CENG4480 Embedded System Development and Applications
The Chinese University of Hong Kong
Laboratory 1: Op Amp – 1

Your Name:
Student ID:

2020 Fall

1   Introduction

This lab session introduces some very basic concepts of operational amplifier "op amp" that every engineering student should know about. The op amp used in this lab is LM324. In this lab we use TINA-TI Simulator to construct some typical op amp circuits and simulate their function and study their behavior.

1.1   Objectives

By completing this lab session, you should know:

- how to use op amp circuits to interface between sensors and MCU;

- how to design amplifier circuit using op amp and

- how to design integrator and differentiator circuit using an op amp

1.2   Procedures

Follow the procedures of each experiment. Write your result in a report sheet with your name and student ID. Submit the report sheet to the tutor after the lab.

2   Setup with TINA-TI Simulator

- Download TINA-TI_ENGLISH from Here or download from TI official web site: Here.

- Extract the .zip file *Tina90-TIen.9.3.200.277.zip*

- Install the simulator by double click the *Tina90-Tien* icon.

- Run the program by double click the icon

- The program will look like Figure 1

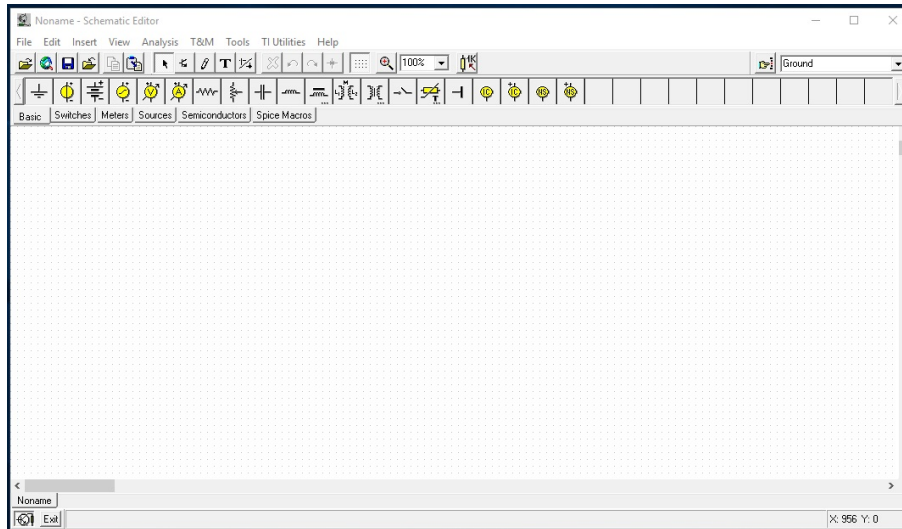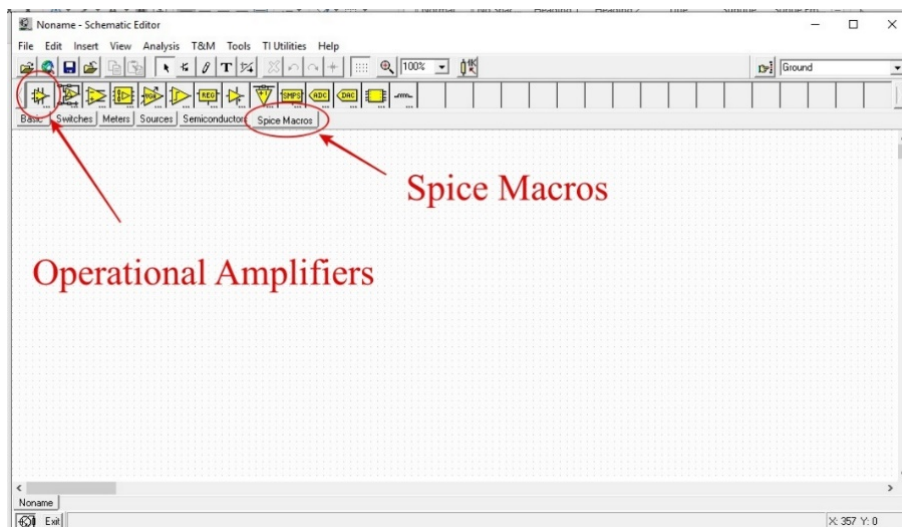To familiar with TINA-TI Simulator, go to TINA-TI Video Training Series at Here.

Figure 1: TINA-TI Simulator

3 Experiment 1. Inverting Amplifier

In this experiment, you will construct and observe the inverting amplifier. Record the input and output waveforms.
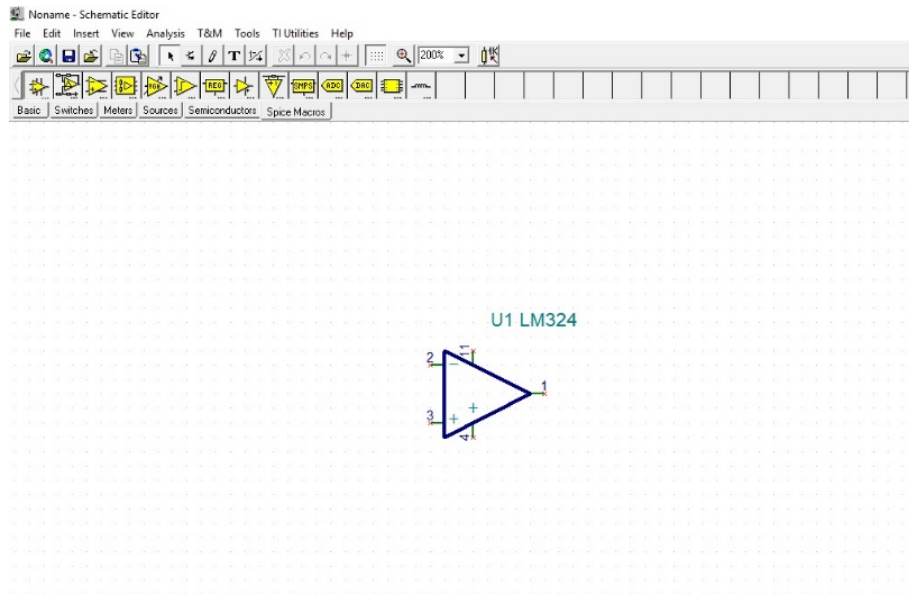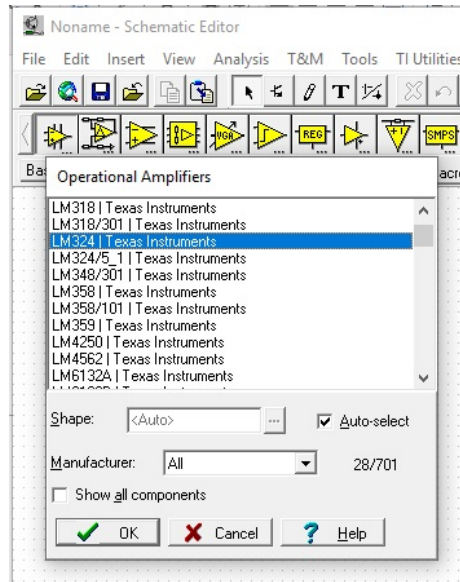
Procedures

1) Run the TINA-TI as shown in Figure 1.
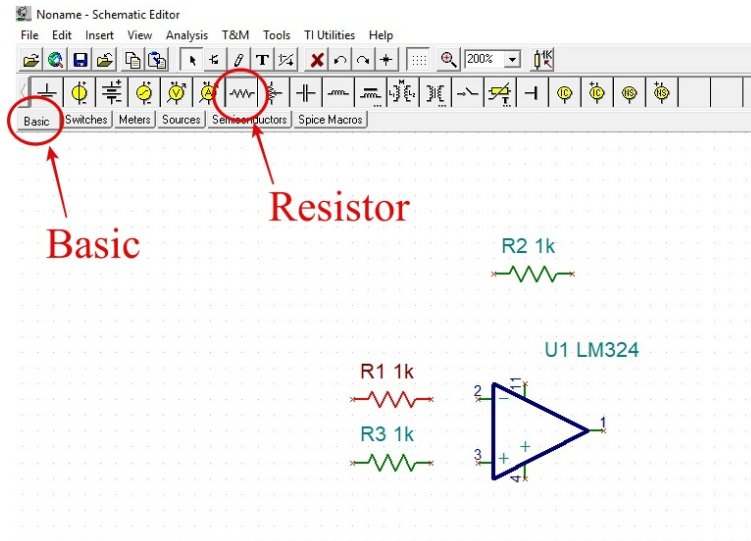

2) Insert the **op amp LM324**

- Press the **Spice Macros** tab
- Select **Operational Amplifiers** as shown
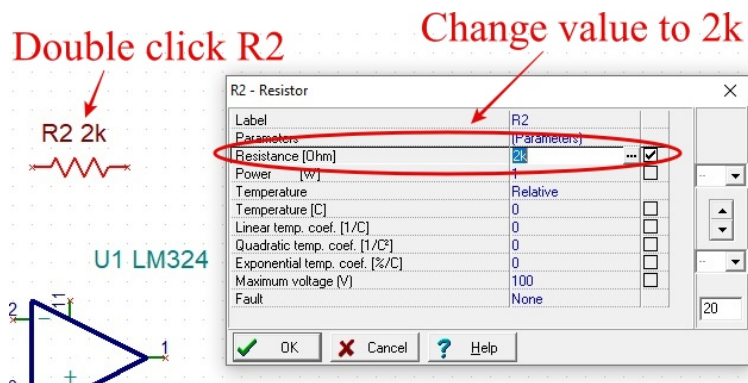- Select **LM324—Texas Instrument** and click **OK**

- Put the LM324 on the window as shown
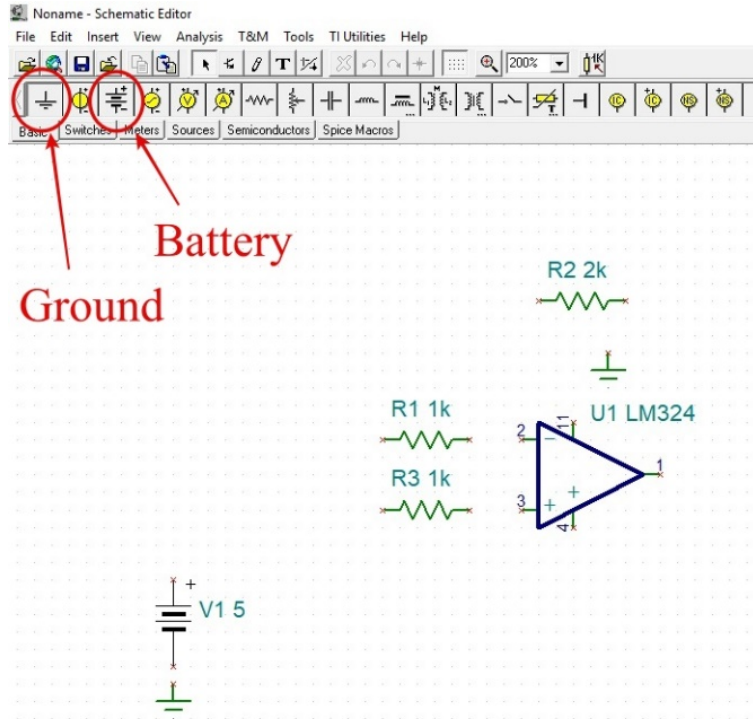
3) Add two 1K and one 2K resistors

- Press the **Basic** tab
- Select **Resistor** and put 3 resistors as shown

- Change R2 value to 2K by double click R2 and enter the value press OK
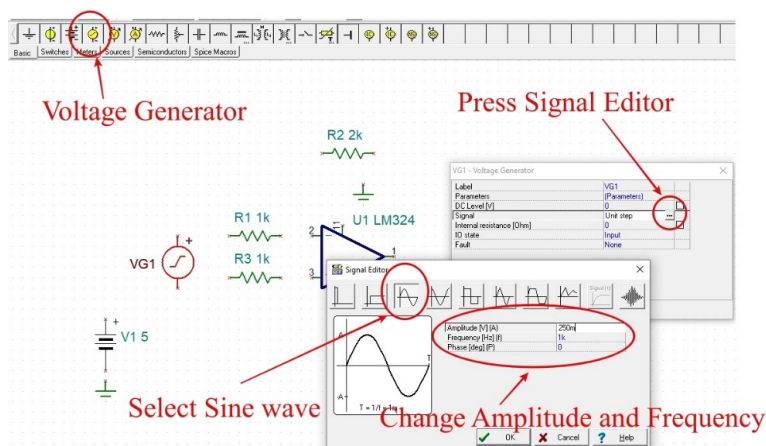

4) Add power supply (Battery) and Grounds to the circuit. Since the default Battery value is 5V, so we no need to change it.

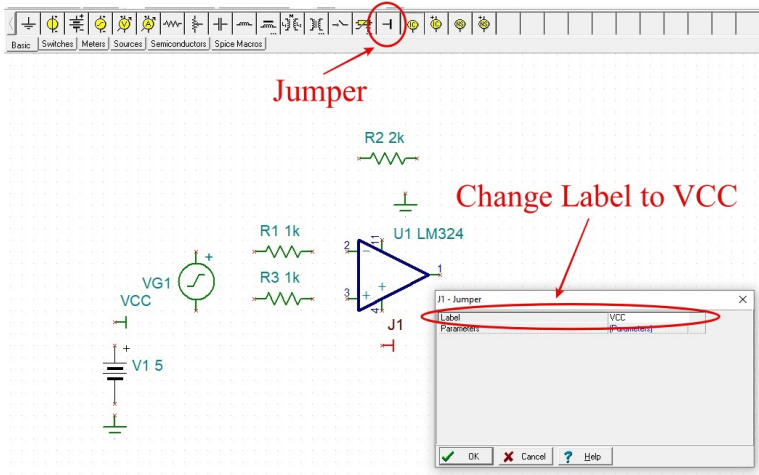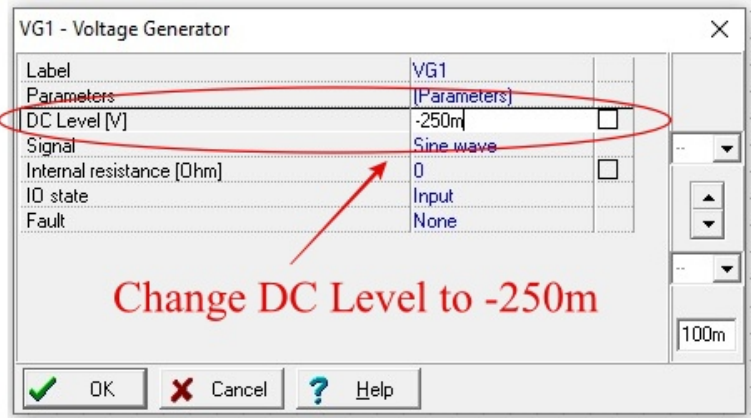5) Add input signal (**Voltage Generator**) to the circuit

- Add Voltage Generator and change the value by double click VG1
- Press **Signal Editor** button
- Select **Sine wave**
- Change the **Amplitude** value to **250m**
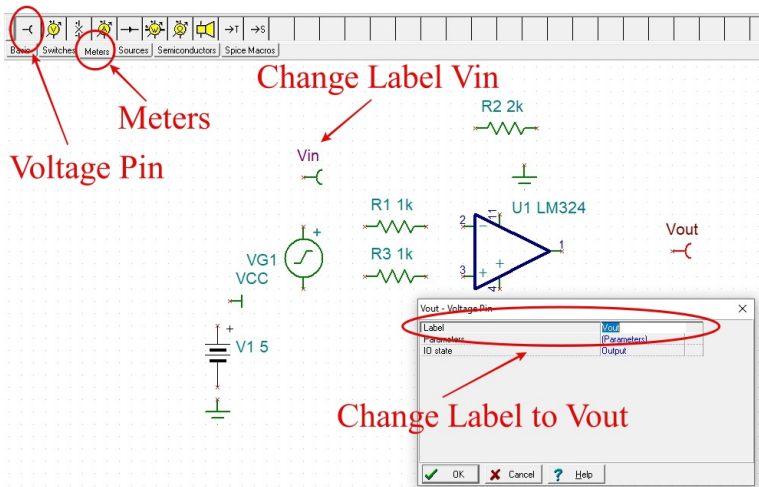- Change the **Frequency** to **1k**

- Change the **DC Level** to -250m

6) Add **Jumpers** to the circuit
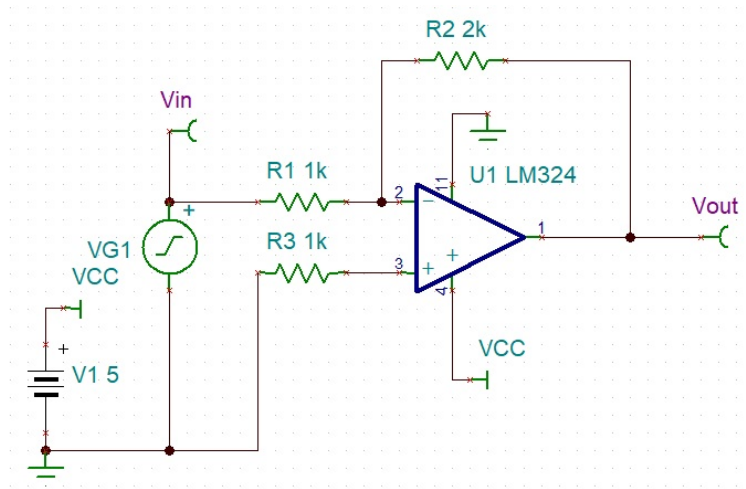
- Add two **Jumpers** and change their label to **VCC**

VG1 - Voltage Generator

Label	VG1
Parameters	[Parameters]
DC Level [V]	-250m
Signal	Sine wave
Internal resistance [Ohm]	0
IO state	Input
Fault	None

Change DC Level to -250m

Jumper

R2 2k

Change Label to VCC

R1 1k   U1 LM324

VG1
VCC   R3 1k

J1

J1 - Jumper

Label	VCC
Parameters	[Parameters]

V1 5

7) Add 2 **Voltage Pins** (V_{in},V_{out}) to the circuit

- Press the **Meters** tab
- Add 2 **Voltage Pins** and change the **Label** to V_{in} andV_{out}

Basic Switches Meters Sources Semiconductors Spice Macros

Change Label Vin

R2 2k

Meters

Vin

Voltage Pin

R1 1k   U1 LM324

Vout

VG1   R3 1k
VCC

Vout - Voltage Pin

|	
Label	Vout
Parameters	[Parameters]
IO state	Output

V1 5

Change Label to Vout

8) Connect all the components as shown in Figure 8

9) Simulate the circuit by adding the Oscilloscope

- Press **T&M** and select **Oscilloscope**
- Configure Oscilloscope setting: **Time/Div: 500u**, **Volts/Div: 500m**, **Channel: V_{in}** as following picture. Then press **Run** button. The V_{in} waveform will be displayed on the oscilloscope.
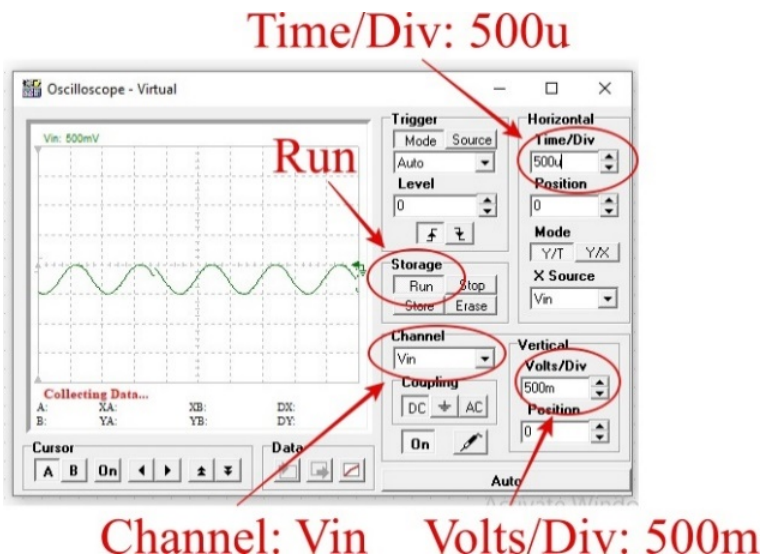
- Then select **Channel** to $\mathbf{V}_{out}$ and **Volts/Div: 500m**. The $\mathbf{V}_{out}$ waveform will be displayed on the oscilloscope with $\mathbf{V}_{in}$ waveform.

10) **[Task 1]** Capture your oscilloscope screen with V_{in} and V_{out} waveforms displayed on the screen. Attached the captured screen and based on the result of the simulation waveforms (use peak to peak Vp-p value of both V_{in} and V_{out} from the oscilloscope screen) state the voltage gain V_{out}/V_{in} in your lab report.
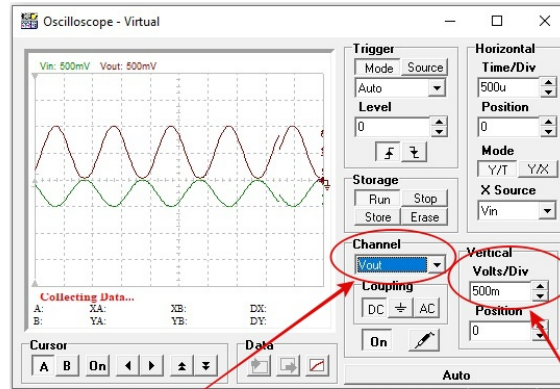

Select Channel to Vout   Volts/Div: 500m
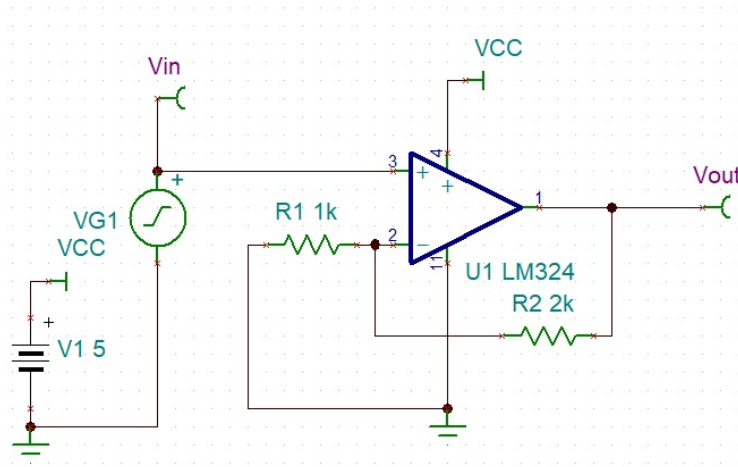
4 Experiment 2. Non-inverting Amplifier

In this experiment, you will construct and observe the Non-inverting amplifier. Record the input and output simulation waveforms.

1) Similar to Experiment 1, use TINA-TI simulator to construct Non-inverting amplifier as shown in the picture below. Configure input signal VG1 (**Voltage Generator**) as the following picture. **(DC Level: 250m, Sine wave, Amplitude: 250m, Frequency: 1k)**
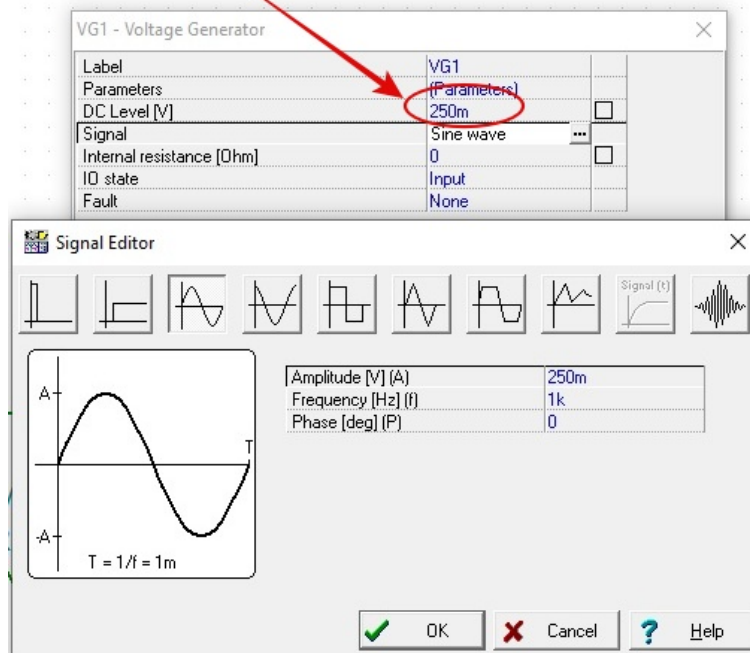
2) [**Task 2**]Capture your oscilloscope screen with V_{in} and V_{out} waveforms displayed on the screen. Attached the captured screen and based on the result of the simulation waveforms (use peak to peak Vp-p value of both V_{in} and V_{out} from the oscilloscope screen) state the voltage gain V_{out}/V_{in} in your lab report.


3) [**Task 3**] In your lab report, give your derivation and compare the calculated with simulation waveforms. Verify the gain relation of the non-inverting amplifier.

$$\frac{V_{out}}{V_{in}} \approx 1 + \frac{R_2}{R_1} \tag{1}$$

5   Experiment 3. Voltage Follower

In this experiment, you will construct and observe Voltage Follower. Record the input and output waveforms.

1) Similar to previous Experiments, use TINA-TI simulator to construct Voltage Follower as shown in the Figure 4. Configure Battery V1 to 5V, input signal VG1 (Voltage Generator) as shown in Figure 4. (DC Level: 250m, Sine wave, Amplitude: 250m, Frequency: 1k)
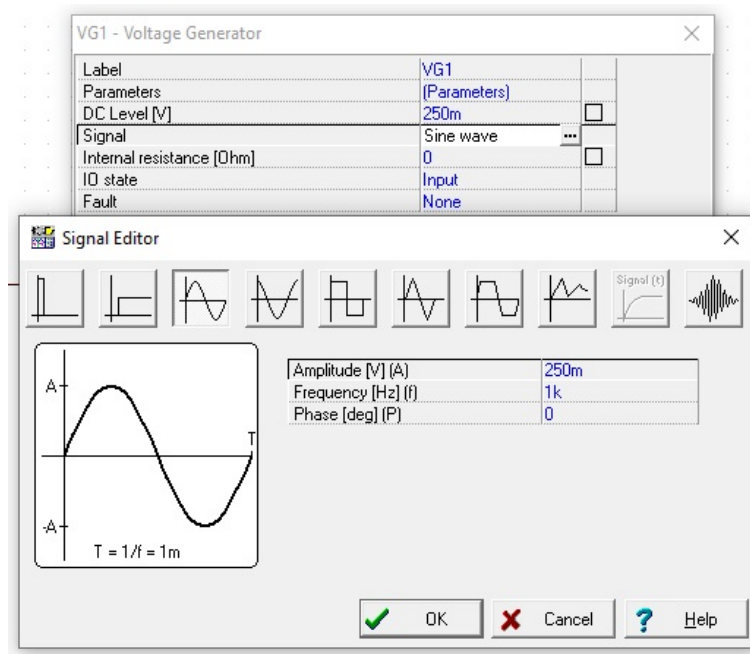
2) **[Task 4]** Capture your oscilloscope screen with V_{in} and V_{out} waveforms displayed on the screen. Attached the captured screen and based on the result of the simulation conclude the voltage gain V_{out}/V_{in} in your lab report.
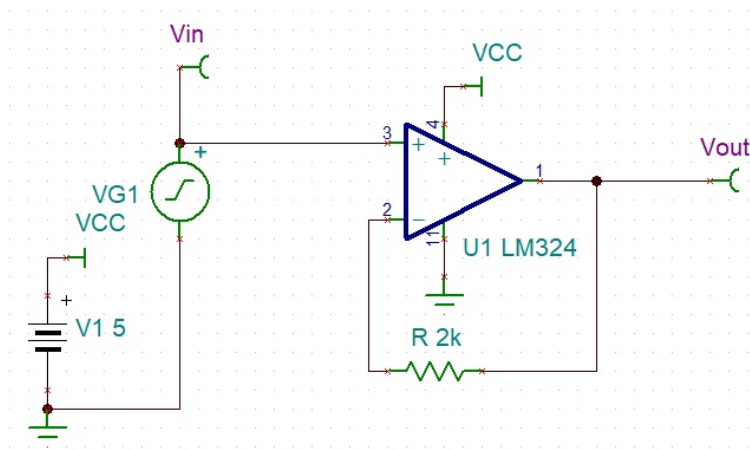
Plz send your reports to tutors, thank you!