EXPERIMENT 1
PSPICE Simulations of Common Source Stage Characteristics

A. Background

Download, print and read the tutorial on the usage of ORCAD/PSPICE available at

http://homes.ieu.edu.tr/~maskar/EEE331/General/ORCAD-PSPICE-Tutorial.pdf

B. Experimental Work

Part 1: Common Source Amplifier- Dc Analysis

1. Construct the given circuit below (Fig. 1.1) in ORCAD by following the steps.

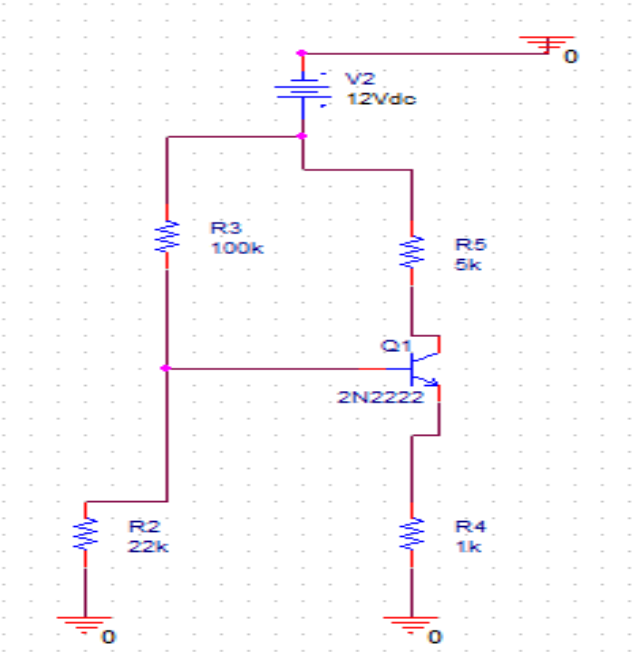

Fig. 1.1

2. Select **File >> New >> Project**. Name your project (Fig. 1.2) as "**Common_Source_Amplifier_NAME_SURNAME**". Asthelocationfield, select the directory as "**D:\MYWORKS\EEE331_ORCAD**". Be sure that you selected "**Analog or Mixed A/D**".

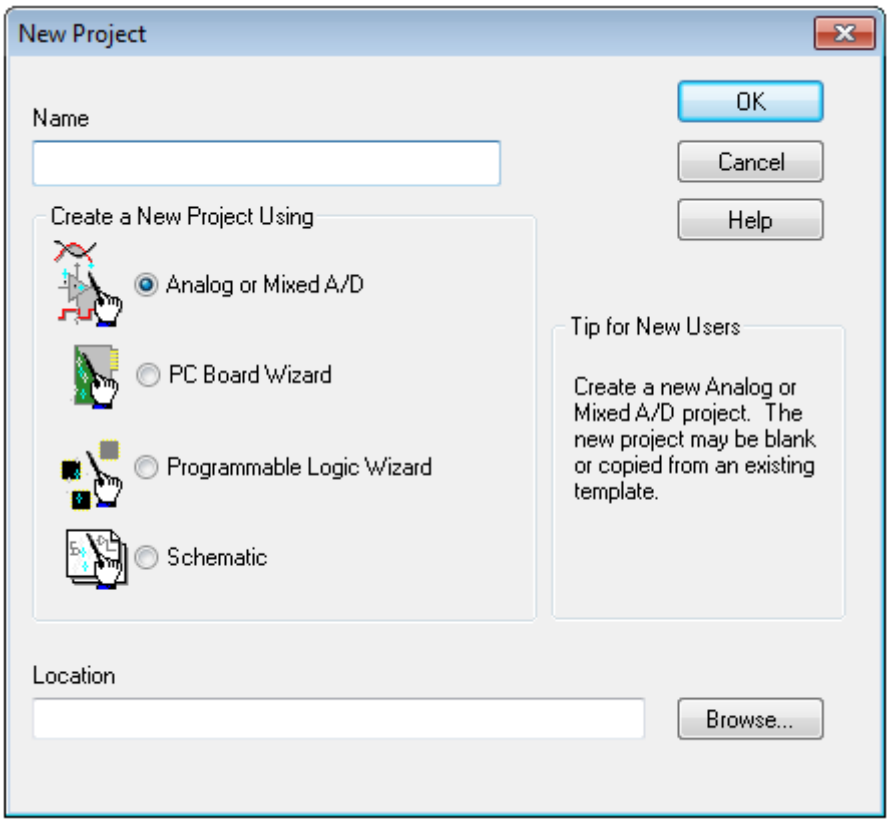

Fig. 1.2

3. Now, select "**Create a blank project**" at the appeared diagram box (Fig. 1.43).

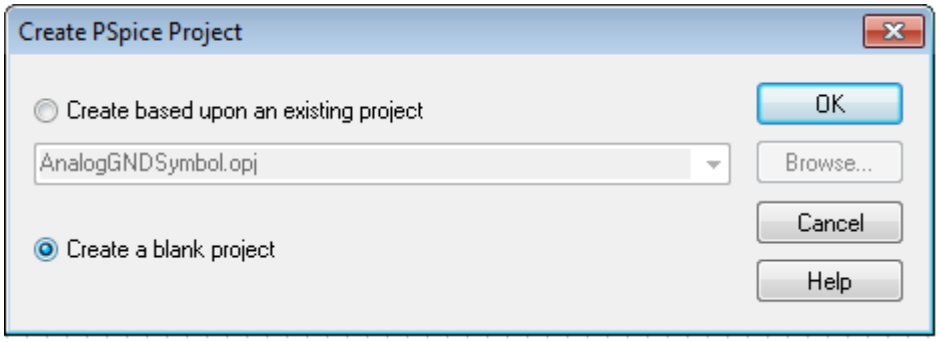

Fig. 1.3

4. Add a sinusoidal voltage source (VSIN) and a resistor(R) to your circuit as described in the ***ORCAD/PSPICE Tutoria***l. To work your circuit properly, don't forget to add **Ground** to your circuit.
5. Set the parameters as shown in Fig. 1.4. (Your laboratory work parameters will be assigned to you during the Laboratory Work

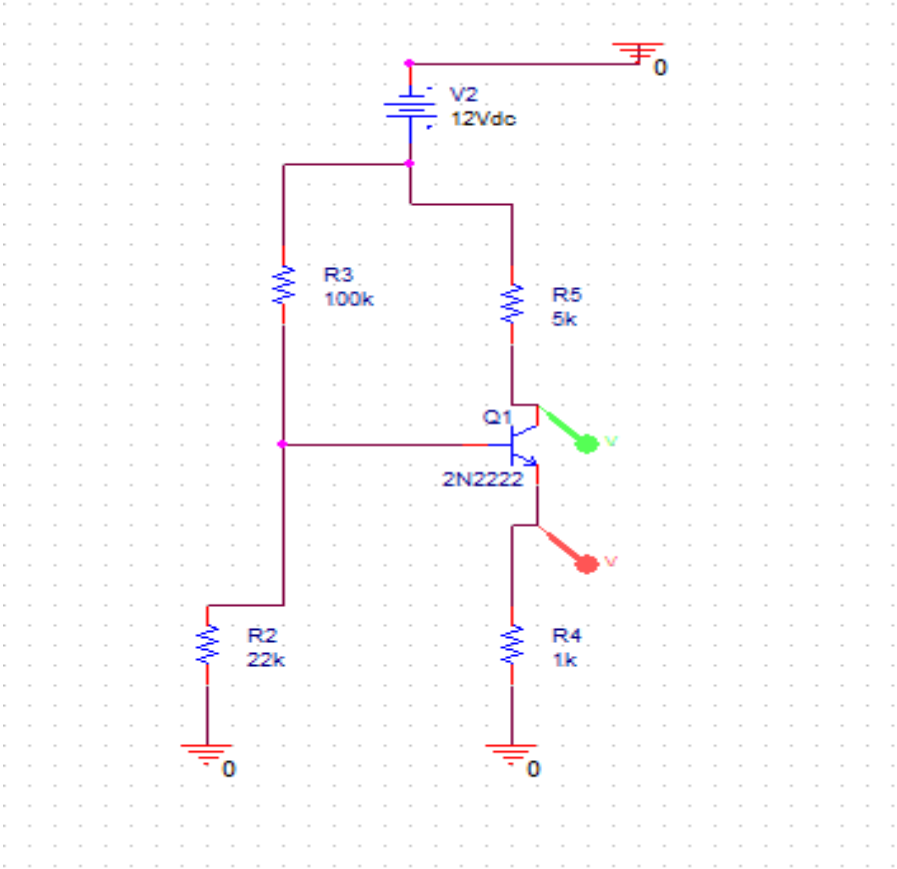

Fig. 1.4

6. After the construction of circuit, create a new profile using Pspice >> New Simulation Profile from toolbar (Fig. 1.5). Write "**Common_Source_Amplifier_NAME_SURNAME**" for New Simulation Name.

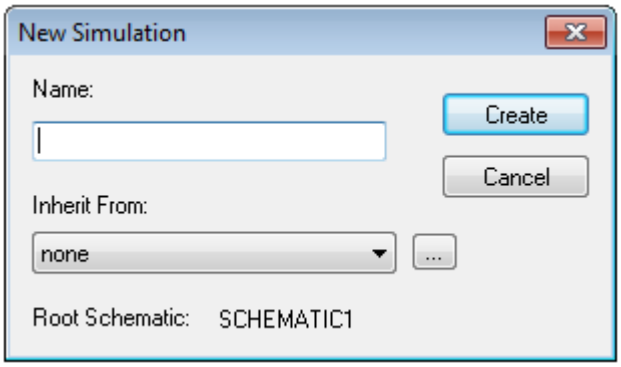

Fig. 1.5

7. Set the simulation settings as given in Fig. 1.6

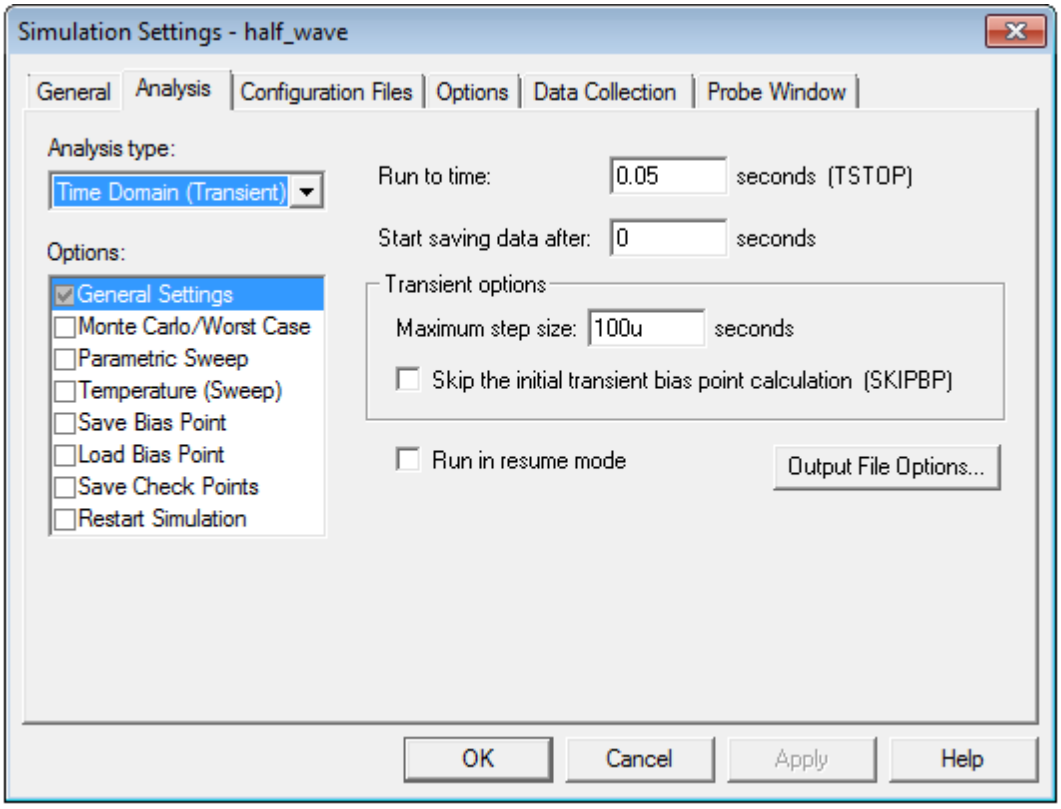

Fig. 1.6

8. Run your program by using toolbar as **Pspice >> Run**.

Prepare a lab report based on the following format.

Insert your circuit schematic and simulation results below (The capture method is given in ***ORCAD/PSPICE Tutorial.***

Report: Part 1 - Common Source Amplifier- Dc Analysis

i. Circuit Schematic

Insert Your Circuit Schematic Here

ii. Simulation Results

Insert Your Simulation Results Here

iii. Discussions

C.2. Part 2: Common Source Amplifier

1. Add a sinusoidal voltage source (VSIN) of 10 mV at 1 kHz , a 10uF capacitor and a 1 k resistor(R) to your circuit as shown in Fig. 1.7

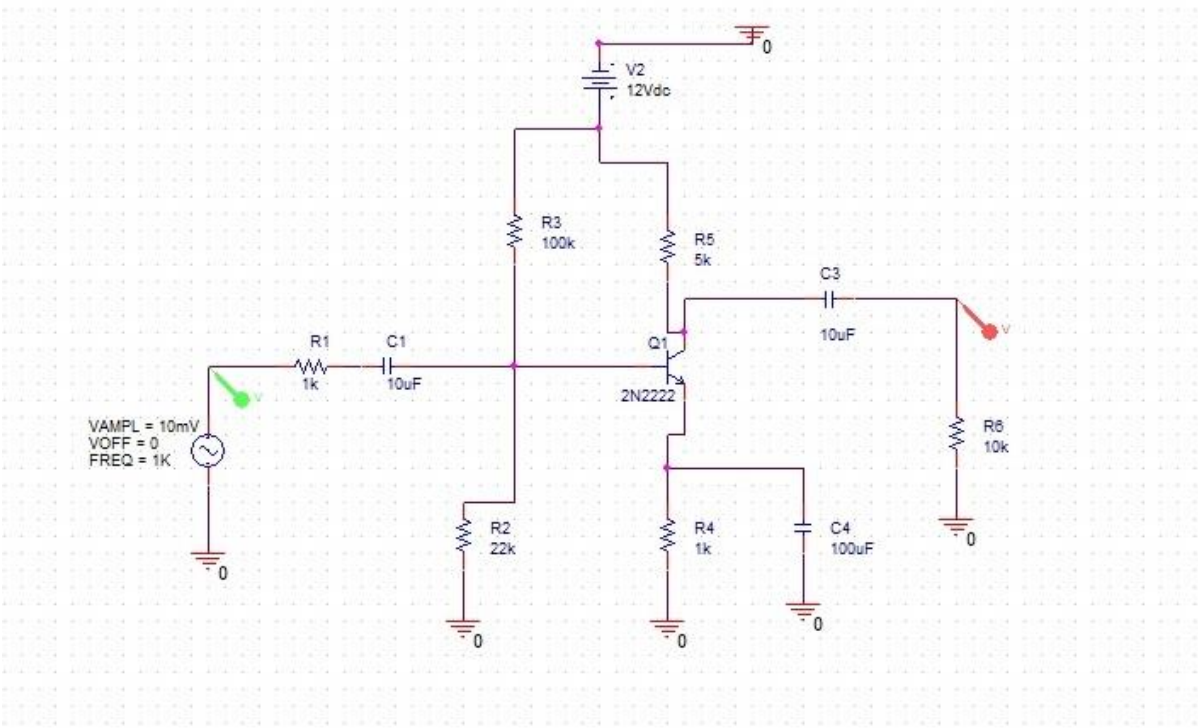

Fig. 1.7

2. ***** Can you observe the ripples at the output? How do they change with different capacitance values? Discuss this issue in the Discussions box.

Prepare a lab report based on the following format.

C.3. Report: Part 2 - Common Source Amplifier

Insert your circuit schematic and simulation results below.

i. Circuit Schematic

Insert Your Circuit Schematic Here

ii. Simulation Results

Insert Your Simulation Results Here

iii. Discussions

iv. Circuit Schematic (*Common Source Amplifier with sinusoidal voltage source (VSIN) of 20 mV at 1 kHz)*

Insert Your Circuit Schematic Here

v. Simulation Results (Common Source Amplifier with sinusoidal voltage source (VSIN) of 20 mV at 1kHz*)*

Insert Your Simulation Results Here

vi. Discussions (Common Source Amplifier with sinusoidal voltage source (VSIN) of 20 mV at 1kHz*)*