

EE3122 Analogue Circuit Fundamentals Lab

In this lab, you will be able to design and operate some simple transistor and op-amp circuits. There will be three mini-projects that you need to finish. Each project chosen is very typical and will help you understand the analog circuits operating principle. You should follow the organization of the guide, record and explain clearly what you did in each part.

The ultimate goal of this lab aims to train up the students with the knowledge in analog circuit analysis, design of amplifier circuit using transistor / op-amp and understand the distortion in amplifier circuits, which will be very useful for the future study.

Scheduling:

This lab should be completed in three weeks—weeks 8,11,12.

Schedules:

Thursday, 3-5:50 PM, Group (L01)

Tuesday, 12-2:50 PM, Group (L02)

Grouping:

Individual experiments to be done

Location of Lab:

Control Systems laboratory, Yeung Kin Man Academic Building P1404.

Report Requirement:

The students need to submit a scientific report after finishing the project. In the lab report, students are supposed to write the procedure of the project, calculated and measured results, discussions, problems encountered, how to solve them, and conclusions.

Other Information:

Please note the University health declaration.

Lab 1: Common emitter amplifier without Bypass Capacitor

Transistor amplifier amplifies an AC input signal that alternates between positive and negative value. Then “presetting” the amplifier’s circuit configuration is required so that the transistor can operate between these two maximum or peak values. This can be achieved using a process known as Biasing. Biasing is very important in amplifier design as it establishes the correct operating point of the transistor amplifier ready to amplify signals, and can reduce distortion to the output signal.

The purpose of this lab is to get familiar with common emitter amplifier and to design a small signal voltage amplifier. In this project, students are supposed to build a common emitter amplifier with emitter resistor and without bypass capacitor, which could help them to get a better understanding of the principle of operational amplifier.

Objectives:

1. Able to design a small signal voltage amplifier.
2. Able to understand the design method of common emitter amplifier.
3. Able to identify the reasons for the errors.

Components and Equipment:

1. 1K/10K/100K resistors;
2. 0.047uF capacitor
2. 2N3904 (NPN) Transistor;
3. Breadboard board;
4. DC power supply, Signal Generator, oscilloscope.

Procedures:

1. Set up the common emitter amplifier circuit shown in Figure 1 on the breadboard.
2. Calculate the transistor bias voltage. Evaluate the circuit gain and the theoretical output voltage.
3. Inject sin-wave signal to the circuit and measure the output of the circuit.
4. Compare the measured output voltages with the theoretical result.
5. Change the frequency of the source and try to draw the frequency response curve.
6. Change the amplitude of the source and observe the distortion.

Note: Tabulate all the measured results and show them to the laboratory supervisor before you leave.

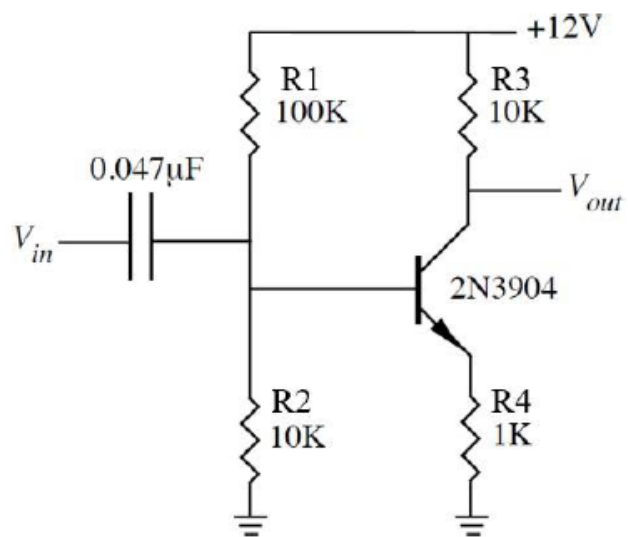


Figure 1 Common Emitter Amplifier

Lab 2: Push-Pull follower

A push-pull amplifier is a type of electronic circuit that uses a pair of active devices that alternately supply current to, or absorb current from, a connected load. This kind of amplifier can enhance both the load capacity and switching speed. Push-pull output does not require an external resistor, and its output logic level depends on the comparator power supply voltage. A push-pull amplifier is more efficient than a single-ended "class-A" amplifier. The output power that can be achieved is higher than the continuous dissipation rating of either transistor or tube used alone and increases the power available for a given supply voltage. Symmetrical construction of the two sides of the amplifier means that even-order harmonics are cancelled, which can reduce distortion. DC current is cancelled in the output, allowing a smaller output transformer to be used than in a single-ended amplifier. However, the push-pull amplifier requires a phase-splitting component that adds complexity and cost to the system; use of center-tapped transformers for input and output is a common technique but adds weight and restricts performance. If the two parts of the amplifier do not have identical characteristics, distortion can be introduced as the two halves of the input waveform are amplified unequally. Crossover distortion can be created near the zero point of each cycle as one device is cut off and the other device enters its active region.

Objectives:

1. Able to measure, analyze and apply feedback to a simple push-pull follower circuit
2. Able to compare the theory with measured parameters of the circuit link.
3. Able to identify the reasons of distortion and to apply DC bias to the enhance the performance of the circuit link.

Procedures:

1. Construct the push-pull output stage shown in Fig. 1(a), driving the input with a sine wave. Sketch and compare the input and output wave forms, and explain the distortion near the zero crossings.
2. Finding the gain for large amplitude signals and determine the maximum amplitude before noticeable clipping.
3. Can you demonstrate a larger output amplitude from this circuit than from the biased common emitter? Is the DC component essentially absent, as claimed in the notes?
4. When add the bias network shown in Fig. 1(b). Biasing should remove or minimize the cross- over distortion. Does it? Sketch the wave forms, noting any remaining irregularities. What is the maximum undistorted amplitude from this arrangement?

Checkpoints: Please call TA and show these

1. Input & output for Fig. 1a showing crossover distortion
2. Maximum amplitude before clipping for Fig. 1a
3. Reduced crossover distortion for Fig. 1b
4. Maximum amplitude before clipping for Fig. 1b

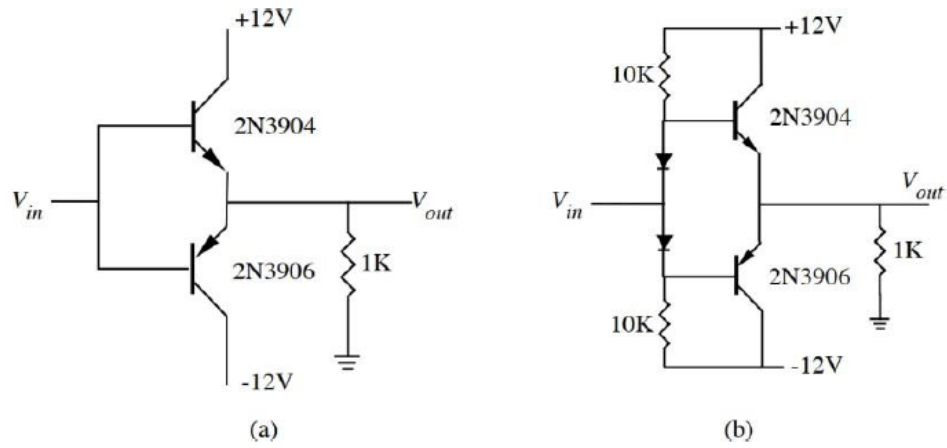


Fig. 1 Push-pull amplifier circuits, with and without biasing.

Lab 3: Basic op-amp circuits

The purpose of this mini project is to get familiar with some basic operational amplifier circuits. When integrated the operational amplifier, resistors, capacitors, and other devices together, we can get some basic amplifier circuits such as summing amplifier, difference amplifier and integrator. In this project, students are supposed to build the summing amplifier and difference amplifier using the operational amplifier and resistor, which could help them to get a better understanding of the principle of operational amplifier.

Objectives:

1. Able to identify the pin configuration of LM741 op-amp.
2. Able to understand the design method of summing amplifier and difference amplifier and finish the corresponding circuit design.
3. Able to get the desired parameters of the circuits and compare the theory with measured parameters.
4. Able to identify the reasons for the errors.

Components and Equipment:

1. 4.7K/10K/100K resistors;
2. LM741 op-amp;
3. Breadboard board;
4. Dual DC power supply, Audio Signal Generator;
5. Digital multi-meter and Dual Channel CRO.

Procedures:

PART I

1. Figure 1 shows the circuit of a summing amplifier. Calculate the relationship between the output voltage (V_o) and the input voltage (V_{I1} and V_{I2}).
2. Based on the given circuit, set up the circuit on the circuit board.
3. Test the circuit. Make the input port grounded and measure the output of the circuit. The output should be close to zero.
4. Inject sine-wave signal to the input terminals. For different amplitude values of V_{I1} and V_{I2} , measure the output voltage V_o and record the corresponding values.
For the first set, $V_{I1}=50 \text{ mV}_{P-P}$, $V_{I2}= 50 \text{ mV}_{P-P}$
For the second set, $V_{I1}=100 \text{ mV}_{P-P}$, $V_{I2}= 50 \text{ mV}_{P-P}$ For the third set, $V_{I1}=100 \text{ mV}_{P-P}$, $V_{I2}= 100 \text{ mV}_{P-P}$ For the fourth set, $V_{I1}=150 \text{ mV}_{P-P}$, $V_{I2}= 100 \text{ mV}_{P-P}$
5. Compare the theoretical and measured output voltage amplitude. Then, calculate the absolute errors.

Checkpoints: Please call TA and show these

1. Fig. 1 –step 3 with grounded input.
2. Fig. 1 –step 4 with sine wave input

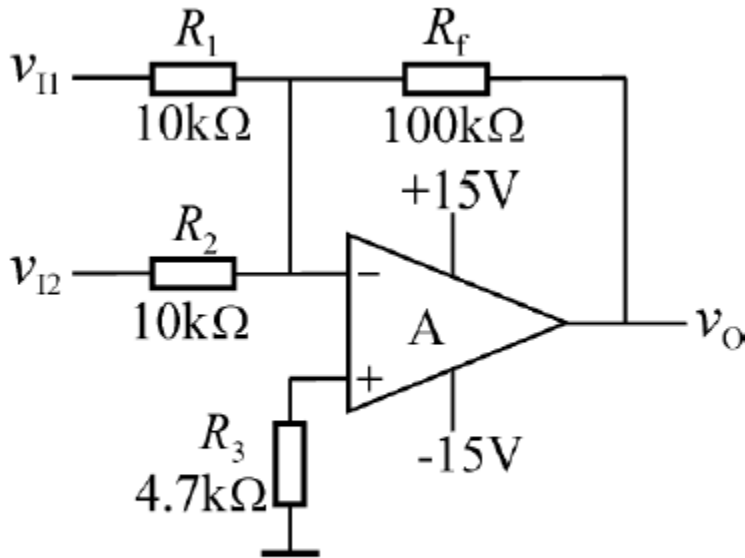


Figure 1 Summing amplifier

PART II

1. Figure 2 shows the circuit of a difference amplifier. Calculate the relationship between the output voltage (V_o) and the input voltage (V_{I1} and V_{I2}).
2. Based on the given circuit, set up the circuit on the circuit board.
3. Test the circuit. Make the input port grounded and measure the output of the circuit. The output should be close to zero.
4. Inject sine-wave signal to the input terminals. For different amplitude values of V_{I1} and V_{I2} , measure the output voltage V_o and record the corresponding values.

For the first set, $V_{I1}=50 \text{ mV}_{P-P}$, $V_{I2}= 100 \text{ mV}_{P-P}$

For the second set, $V_{I1}=50 \text{ mV}_{P-P}$, $V_{I2}= 200 \text{ mV}_{P-P}$

For the third set, $V_{I1}=50 \text{ mV}_{P-P}$, $V_{I2}= 300 \text{ mV}_{P-P}$

For the fourth set, $V_{I1}=100 \text{ mV}_{P-P}$, $V_{I2}= 300 \text{ mV}_{P-P}$

5. Compare the theoretical and measured output voltage amplitude. Then, calculate the absolute errors.

Checkpoints: Please call TA and show these

1. Fig. 2 –step 3 with grounded input.
2. Fig. 2 –step 4 with sine wave input

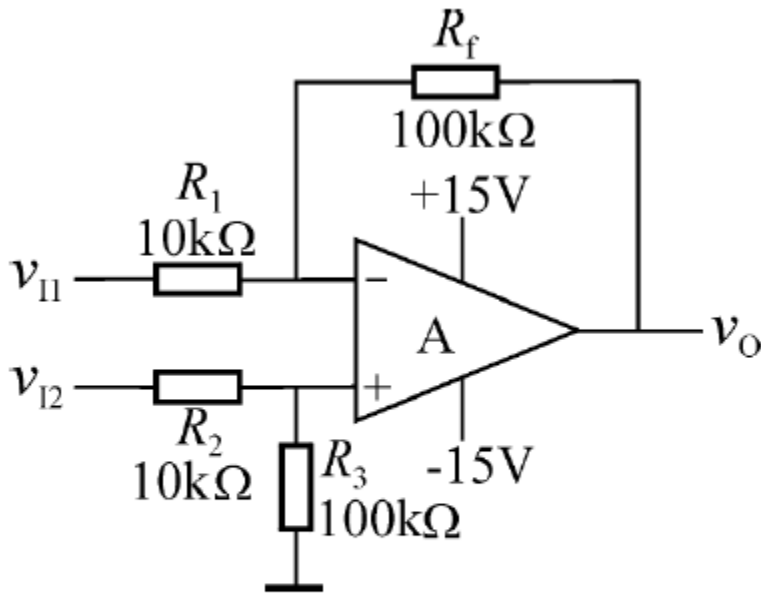


Figure 2 Difference amplifier

Discussion -- to be answered in final report

1. Is there difference between the calculated output voltage and measured output voltage? If so, analyze the reasons for the difference between them.
2. What is the function of R_3 in the circuit? Give a brief description.
3. For the summing amplifier, when the values of R_1 and R_2 are changed, do we need to tune the values of R_3 ? Can you give the relationship between R_1 , R_2 , R_f and R_3 for the summing amplifier?
4. For the difference amplifier, when the values of R_1 and R_2 are changed, do we need to tune the values of R_3 ? Can you give the relationship between R_1 , R_2 , R_f and R_3 for the difference amplifier?