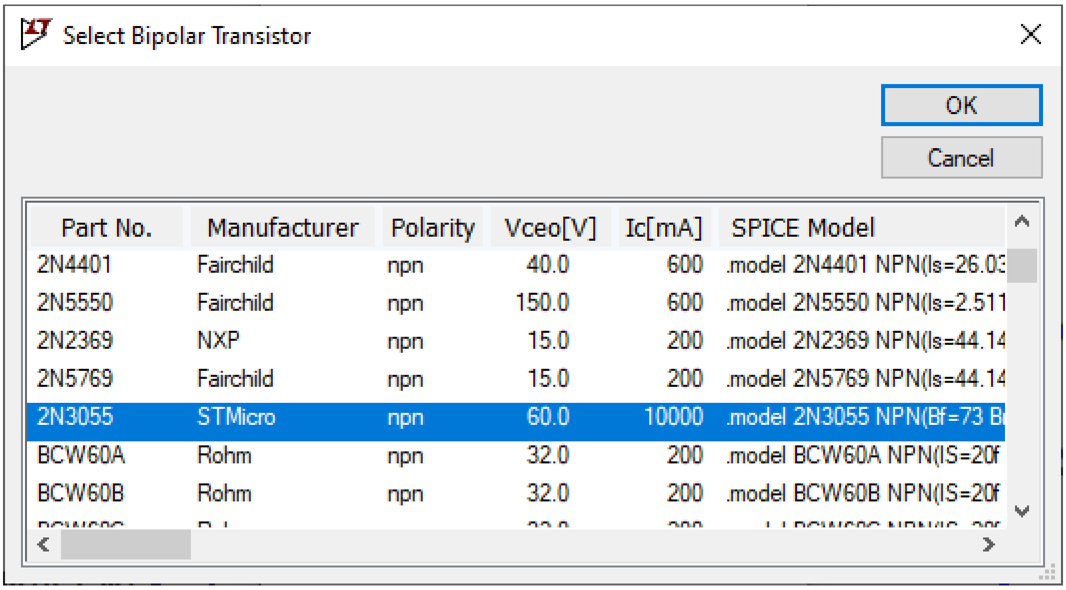


Fig. 2. Picking the power transistor model

3. Connect the class C power amplifier in Fig. 3:

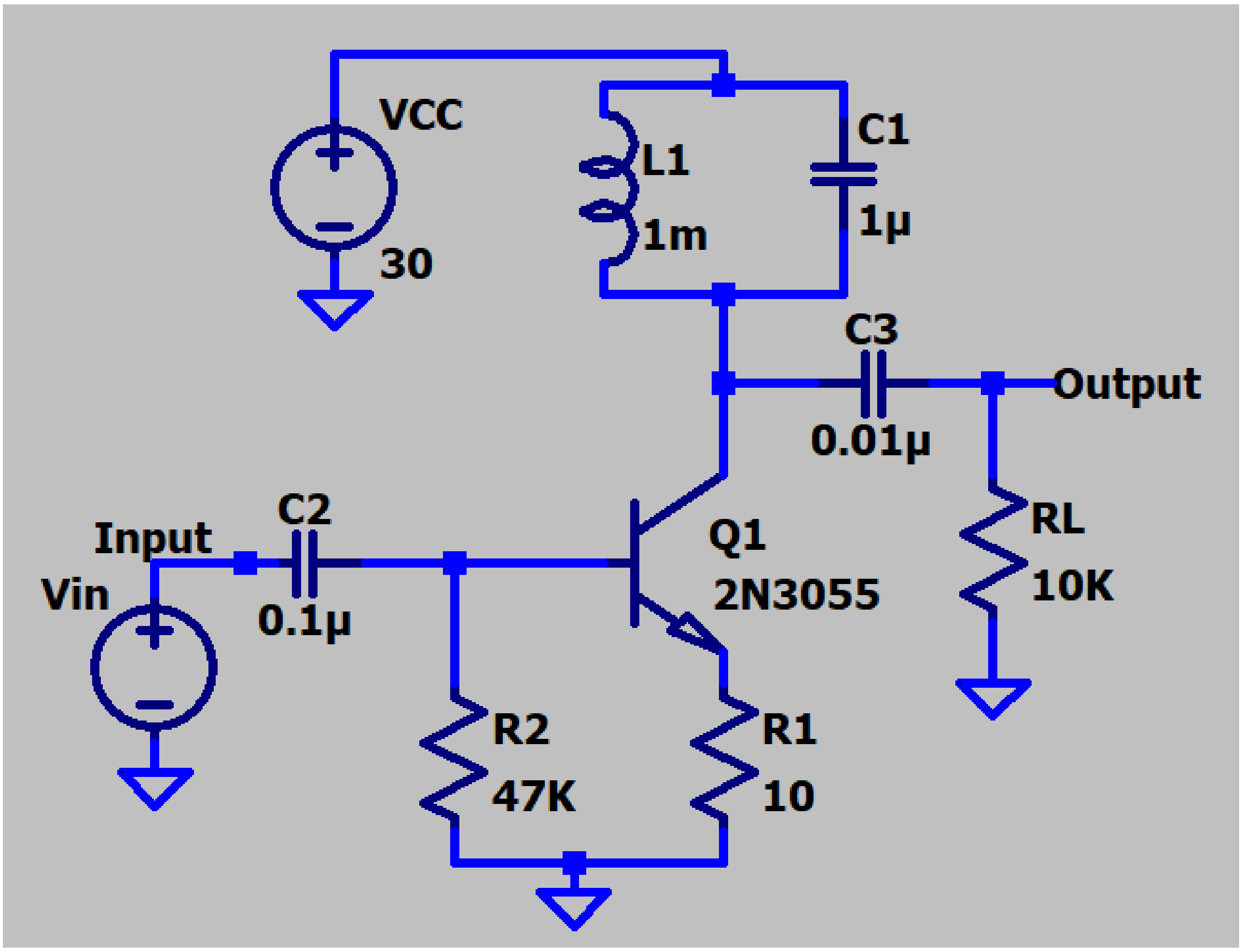


Fig. 3. Class C power amplifier circuit

The resonant frequency of the given class C amplifier is determined by the inductor (L) and capacitor's (C) values, the parallel LC circuit is commonly known as the tank circuit (see Fig. 4).

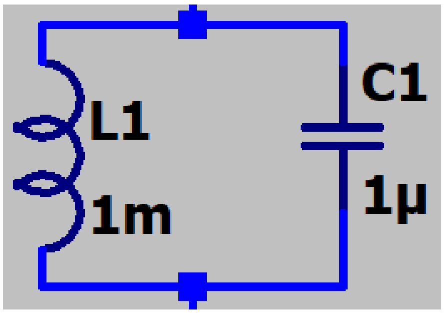


Fig. 4. The tank circuit

Class C amplifiers only amplify the signals that have a frequency equal to the resonant frequency, they attenuate other signals having different frequencies. In our case, the resonant frequency is equal to 5.03 KHz which is calculated by the following formula:

$$F_r = \frac{1}{2\pi\sqrt{LC}} = \frac{1}{2\pi\sqrt{0.001 * 0.000001}} = 5.03 \text{ KHz}$$

Set the VCC voltage value as 30, and Vin voltage in accordance with the specifications mentioned in Fig. 5, label the input and output pins.

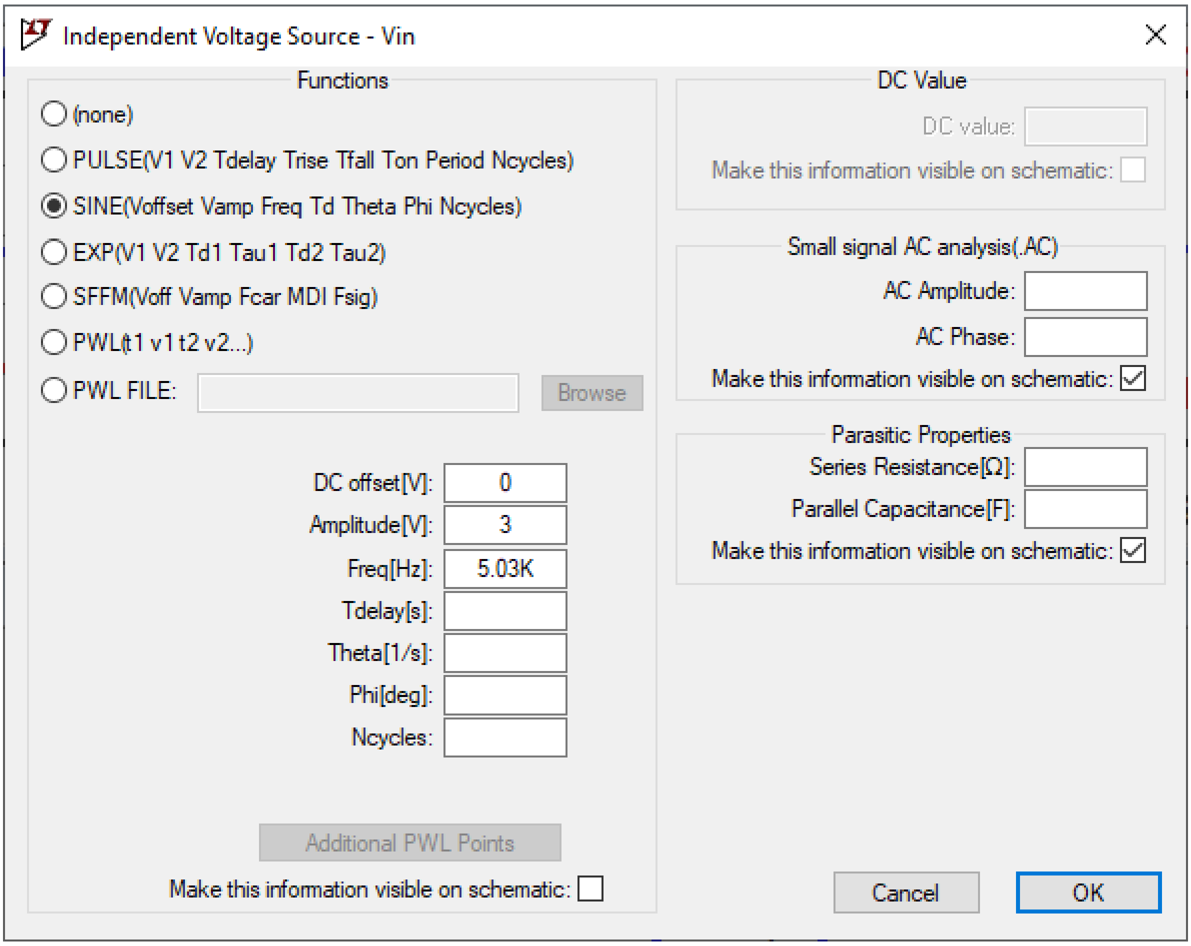


Fig. 5. Vin (sine wave) parameters

Run transient analysis, set stop time as 5m, plot the input and output signals as illustrated in Fig. 6:

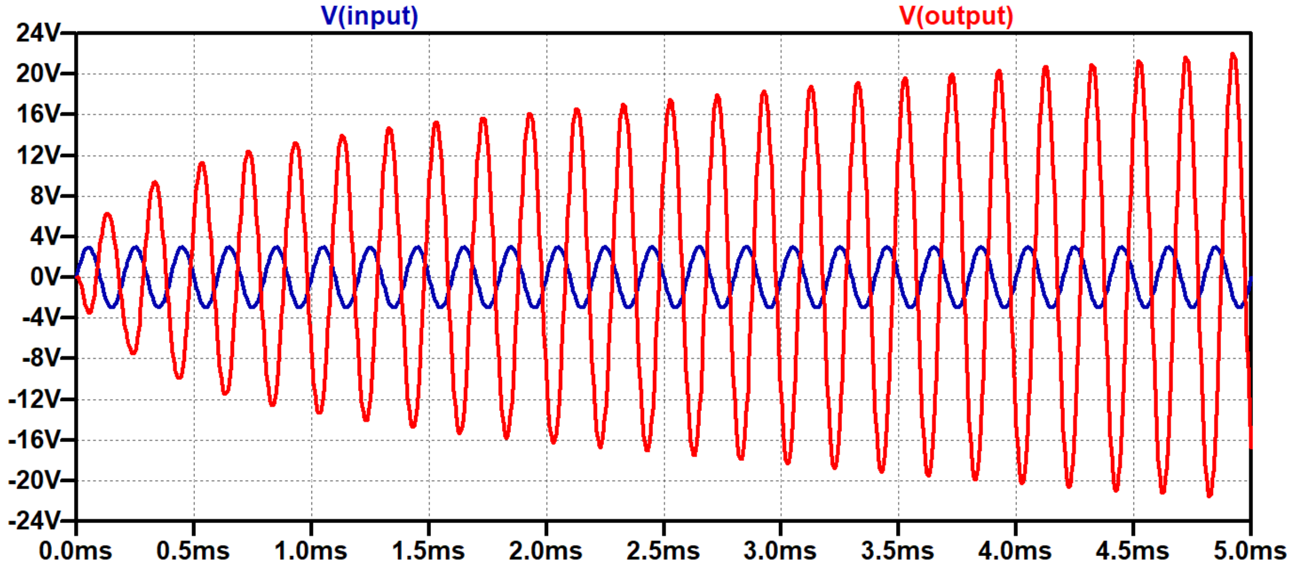


Fig. 6. Input & amplified output signals

4. Right click on the Vin, set the frequency to **0.5 KHz**, run the simulation again to see the output, you will observe that the output is less than the input as seen in Fig. 7, because the amplifier attenuates the signal as the frequency of the input signal is not equal to resonant frequency (5.03 KHz).

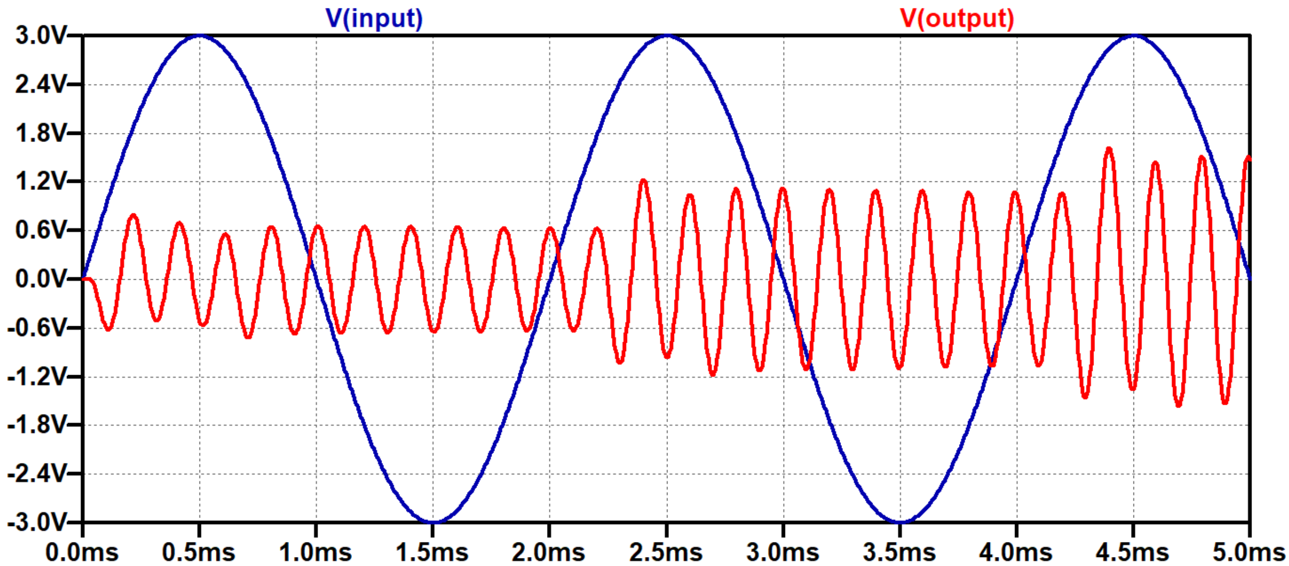


Fig. 7. Input & attenuated output signals

For the part two of this experiment, add two other sine wave sources, connect them in series, as shown in Fig. 8:

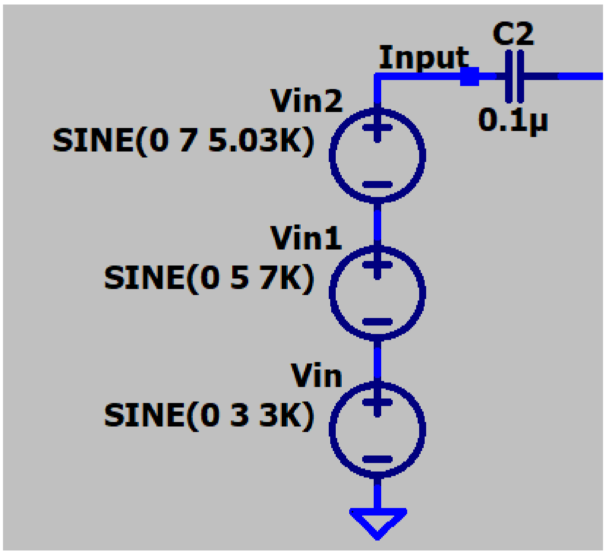


Fig. 8. Voltage sources in series

Set their values in accordance with the table below,

Source name	Amplitude (V)	Frequency
Vin	3	3K
Vin1	5	7K
Vin2	7	5.03K

Run transient analysis, set stop time as 5m, plot the input and output signals as in Fig. 9:

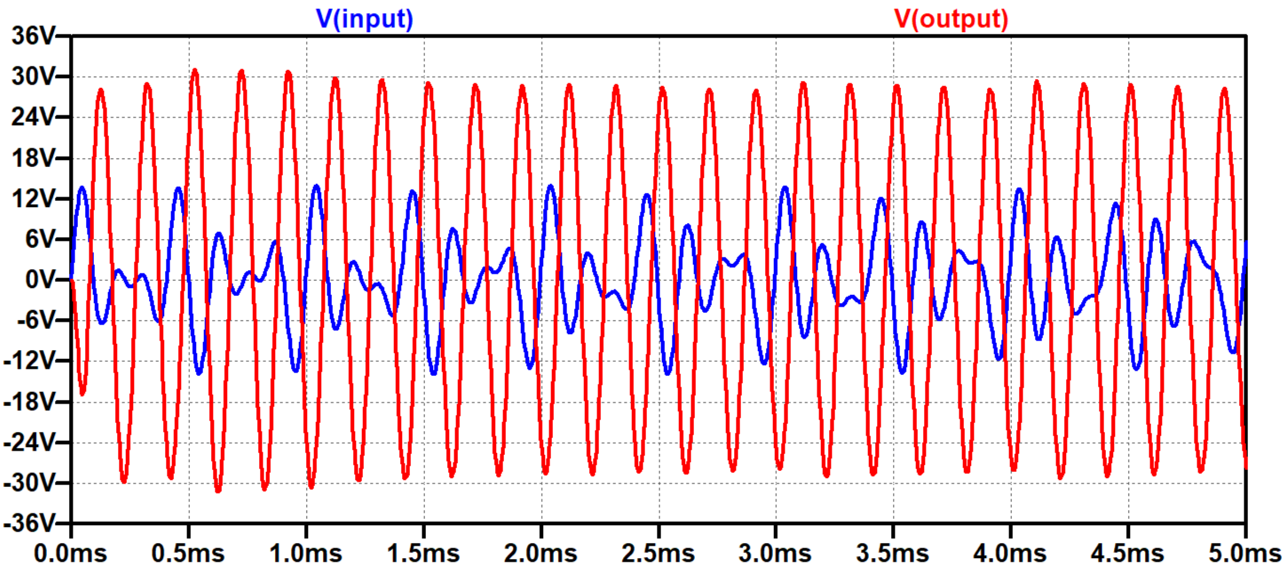


Fig. 9. Input signal (sum of 3 sinusoidal signal) & the amplified output signal

The output signal in Fig. 9 has a frequency of 5.03 KHz, the other two signals with frequencies of 3 KHz and 7 KHz are omitted, this can be seen more clearly by taking the Fast Fourier Transform (FFT) of the input and output signals, this can be done easily by **right clicking** on the plot then selecting **view > FFT > Click OK** (see Fig. 10).

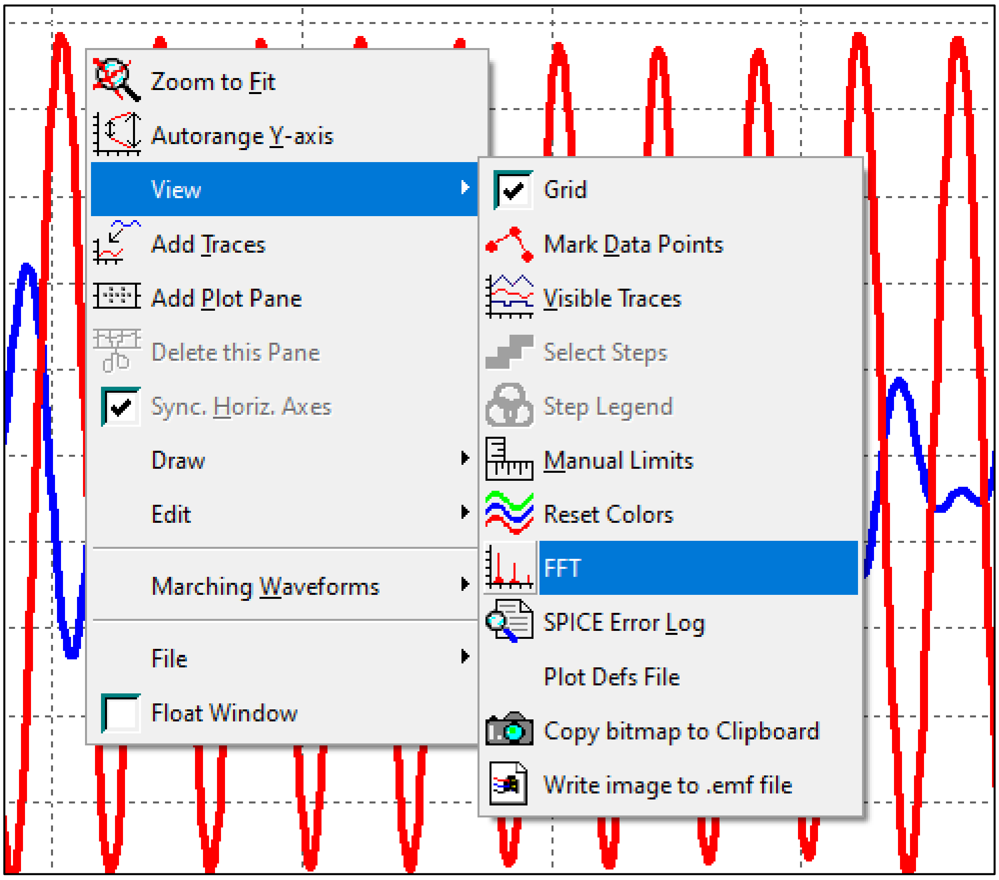


Fig. 10. Illustrating how to take FFT of a signal in LTspice

Fig. 11 shows the FFT of the input and the output signal, the FFT of the input signal has 3 main beams (at 3 KHz, 5 KHz and 7 KHz), while the FFT of the output signal has only one main beam at 5 KHz, this indicates that the class C amplifier neglects other signals with frequencies different than the resonant frequency.

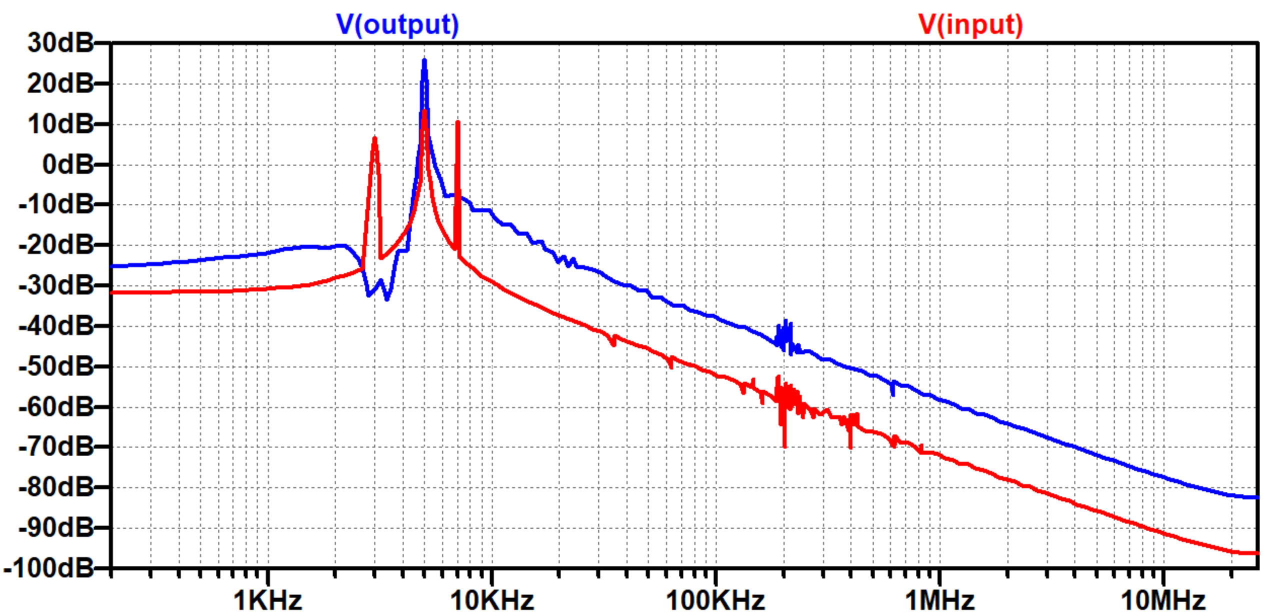


Fig. 11. FFT of the input and the output signal

Report Questions:

1. What does 2N3055 transistor look like? Add its picture and explain why we used this transistor for this experiment?

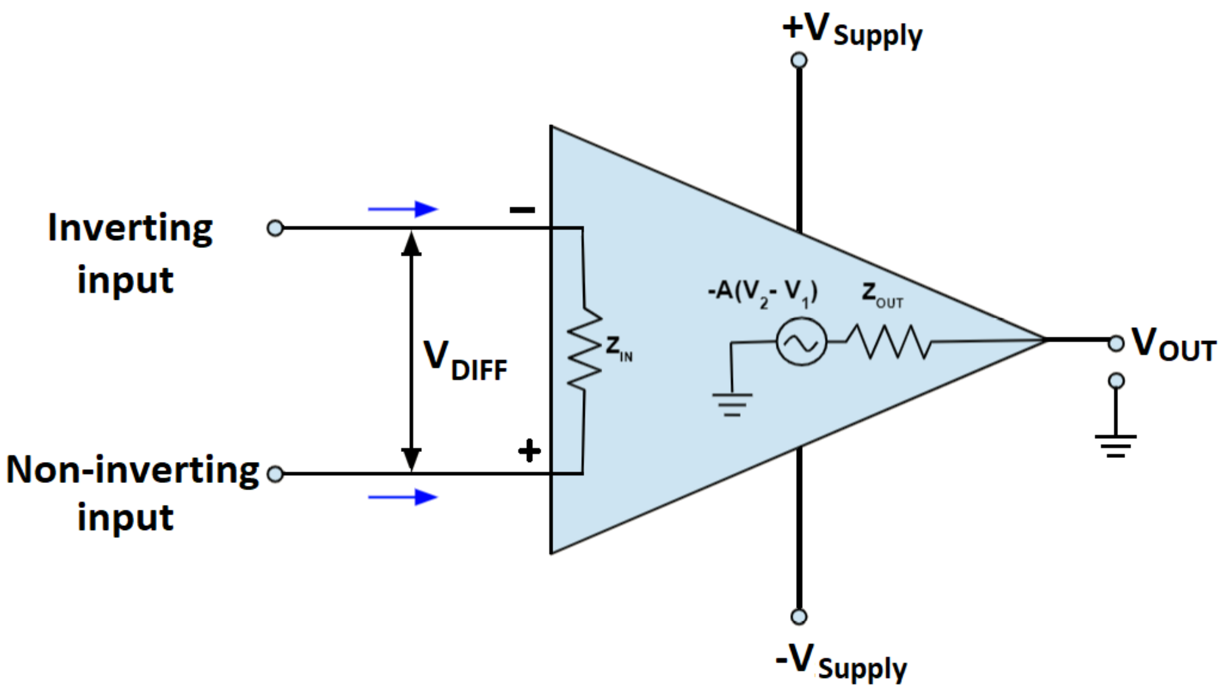
2. What is a heat sink? What is it used for?

3. A tuned class C amplifier has a resonant frequency of 1 MHz, the capacitor is 10 uF, an inductor of which value should be used?

Experiment (10)
Operational Amplifiers

Objectives:
To study the fundamental structure of CMOS operational amplifiers.

Introduction:
An operational amplifier (op-amp) is a type of amplifier that amplifies the voltage difference between the two input pins, op-amps usually have three terminals: two high-impedance inputs and a low-impedance output (although some op-amps have an additional differential output). The inverting input is denoted with a minus (-) sign, and the non-inverting input denoted with a positive (+) sign. Operational amplifiers have an extremely high gain, they are used in various analog circuits including mixers, filters, sensors, buffers, etc.



Most op amps are used for voltage amplification, however there are four categories of op-amps:

1. Voltage amplifiers (voltage input, voltage output)
2. Current amplifiers (current input, current output)
3. Transconductance amplifiers convert a voltage input to a current output.
4. Transimpedance amplifiers convert a current input to a voltage output.

Procedure:
1. Add 3 **pmos4** transistors and 6 **nmos4** transistors.
2. Click on SPICE directive .op, write: **.include cmos.txt**

3. Connect the op-amp circuit in Fig. 1:

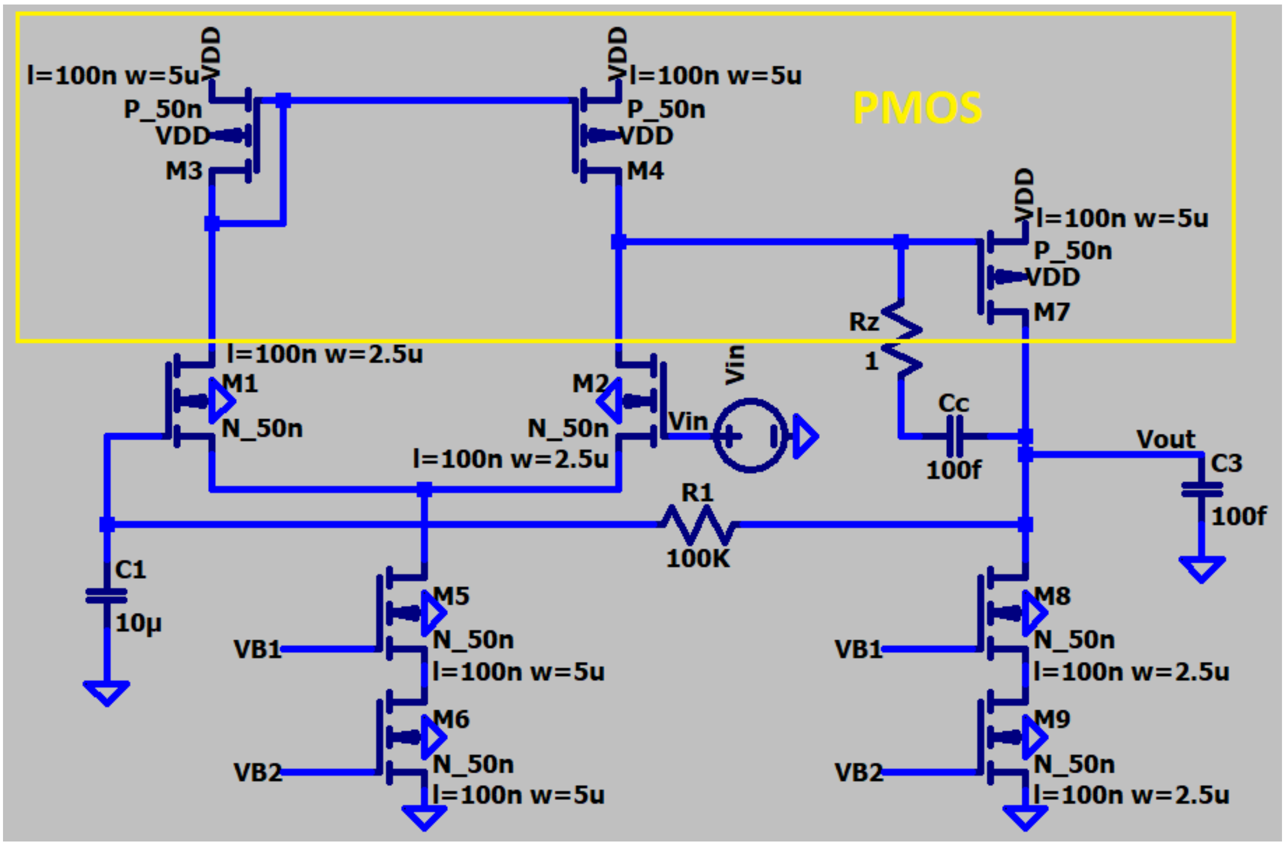


Fig. 1. CMOS operational amplifier

4. Carefully modify the transistors' widths and lengths according to the circuit diagram in Fig. 1, do not forget to connect Grounds and VDDs.

5. The input voltage source (Vin) has such parameters:

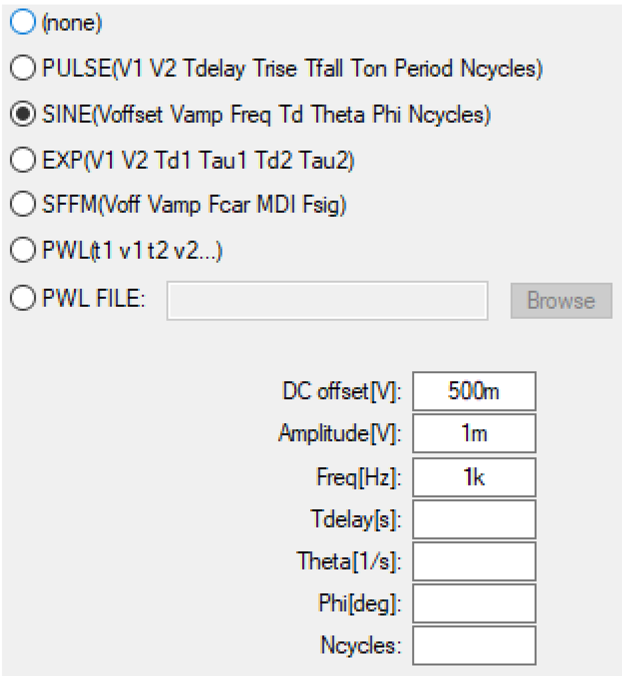


Fig. 2. Vin parameters

6. Add three DC voltage sources, label them as VDD, Bias1 and Bias2 as shown in Fig. 3. Their values are as following:

- VB2 = 361 m
- VB1 = 543 m
- VDD = 1

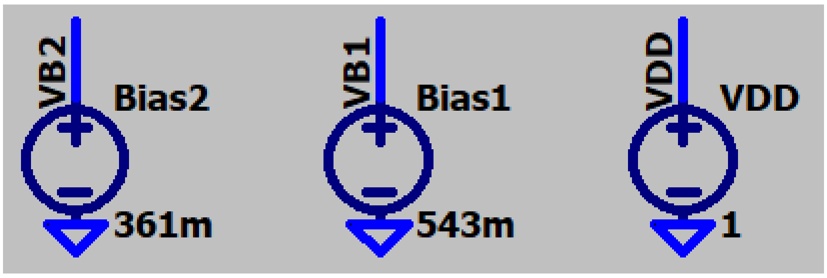


Fig. 3. Bias voltages and VDD

7. Run transient analysis (Stop time = 10m).

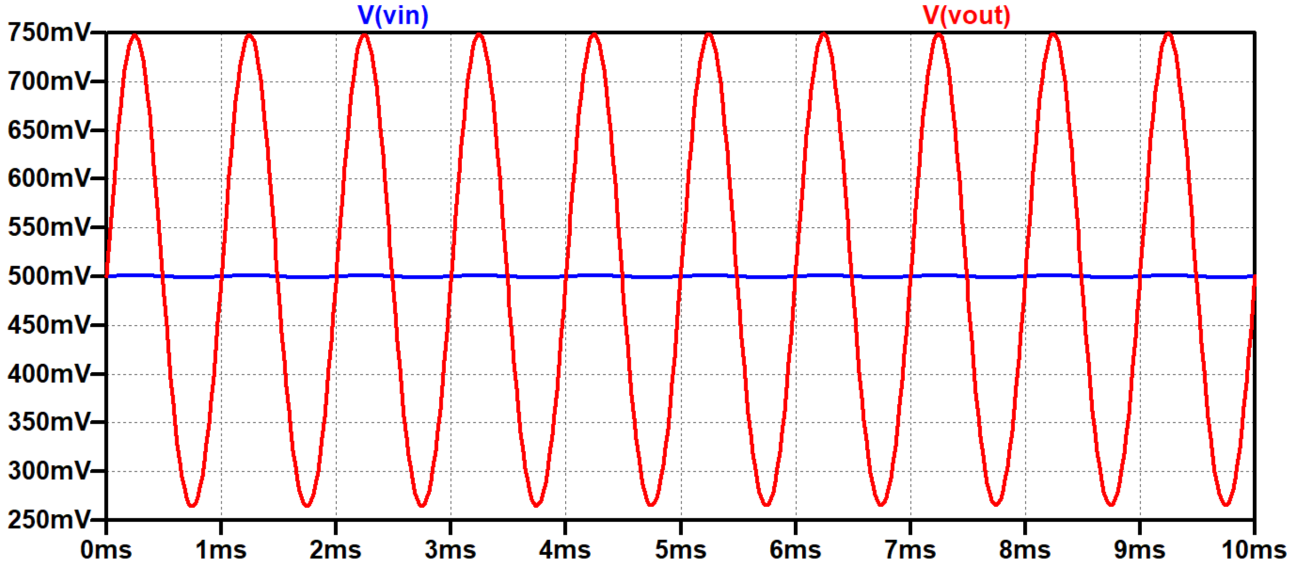


Fig. 4. Input voltage signal vs. the amplified output signal

8. We can also plot the frequency response of the op-amp, close the waveform, then change the parameters of the input voltage source (Vin) as in Fig. 5.

9. Press OK.

10. Run AC analysis for the output voltage (Vout) in accordance with the specifications mentioned in Fig. 6.

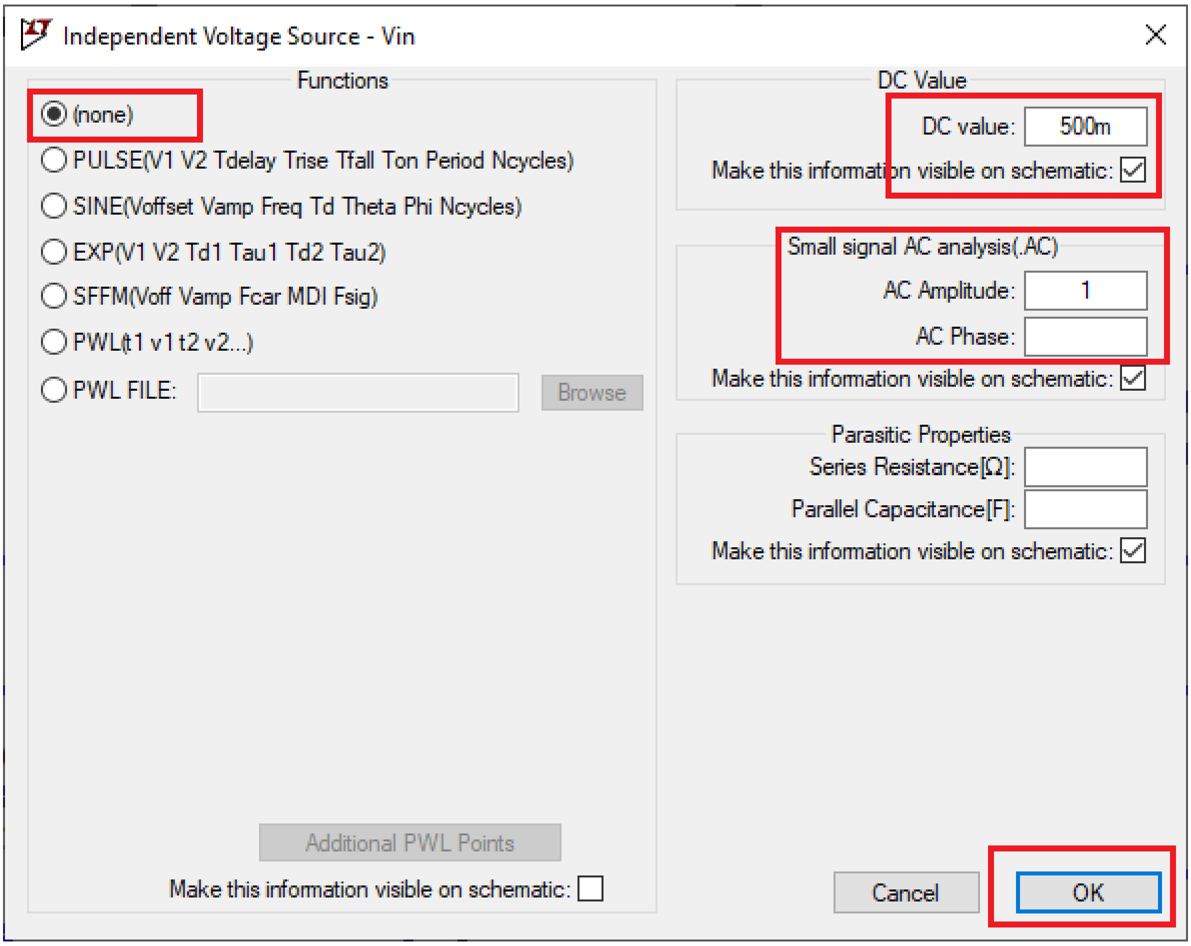


Fig. 5. Adjusting input voltage source for the AC analysis

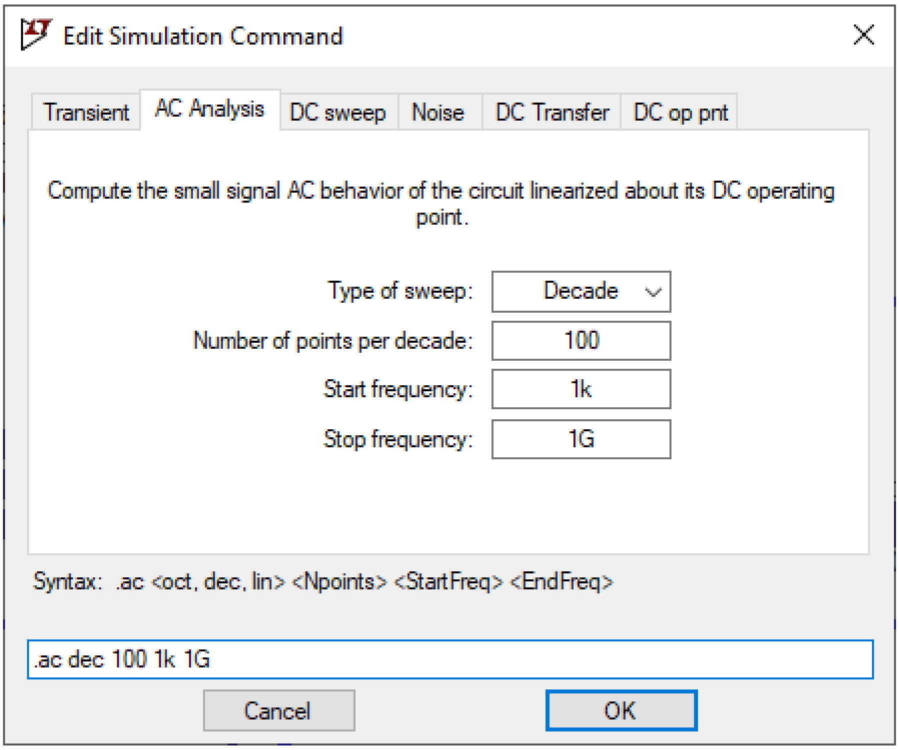


Fig. 6. AC analysis command

The results are shown in Fig. 7:

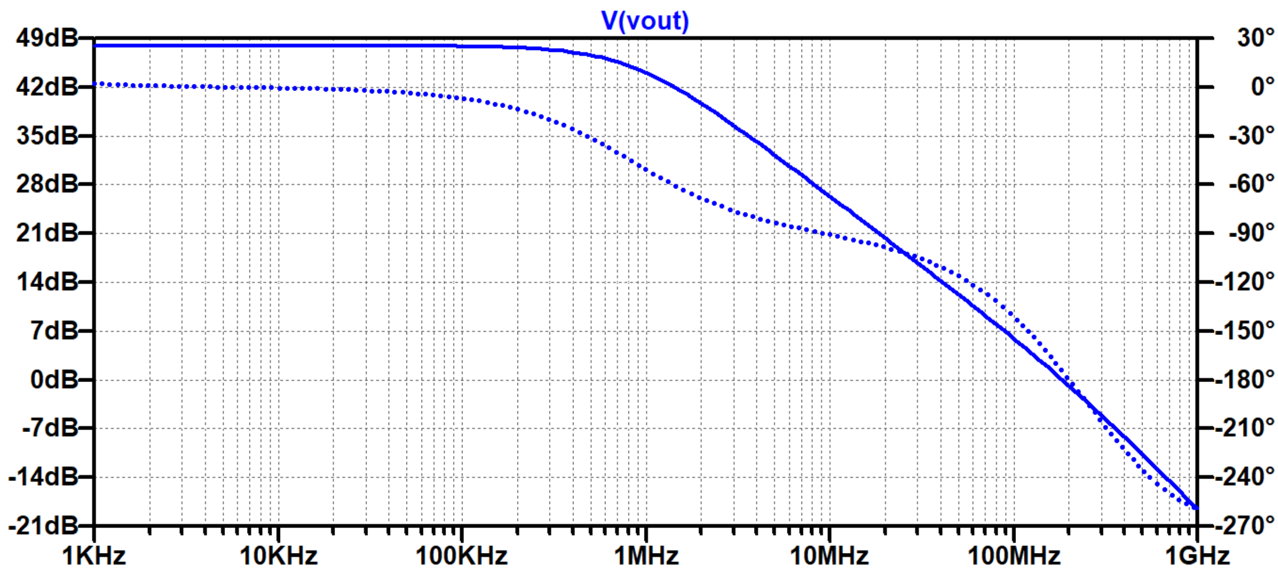
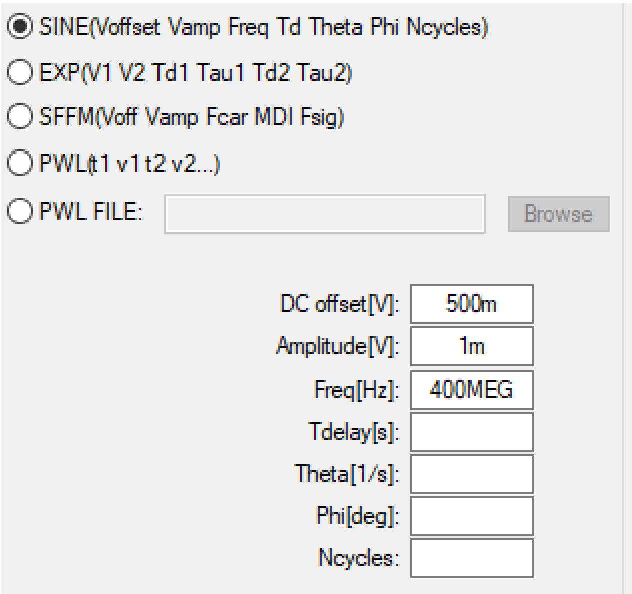


Fig. 7. Frequency response of the op-amp

Report Requirements:

1. Calculate the voltage gain of the op-amp at 1 kHz frequency.
2. At which frequency does the op-amp has a unity gain?
3. Plot the input and output voltage signals when the input signal has the following parameters, what do you notice?



4. Repeat the previous step for these parameters, what will the output signal look like and why?