192 252
Electrical Engineering Laboratory II Manual

Part I: Electronic Circuits

Department of Electrical Engineering
Khon Kaen University
Academic Year 2017/2018

Table of Contents

General Guidelines

Often, a damaged equipment is because of negligence or due to the assumption that nothing serious could happen. Therefore, you should always keep in mind the following instructions when performing the experiments:

1. NEVER ever short any outputs of the power supply.
2. When powering up the circuit, check the voltage reading.
3. Never try to measure the current of the power supply by connecting the ammeter straight across the power supply. Always make sure that there is at least a series resistor.
4. When making changes to the circuit, turn off the power. It is during these occasions where you will most likely short the outputs of the power supply.
5. Never connect a probe to the signal generator co-ax output. The probes are reserved for scopes only.
6. Never connect an LED to the power supply without adding a series resistor.
7. When connecting electrolytic capacitors to DC voltages, always observe the correct polarity.

Below are shorthand symbols you will encounter in the lab manual:

P This means that you should follow the procedure given. You do not need to take down any measurement at this point. Any question asked in this section need not be answered, but are there to let you think about the question by yourself.

R This indicates that you have to record the data or take down some readings. Sketching of the measured waveforms or plotting of the measured data might be required in this section.

Q This indicates that you have to answer the questions.

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-01: DIODE CHARACTERISTICS AND A FULL-WAVE BRIDGE RECTIFIER CIRCUIT

OBJECTIVES

1. To investigate the current voltage relationships of a PN junction diode.
2. To familiarize the students to the measurements of voltage and current in electronic circuits.
3. To be able to use diodes as a bridge rectifier circuit.

INTRODUCTION

Semiconductor devices are electronic components that exploit the electronic properties of semiconductor materials, principally silicon, germanium, and gallium arsenide. Semiconductor devices have replaced thermionic devices (vacuum tubes) in most applications. They use electronic conduction in the solid state as opposed to the gaseous state or thermionic emission in a high vacuum.

Semiconductor devices are manufactured both as single discrete devices and as integrated circuits (ICs), which consist of a number—from a few to millions—of devices manufactured and interconnected on a single semiconductor substrate.

There are widely many semiconductor devices used in electronic circuits and systems. One example of such devices in discrete form is silicon PN junction diodes.

THEORY

The PN junction diode is a device formed from a junction of n-type and p-type semiconductor material. The lead connected to the p-type material is called the **anode** (A) and the lead connected to the n-type material is the **cathode** (K). In general, the cathode of a diode is marked by a solid line on the diode, as shown in Figure 1.

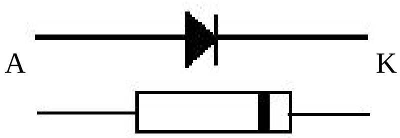


Figure 1. Diode circuit symbols

The equation for the voltage and current relation is known as *Shockley equation*.

$$I_D = I_s\left(e^{V_D/nV_t} - 1\right) \qquad (1)$$

where the emission coefficient n varies between 1-2 depending on the level of recombination effect, I_S is the diode reverse saturation current, and $V_t = kT/q$ is the thermal voltage (approximately 25.8 mV at 300 K).

The primary function of the diode is the rectification. When it is forward biased (higher potential at the anode lead), it will pass current. When it is reverse biased (higher potential at the cathode lead), the current is blocked. The typical characteristic curves of an ideal diode and a real diode are illustrated in Figure 2.

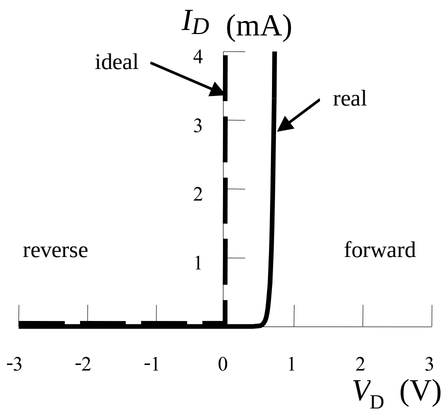


Figure 2. Typical ideal and real diode characteristics.

One of the most popular applications of a PN junction diode is *rectification* which is simply defined as the conversion of alternating current (AC) to direct current (DC). The circuit, shown in Figure 3, is a *half-wave rectifier* circuit using only one PN junction diode. This circuit only allows one half of an AC waveform to pass through to the load.

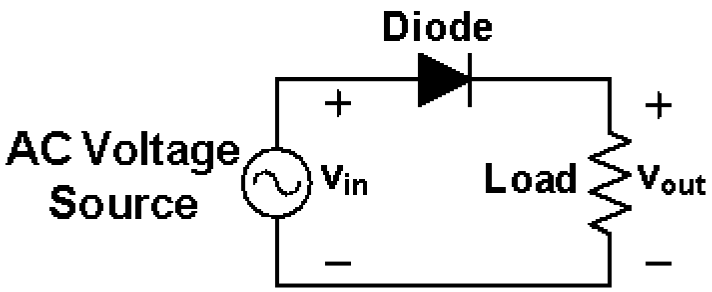


Figure 3. A half-wave rectifier circuit.

When four diodes are connected as shown in Figure 4, the circuit is called a *full-wave bridge rectifier*. The input to the circuit is applied to the diagonally opposite corners of the network, and the output is taken from the remaining two corners.

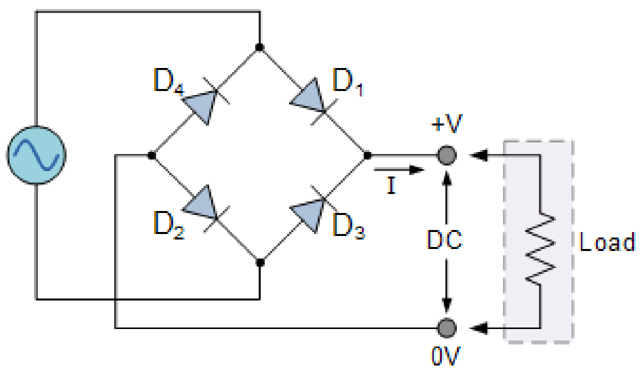


Figure 4. A full-wave bridge rectifier circuit.

The four diodes labeled D_1 to D_4 are arranged in "series pairs" with only two diodes conducting current during each half cycle. During the positive half cycle of the supply, diodes D_1 and D_2 conduct in series while diodes D_3 and D_4 are reverse biased and the current flows through the load as shown in Figure 5.

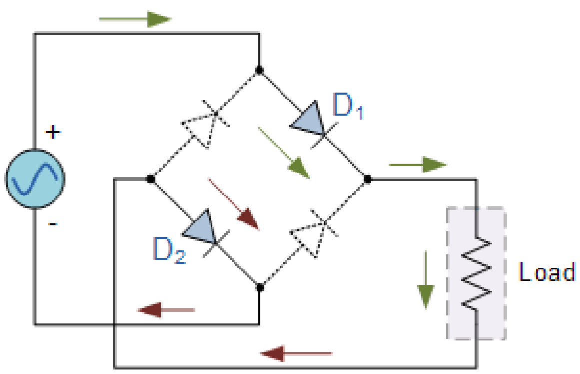


Figure 5. The circuit operation during a positive half-cycle of the supply.

Figure 6 shows the circuit operation during the negative half cycle of the supply, diodes D_3 and D_4 conduct in series, but diodes D_1 and D_2 switch "OFF" as they are now reverse biased. The current flowing through the load is the same direction as before.

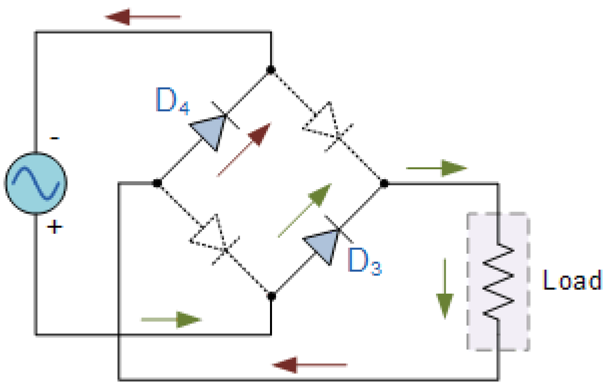


Figure 6. The circuit operation during a negative half-cycle of the supply.

Although four individual diodes can be used to make a full wave bridge rectifier, pre-made bridge rectifier components are available "off-the-shelf" in a range of different voltage and current sizes, as shown in Figure 7, that can be selected for full-wave rectifier applications.

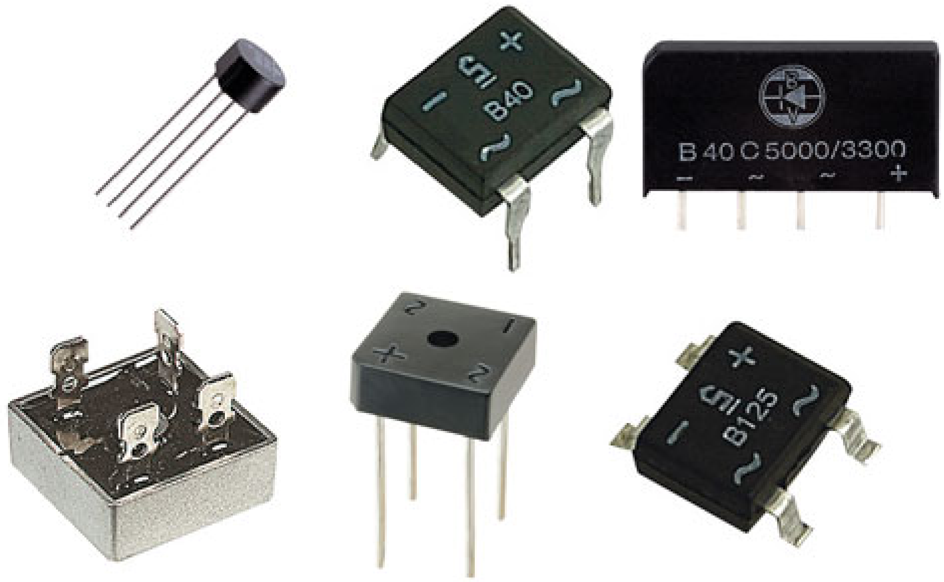


Figure 7. Examples of bridge diodes.

PRELIMINARY REPORT

Search for a datasheet of 2W04 bridge diode and answer the following questions:

- What is the average rectified output current of this bridge diode?
- What is the forward voltage drop (per element) of this bridge diode?

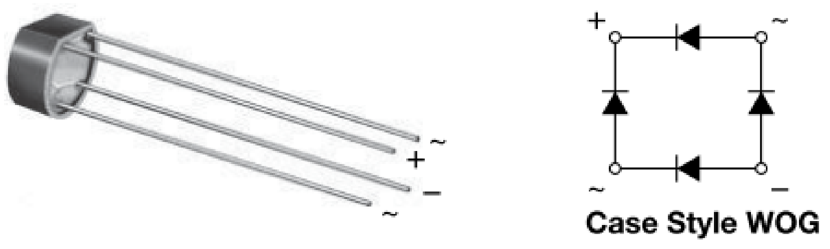
APPARATUS

1. DC power supply
2. Multimeter
3. Protoboard
4. Function generator and oscilloscope
5. 2W04 Diode Bridge
6. 1 kΩ 1/4W Resistor

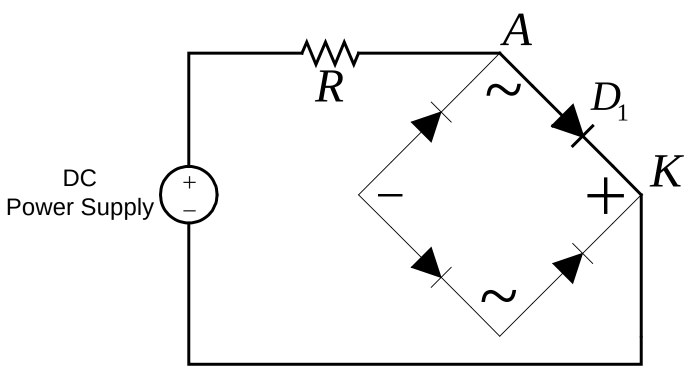
PROCEDURE (P1 – P4 and R1 – R4; 40 points)

Diode V-I characteristic

P1 Identify one diode element of the 2W04 bridge diode to be used as a single diode for this experiment. For example, if a diode element D_1 is selected, as shown in Figure 8(b), then perform the following experiment with the "~" terminal as an anode (*A*) and the "+" terminal as a cathode (*K*).



(a)



(b)

Figure 8. (a) 2W04 pin diagram. (b) Circuit for diode V-I characteristic measurement.

P2 Construct a single diode test circuit as shown in Figure 8(b). Use the value of resistor R = 1 kΩ to limit the current through the diode.

P3 Set the DC output voltage from the DC power supply (V_S) between -2 to 2 V according to Table 1. In each step use a multimeter to measure the values of the voltages across the resistor V_R and across the diode V_D. The diode current can then be calculated from the voltage across the resistor V_R divided by the value of resistor.

$$I_D = \frac{V_R}{R} \quad (2)$$

R1 **(10 points)** Fill V_D and all the diode currents in Table 1 and plot the V_D-I_D characteristic of this diode.

Table 1

V_S (V)	0	0.5	0.75	1	1.25	1.5	2	-0.5	-1	-2
V_D										
I_D										

Bridge diode as a full-wave rectifier

P4 Figure 9 shows a *full-wave rectifier* circuit using a bridge diode. Use a function generator as an AC voltage source to supply a sinusoidal voltage with 10 V_{P-P} amplitude and 100 Hz frequency. Connect a load resistor = 1 kΩ to the output of the bridge diode. Use oscilloscope to display voltage waveforms at the AC source and the load.

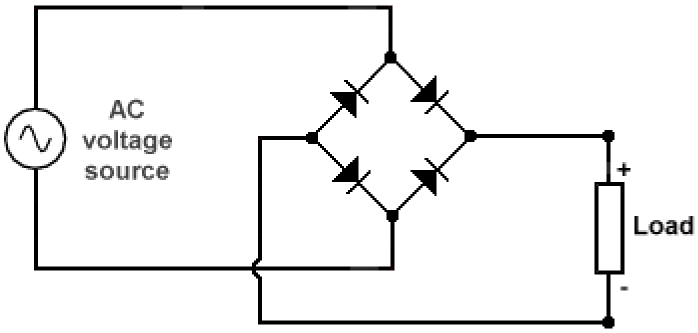


Figure 9. Full-wave rectifier circuit.

R2 **(10 points)** Sketch the input waveform measured at the AC voltage source and the output waveform measured at the load. Figure 10 shows an example of the waveforms that should be obtained from your measurement.

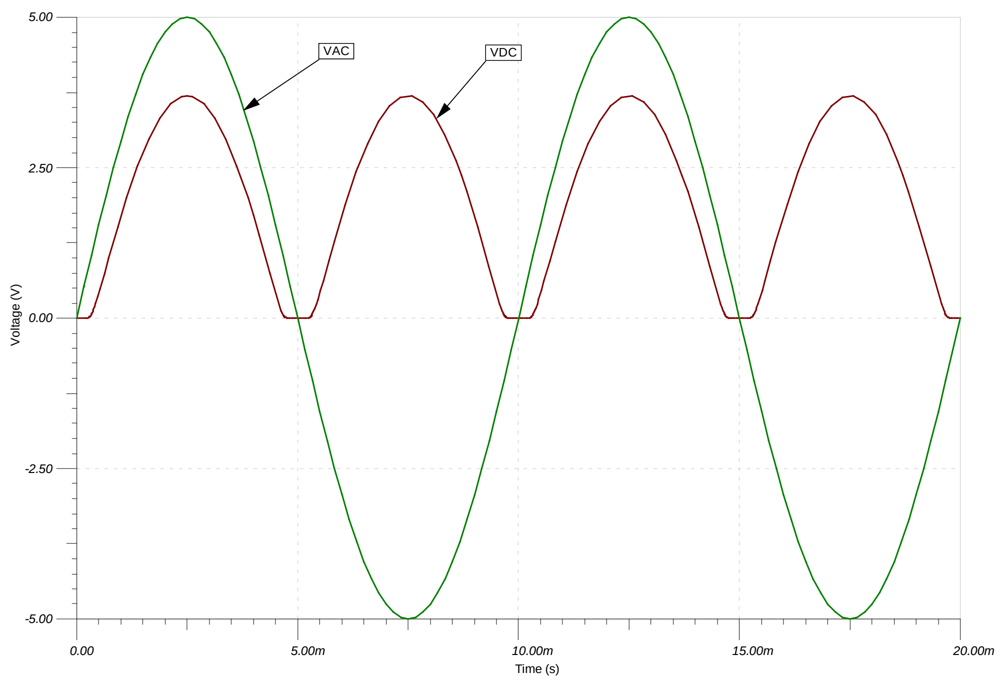


Figure 10. Voltage waveforms of a full-wave rectifier circuit.

R3 (10 points) Increase the frequency of the function generator to the maximum value (20 kHz). Sketch the waveforms at the AC source and load.

R4 (10 points) Adjust the frequency back to 100 Hz and reduce the amplitude to 1 V_{P-P}. Sketch the waveforms at the AC source and load.

QUESTIONS (Q1 – Q5; 20 points)

Q1 (4 points) What can be concluded form the relation between voltage and current of the PN junction diode?

Q2 (4 points) Explain the consequences if the resistor R = 1 kΩ in Figure 8 is not used; i.e. connect the DC power supply direct to the diode D_1.

Q3 (4 points) From the full-wave rectifier circuit experiment, did you notice that the amplitude of the voltages at the AC source and the load are different? Explain why.

Q4 (4 points) When the frequency of the AC source is increased, did you notice any differences of the voltage at the load? If so, explain why.

Q5 (4 points) When the amplitude of the AC source is decreased, did you notice any differences of the voltage at the load? If so, explain why.

Boonying Charoen
October 2013

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-02: DC POWER SUPPLY CIRCUIT

OBJECTIVES

1. To design and construct a 12 V DC power supply circuit.
2. To be able to build a full-wave bridge rectifier, filter, and regulator circuits.
3. To use a 3-terminal IC as a voltage regulator in the DC power supply circuit.

INTRODUCTION

Most of the direct current (DC) power used in electronic circuits is derived by converting 50 Hz, 220 V_{RMS} alternating current (AC) power to direct current power. This AC to DC conversion usually involves a step-down transformer, rectifier, filter, and a regulator. The step-down transformer is used to decrease the AC line voltage from 220 V_{RMS} to a lower value near the dc voltage needed. The output of the step-down transformer is then fed into a diode rectifier circuit that only outputs positive halves of the input sinusoid. A filter is then used to smooth the rectifier output to achieve a nearly constant dc voltage level. Figure 1 shows an example of an unregulated DC power supply. The step-down transformer decreases the AC voltage to 12 V_{RMS} (approximately equals to 17 V peak).

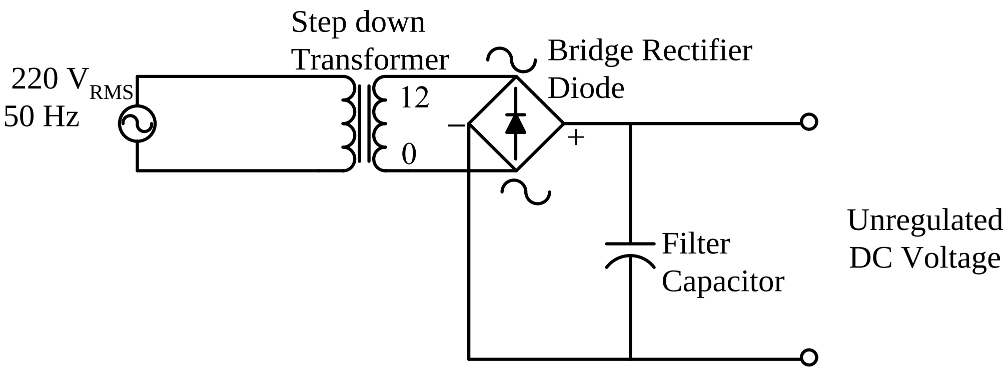


Figure 1. Unregulated DC power supply.

A regulator circuit can be added after the filter to ensure a constant output voltage in spite of changes in load current and input voltages. One of the most popular and easy to use voltage regulator is the 3-terminal fixed voltage IC regulator as shown in Figure 2.

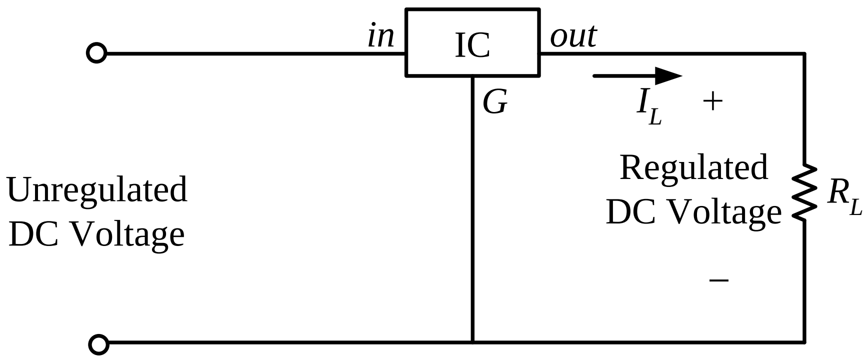


Figure 2. Regulated DC power supply using a 3-terminal IC regulator.

THEORY

The most simple and most basic type of power supply filter is a single shunt capacitor filter, as shown in Figure 3. We saw in the previous lab that the full-wave rectifier produces a full-wave output waveform which is twice that of the frequency of the input supply frequency. We can increase its average DC output level even higher by connecting a suitable filter or smoothing capacitor across the output of the bridge circuit as shown in Figure 3.

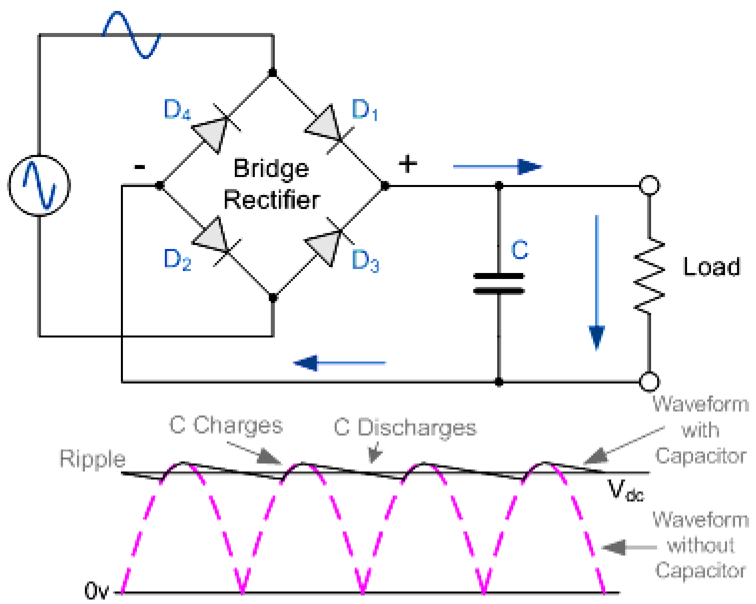


Figure 3. Full-wave bridge rectifier with a capacitor filter.

The filter capacitor converts the full-wave rippled output of the rectifier into a smoother DC output voltage. Generally, for DC power supply circuits, the filter capacitor is an aluminium electrolytic type that has a capacitance value of 100 µF or more with repeated DC voltage pulses from the rectifier charging up the capacitor to peak voltage. However, there are two important parameters to consider when choosing a suitable filter capacitor and these are its *Working Voltage*, which must be higher than the no-load output value of the rectifier and its *Capacitance Value,* which determines the amount of ripple that will appear superimposed on top of the DC voltage.

Too low a capacitance value and the capacitor has little effect on the output waveform. But if the filter capacitor is sufficiently large enough (parallel capacitors can be used) and the load current is not too large, the output voltage will be almost as smooth as pure DC. As a general rule of thumb, we are looking to have a ripple voltage as little as possible. For example, a typical value is less than 100 mV peak to peak for a 10 V DC power supply (or less than 1%).

The maximum ripple voltage present for a full-wave rectifier circuit is not only determined by the value of the filter capacitor but by the frequency and load current, and is given as

$$v_{Ripple} \propto \frac{I_L}{f\,C} \tag{1}$$

One practical and cheaper approach to reduce ripple voltage is to use an off the shelf 3-terminal IC voltage regulator, such as a LM78xx (where "xx" stands for the output voltage rating) for a positive output voltage or its inverse equivalent the LM79xx for a negative output voltage which can reduce the ripple by more than 70 dB (see datasheet) while delivering a constant output current of over 1 A. Figure 4 depicts picture, pin diagram, and a typical circuit of a 12 V 3-terminal IC voltage regulator in TO220 case style.

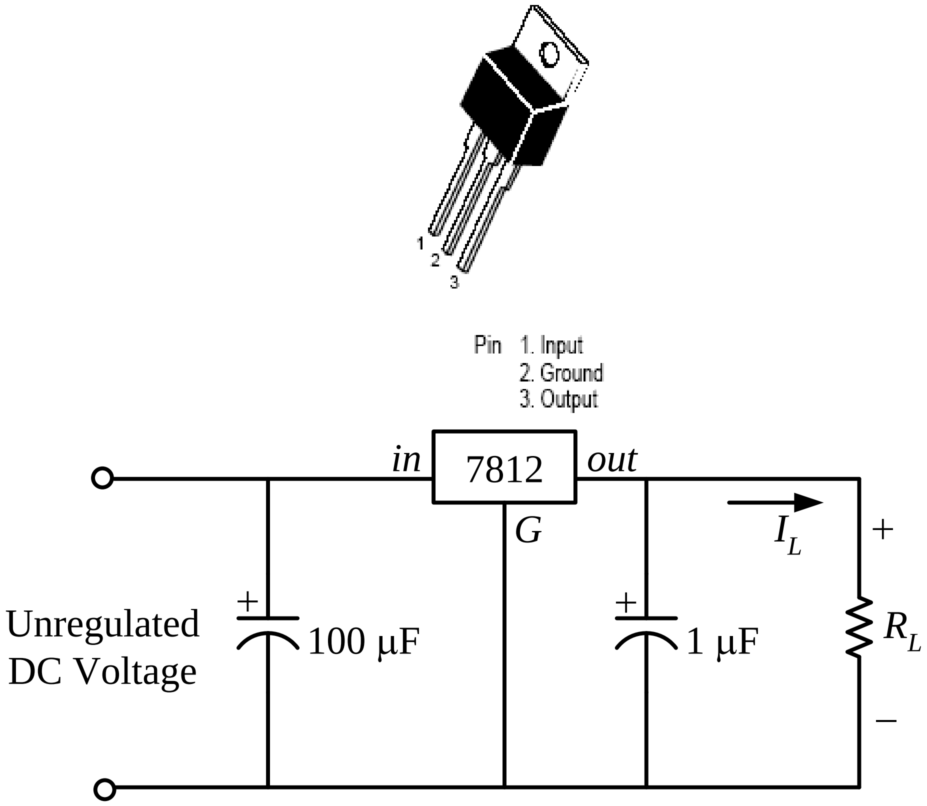


Figure 4. 3-terminal IC voltage regulator.

PRELIMINARY REPORT

Search for a datasheet of 7812 IC regulator and answer the following questions:

1. What is the maximum input voltage to this IC?
2. What is the value of ripple rejection?
3. What is the value of drop out voltage?

APPARATUS

1. Multimeter
2. Oscilloscope
3. Protoboard
4. 220/12 V_{RMS} transformer
5. 2W04 diode bridge and LED
6. 100-Ω, 5-W and 1-kΩ, ¼-W resistors
7. 100-µF, 50 V; 1-µF, 50-V; 1-µF, 25-V electrolytic capacitors

PROCEDURE (P1 – P4 and R1 – R8; 40 points)

Full-wave bridge rectifier with a capacitor filter

P1 Plug in the AC power plug which is connected to the transformer primary winding into an AC power 220 V_{RMS} outlet and turn on the switch.

R1 **(5 points)** Measure the transformer secondary AC voltage using both multimeter and oscilloscope. Compare the measured results.

P2 Construct the circuit as shown in Figure 1, with a 100-µF, 50-V electrolytic capacitor as a filter. Connect a 100-Ω, 5-W load resistor *R* across the filter capacitor. Use the oscilloscope to display voltage waveforms at the input of the bridge diode and at the load resistor.

R2 **(5 points)** Record the waveforms at the bridge diode input and load resistor.

R3 **(5 points)** Remove the 100-µF capacitor from the circuit in Figure 1. Record the waveforms at the bridge diode input and load resistor.

R4 **(5 points)** Now, use a 1-µF, 50-V capacitor as a filter capacitor in the circuit (Figure 1). Record the waveforms at the bridge diode input and load resistor.

3-terminal IC voltage regulator

P3 Construct the circuit as shown in Figure 4, with a 100-µF, 50-V electrolytic capacitor as an input filter and a 1-µF, 25-V electrolytic capacitor as an output filter. A 100-Ω, 5-W load resistor R_L is connected to the IC 7812 output. Connect the output of the bridge diode (unregulated DC voltage) to the IC 7812 input. Use the oscilloscope to display the voltage waveforms at the input of the IC 7812 and at the load resistor.

R5 **(5 points)** Record the waveforms at the IC 7812 input and the load resistor.

R6 **(5 points)** Calculate for the load current by using Equation 2.

$$I_L = \frac{V_L}{R_L} \tag{2}$$

R7 **(5 points)** Use a multimeter to measure the voltage difference between the input and output of the IC 7812 voltage regulator. Calculate for the power dissipation of the IC 7812 voltage regulator by using Equation 3.

$$P_{7812} = (V_{in} - V_{out}) I_L \tag{3}$$

P4 Use your index finger to touch the IC 7812 body, can you feel the heat that is being dissipated from the IC? Move your index finger to touch the load resistor body, can you feel the heat that is being dissipated from the load? Can you estimate the body temperature of both devices?

P5 It is common to connect an LED to display output status of the DC power supply. A series resistor is selected to limit the current to an appropriate value. In this experiment, connect a 1-kΩ, ¼-W resistor as shown in Figure 5.

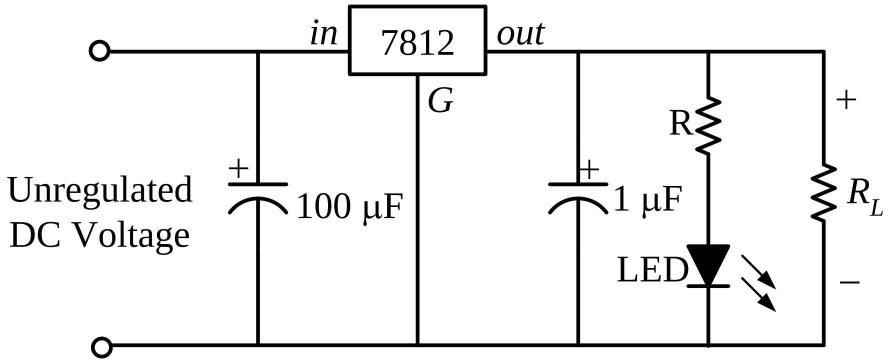


Figure 5. LED indicator connection

R8 **(5 points)** Use a multimeter to measure the voltage across the resistor R. Then, calculate for the LED current.

QUESTIONS (Q1 – Q5; 20 points)

Q1 **(4 points)** From the oscilloscope and multimeter measurements of the transformer output voltage, can you notice any difference in the obtained voltage values? If so, explain why.

Q2 **(4 points)** From the full-wave rectifier circuit with a filter capacitor, can you notice ripple voltages? If not, explain why.

Q3 **(4 points)** From the full-wave rectifier circuit experiment, do you notice that the ripple voltage at the load increases when the value of capacitance is reduced? Explain why.

Q4 **(4 points)** What can be concluded form the heat source that is being generated at the 7812 IC voltage regulator?

Q5 **(4 points)** Suggest a method to check whether a heat sink is required for the IC voltage regulator used in your DC power supply design.

FINAL REPORT

Write a report on how this experiment is performed which should include the followings:

1. Discuss the waveforms obtained at each section of the power supply.
2. Explain the results obtained from the full-wave rectifier circuit with different capacitance values.
3. Discuss the results when a voltage regulator is added and conclude the overall performances of the complete DC power supply.

Boonying Charoen
October 2013

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-03: BJT CHARACTERISTICS AND BIASING CIRCUITS

OBJECTIVES

1. To understand the relationship between base current and the collector current
2. To be able to construct and to read the V-I characteristic curve of a BJT
3. To see differences among biasing with and without DC feedback

INTRODUCTION

Bipolar junction transistor (BJT) is a basic building block of many electronic systems. In principle, transistors can be considered as current valves, which are controlled by current. In the first part of this experiment, the mechanism of this current controlled current valve will be demonstrated through the concept of V-I characteristic curves of the BJT. The curve obtained in this experiment shall exhibit all three BJT operation modes.

To use the BJT, ones must set the BJT to its operating point. Doing so is called biasing. It is known that the current gain (β) of the BJT is not as reliable as students might thought. It depends on the operating temperature and many other things. The current gain obtained from the datasheet is merely a typical value under a laboratory condition. A DC feedback resistor makes the biasing circuit more stable. In the later parts of this experiment, students shall see how unstable the biasing circuit without the feedback is, and how the feedback resistor helps.

THEORY

BJTs come in many shapes (aka cases or packages) as shown in Figure 1. To avoid confusion, engineers use electronic symbols to draw schematics instead of picture. Figure 2 shows the BJT symbols, both NPN and PNP. To identify pin connection, students should consult the device datasheet.

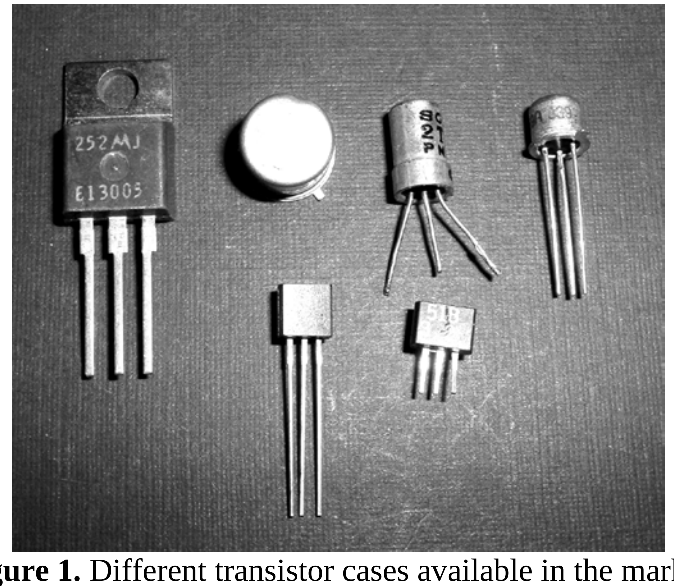


Figure 1. Different transistor cases available in the market.

(Photo from http://commons.wikimedia.org/wiki/File:Transistors.agr.jpg)

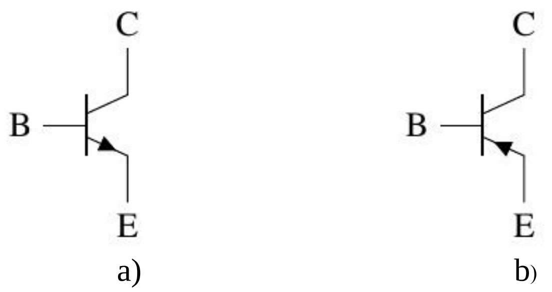


Figure 2. Schematic symbols of (a) NPN BJT and (b) PNP BJT.

In general, we operate BJTs in three modes namely, cutoff mode, saturation mode, and forward active mode. There is also another mode called reverse active mode. But this is not covered in this experiment.

Cutoff Mode

If V_{BE} is less than the threshold voltage, which is around 0.7 V, the BJT is not conducting current. We consider C-E path as an open circuit.

Saturation Mode

If V_{BE} is higher than the threshold voltage but V_B is getting closed to or greater than V_C, the BJT is almost fully conducted. That is, we consider C-E path as a closed circuit. Noted that V_{CE} will be low. But it will not reach 0 V.

If we can control a BJT to enter the cutoff mode and the saturation mode by controlling either V_{BE} or I_B, it means that we are using the BJT as a switch. When we use it as a switch, the stability of the collector current is not important.

Forward Active Mode

If we design the circuit so that V_{BE} is greater than the threshold voltage and the V_C is somewhat higher than the V_B, it means that we put our BJT in to the forward active mode. In this mode, the relation between base current (I_B) and the collector current (I_C) can be described using Equation 1.

$$I_C = \beta I_B \tag{1}$$

When we want a controlled current source or an amplifier, we put our BJT in the forward active mode.

What does biasing mean to BJT circuit?

Biasing is a way we use to choose the operation mode of the BJT. When we design the biasing circuit, it means we are choosing the voltages across the BJT terminals (V_C, V_B, and V_E), and the current flow through the BJT terminals (I_C, I_B, I_E).

Biasing without DC feedback

Consider the fixed bias circuit shown in Figure 3. Applying KVL around B-E loop yields

$$I_C = \beta \frac{V_{BB} - V_{BE}}{R_{BB}} \tag{2}$$

The collector current (I_C) in equation (2) largely depends on the BJT current gain (β). If the β varies, then the I_C will also vary.

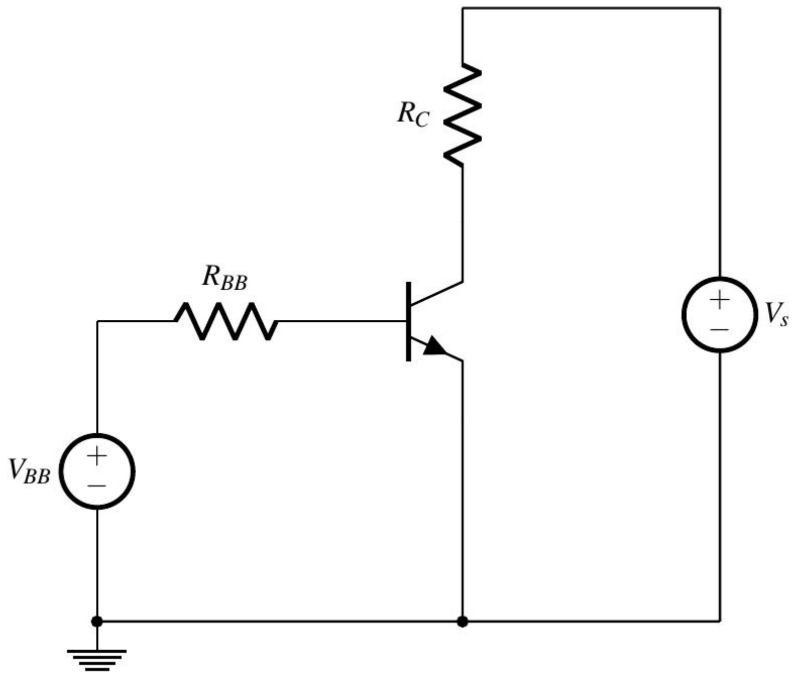


Figure 3. A fixed bias without a DC feedback.

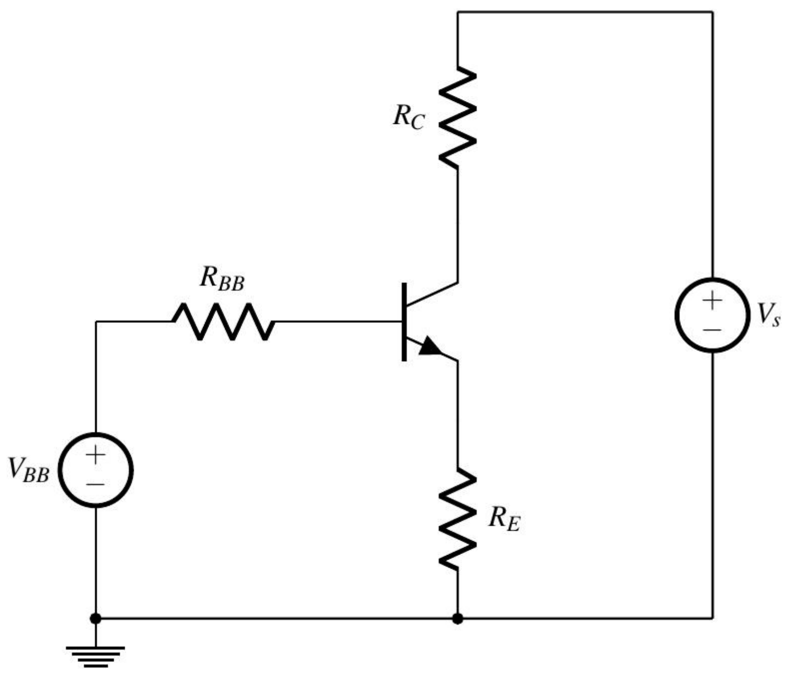


Figure 4. A fixed bias with a DC feedback.

Biasing with DC feedback

Let us insert a resistor namely R_E between the emitter and ground as shown in Figure 4. Applying KVL around B-E loop yields

$$I_C = \beta \frac{V_{BB} - V_{BE}}{R_{BB} + \beta R_E} \tag{3}$$

If βR_E is far greater than R_C, then the equation (3) can be approximated to

$$I_C \cong \frac{V_{BB} - V_{TH}}{R_E} \tag{4}$$

With a DC feedback resistor, the influence of β on I_C is reduced. And under a certain condition, the I_C does not depend on β anymore. This means that we can control the I_C regardless how unreliable the β is.

PRELIMINARY REPORT

Derive Equation 2 and 3 using Equation 1 and the principle of KVL.

APPARATUS

1. Two variable DC power supplies
2. A micro-ammeter
3. A milli-ammeter
4. A multimeter

MATERIALS

R	680 Ω 5% ¼ W	x	1
	3.3 kΩ 5% ¼ W	x	1
	560 kΩ5% ¼ W	x	1
Q	2N2222A	x	1

PROCEDURE (P1 – P4 and R1 – R3; 40 points)

PART A: V-I Characteristic of a BJT

The idea of the experiment is to vary I_B to several values. For each value of I_B, vary V_{CE} and observe I_C. The relation between V_{CE} and I_C is called V-I curve or V-I characteristic of a BJT.

P1 Build a circuit follow the schematic in Figure 5. Please be careful to connect the pin correctly according to the pinout shown in Figure 6.

a. V_s and V_{BB} are variable DC power supply.
b. The ammeter used to measure I_B is the micro-ammeter A_1.
c. The ammeter used to measure I_C is the milli-ammeter A_2. Be careful of the polarity of both ammeters.
d. Set V_s and V_{BB} to 0 volt. At this point, it is expected that I_B shall be 0 µA and I_C shall be 0 mA.

P2 Vary V_s so that V_{CE} is varied from 0 V to 5 V. The step size of the V_s is 0.1 V when $0.0 \text{ V} \leq V_s \leq 1.0 \text{ V}$, and is 0.5 V otherwise. For each V_{CE}, measure I_C. Record values of I_B, I_C and V_{CE}. Repeat this with different values of V_{BB} so that I_B is set to 20, and 40 µA, respectively.

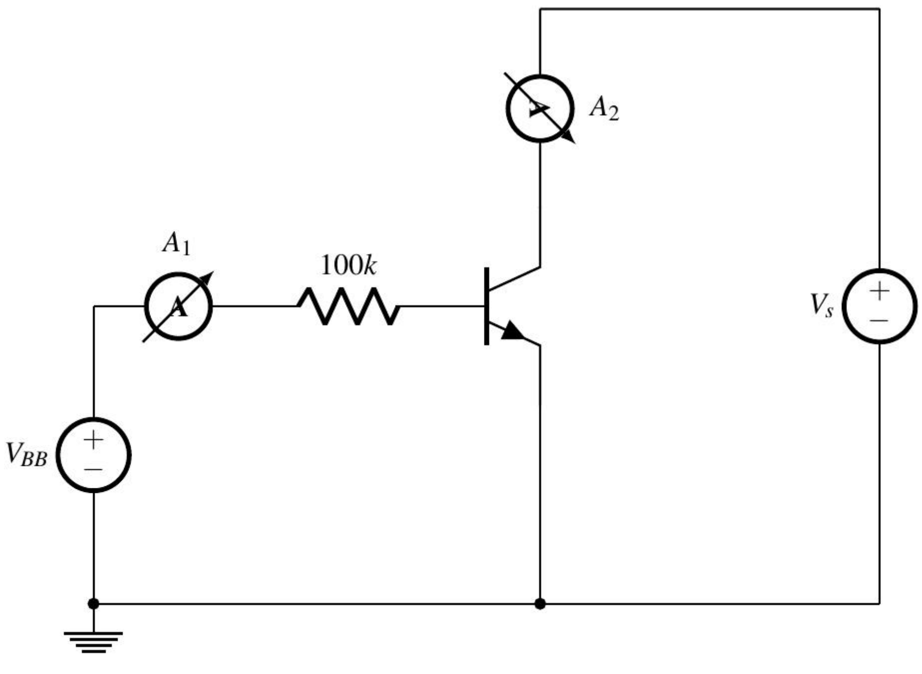
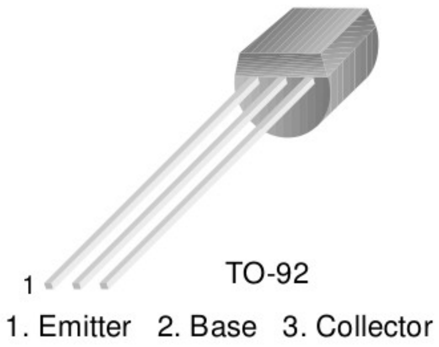
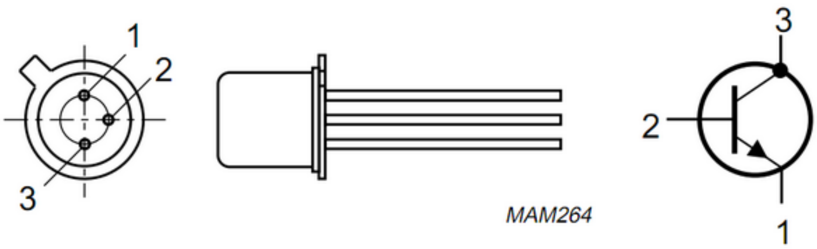


Figure 5. Circuit for V-I characteristic of a BJT.



a) TO-92 Case style



b) TO-18 Case Style

Figure 6. Pin out of 2N2222.

R1 (20 points) Fill in Table 1 and Table 2 below with I_B, I_C and V_{CE} obtained from P2. Plot graphs, one for each I_B, using V_{CE} as X axis and I_C as Y axis. All graphs shall be on the same plot (same axes).

Table 1. I_C and V_{CE} at different I_B (0.0 V ≤ V_{CE} ≤ 1.0 V)

I_B											
V_{CE} (V)	0.0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1.0
I_C											
I_B											
V_{CE}											
I_C											
I_B											
V_{CE}											
I_C											

Table 2. I_C and V_{CE} at different I_B (1.0 V ≤ V_{CE} ≤ 5.0 V)

I_B					
V_{CE} (V)	1.0	2.0	3.0	4.0	5.0
I_C					
I_B					
V_{CE}	1.0	2.0	3.0	4.0	5.0
I_C					
I_B					
V_{CE}	1.0	2.0	3.0	4.0	5.0
I_C					

PART B: Fixed bias

A circuit supplying current for the BJT is called a biasing circuit. The simplest biasing technique is the fixed bias.

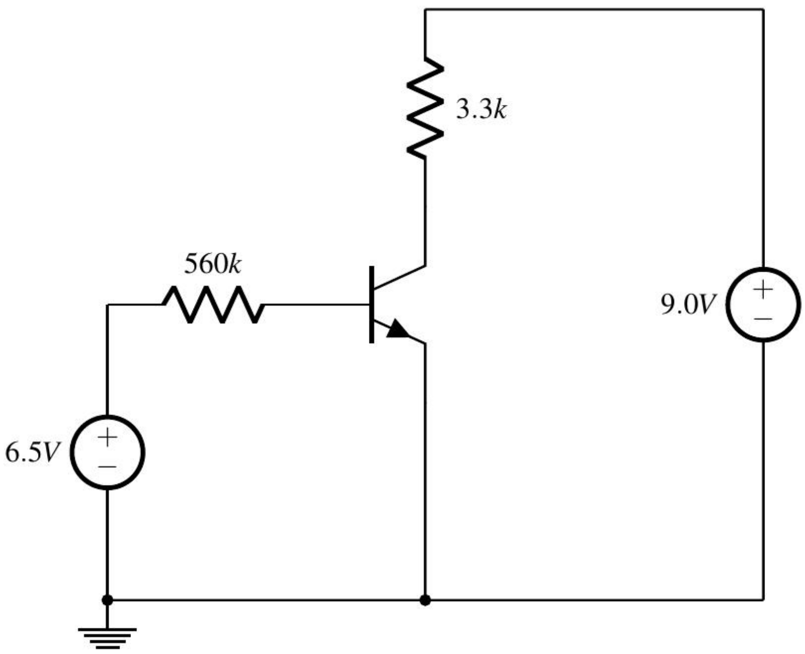


Figure 7 A fixed biasing circuit without DC feedback

P3 Construct a fixed biased BJT circuit as shown in Figure 7. Determine values of I_B and I_C from voltages across the resistor R_B and R_C, respectively. The values of resistors used in the calculation should be taken from the direct measurement outside the circuit. Calculate the current gain.

R2 (10 points) Put I_B, I_C, and β obtained from your group and from the other groups into Table 3.

Table 3. Collector currents and current gains among different groups

G	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
I_B															
I_C															
β															

Students may notice that the collector current (I_C) in table 3 varies among groups. The next part will show that this collector current variation current can be reduced by introducing a DC feedback resistor into the circuit.

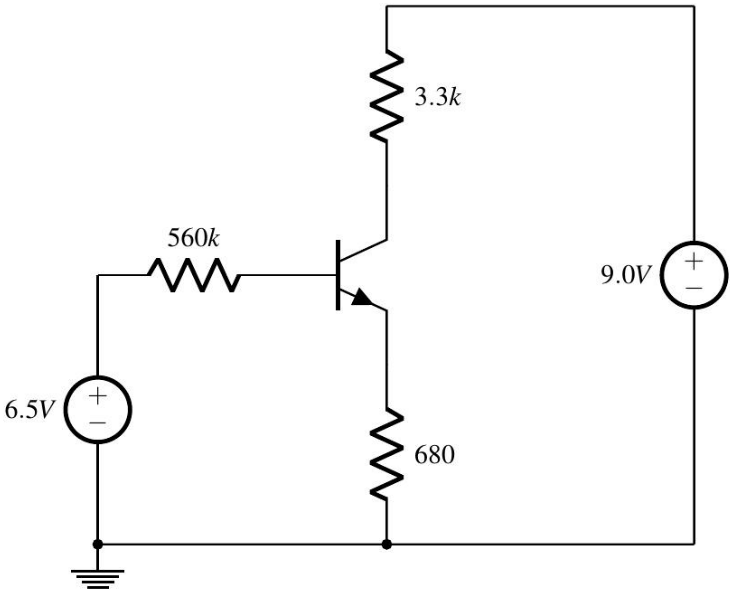


Figure 8. Voltage divider biasing circuit with DC feedback.

P4 Construct a circuit as shown in Figure 8. Repeat everything in P3.

R3 (10 points) Record data from your group and from the other groups into Table 4.

Table 4. Collector currents and current gains among different groups

G	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
I_B															
I_C															
β															

It is expected that the I_C among groups shall be more or less the same.

QUESTIONS (Q1 – Q7; 20 points)

Q1 (3 points) Why do we set different voltage step sizes in P2?

Q2 (3 points) Please indicate the saturation region, the cutoff region, and the forward active region in your graph.

Q3 (3 points) What is the application of BJT circuit without DC feedback?

Q4 (3 points) Why do the collector currents among different groups differ?

Q5 (2 points) Do the current gain among different groups in Table 4 differ?

Q6 (3 points) Why are the collector current among different groups more or less the same?

Q7 (3 points) Why do we need to take the resistor out of the circuit before we measure its resistance?

Kittipitch Meesawat
January 2018

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-04: COMMON EMITTER AMPLIFIERS

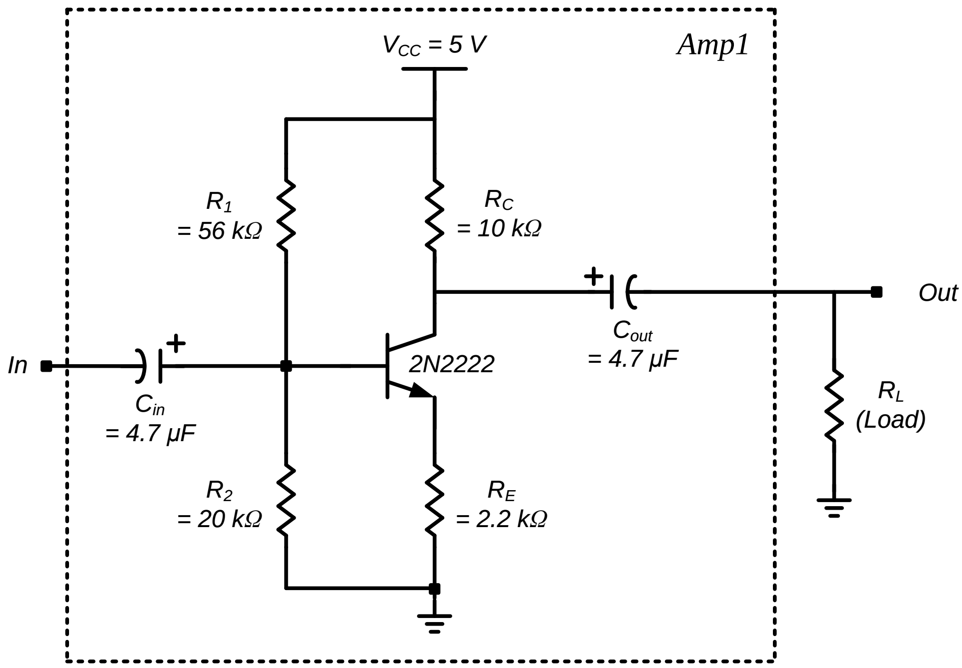
OBJECTIVES

1. To familiarize students with the single-stage BJT common-emitter (CE) amplifier and its important properties.
2. Students should be able to perform AC analysis, calculate the amplifier gain, input impedance, and output impedance by hand, and verify them in experiment.

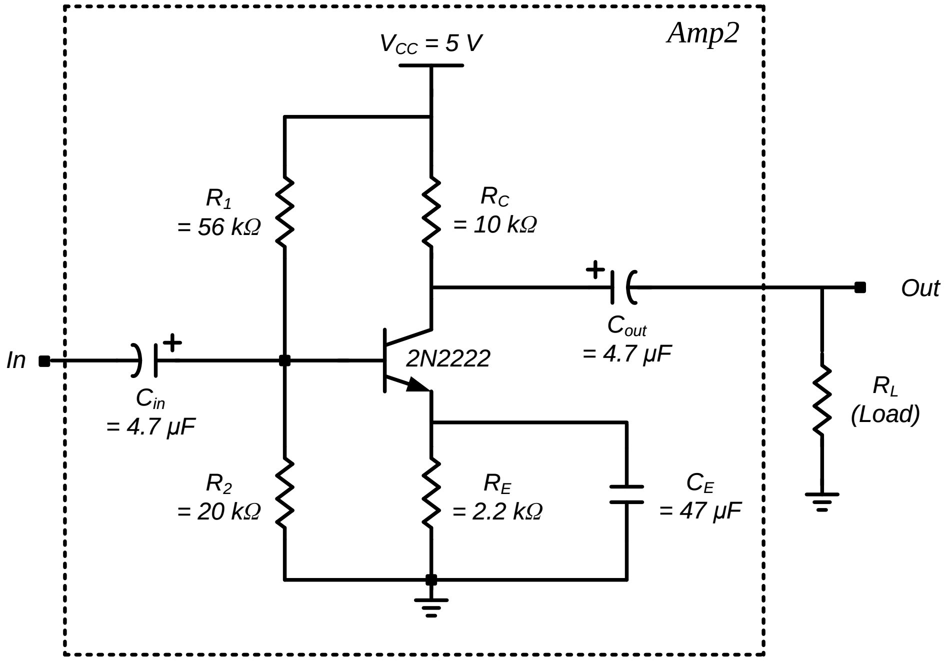
INTRODUCTION

In the previous experiment, we learned how to establish a desired DC operating condition for a BJT CE amplifier. We learned how adding a feedback resistor to the emitter of a BJT amplifier could affect its bias stability. We are now ready to feed in a sinusoidal signal to our circuit and see how well it performs as an AC-signal amplifier.

In this experiment, we will work on two amplifiers as shown in Figure 1. Similar to the one in the previous experiment, Figure 1(a) is a BJT CE amplifier with a feedback resistor R_E (also called emitter degeneration). Students should have measured DC parameters ready from the previous experiment (β and I_C) if the same circuit is used. We will use them to calculate AC parameters (r_π and g_m). Figure 1(b) is a BJT CE amplifier with a feedback resistor R_E and a bypass capacitor C_E.



(a)



(b)

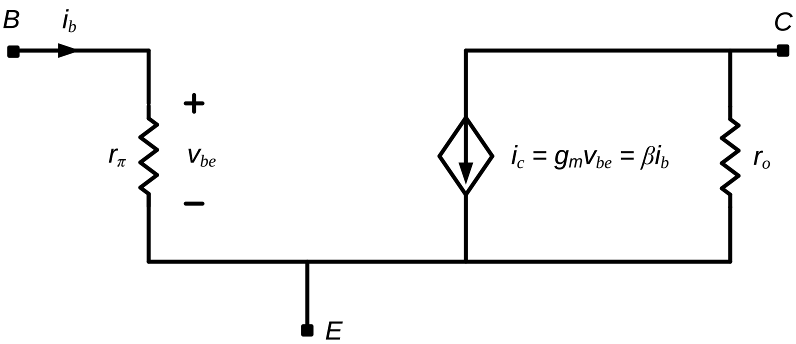
Figure 1. BJT CE Amplifiers with (a) a feedback resistor and (b) a feedback resistor and a bypass capacitor.

Note that, DC-wise, both amplifiers are the same. Their bias conditions are quite stable due to their respective feedback resistors.

THEORY

AC Analysis

A small-signal model of the NPN BJT is shown in Figure 2. It represents the BJT as a voltage-controlled current source at a given bias point. Obviously, the model parameters g_m, r_π and r_o depend on the value of the DC bias current I_C. V_T is the thermal voltage constant, which is equal to 26 mV at room temperature. We will also assume that r_o is relatively large and can be ignored from analysis in this experiment.



$$r_\pi = \frac{\beta V_T}{I_C}$$

$$g_m = \frac{I_C}{V_T}$$

$$r_o = \frac{V_A}{I_C}$$

Figure 2. A simplified hybrid-π small-signal model of the NPN BJT.

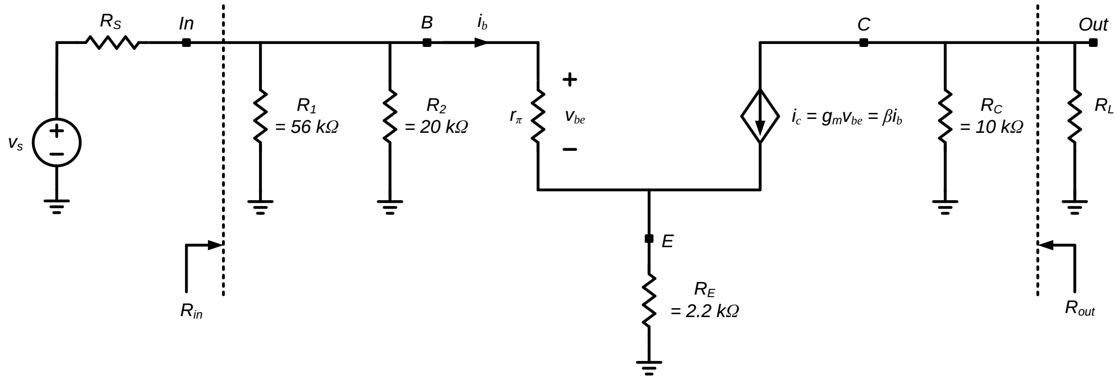


Figure 3. An AC equivalent circuit of the amplifier shown in Figure 1(a).

Based on superposition theorem, an AC equivalent circuit of the amplifier in Fig. 1(a) can be drawn as illustrated in Fig. 3. All DC voltage sources are shorted to ground and all capacitors represent short circuits. From KCL and KVL,

$$v_{out} = -\beta i_b (R_C \| R_L)$$

$$v_{in} = i_b r_\pi + i_e R_E = i_b [r_\pi + (1+\beta) R_E]$$

$$A_v = \frac{v_{out}}{v_{in}} = \frac{-\beta (R_C \| R_L)}{r_\pi + (1+\beta) R_E}$$

Typically, $\beta \gg 1$ and $\beta R_E \gg r_\pi$, so the voltage gain is approximately

$$A_v \cong \frac{-(R_C \| R_L)}{R_E}$$

The input resistance and output resistance of the amplifier are

$$R_{in} = R_1 \| R_2 \| r_\pi + (1+\beta) R_E$$

$$R_{out} = R_C$$

Notice that, for a CE amplifier with a feedback resistor, the voltage gain does not depend on β when β is large enough.

PRELIMINARY REPORT

1. Draw an AC equivalent circuit for the amplifier with a feedback resistor and a bypass capacitor shown in Figure 1(b) and derive expressions for its voltage gain (A_v), input resistance (R_{in}), and output resistance (R_{out}).

2. For both amplifiers in Figure 1(a) and 1(b), please do the followings:
   - Assume R_L is very large (R_L is an open circuit). Calculate A_v, R_{in}, and R_{out}.
   - Assume R_L is 10 kΩ. Repeat the calculations for A_v, R_{in}, and R_{out}.

APPARATUS

1. DC power supply
2. Function generator
3. Oscilloscope
4. 2N2222 transistor, resistors, and capacitors as listed in Figure 1

PROCEDURE (P1 – P3 and R1 – R4; 40 points)

P1 Construct the amplifier circuit as shown in Figure 1(a) and leave R_L as an open circuit. Be sure to use the correct polarity for the coupling capacitors (C_{in} and C_{out}). It is likely that you already have this circuit constructed from the previous experiment. Check that the correct bias voltages are established.

R1 (10 points) Apply a 0.2-Vpp 1-kHz sinusoidal signal at the input using a function generator. Record the input and output waveforms on the oscilloscope and calculate the voltage gain (A_v).

P2 To measure the output resistance (R_{out}) of the amplifier, add a 10-kΩ resistor (R_L) to ground at the output as shown in Figure 4.

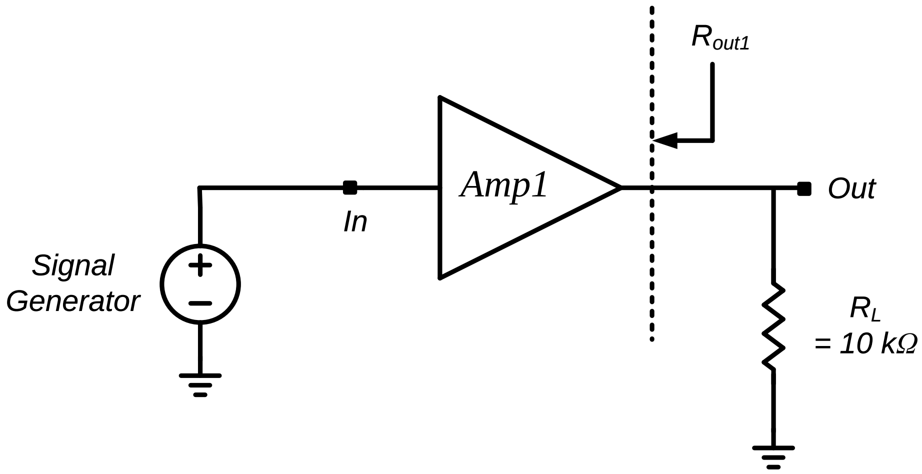


Figure 4. Output resistance measurement.

R2 (10 points) Using the same input as in the previous step, record the input and output waveforms on the oscilloscope and calculate the voltage gain. The output resistance of the amplifier can be calculated from the loading effect of R_L as

$$\frac{R_L}{R_{out}+R_L}=\frac{10k\Omega}{R_{out}+10k\Omega}=\frac{v_{out}(R_L=10k\Omega)}{v_{out}(\text{no }R_L)}$$

P3 Construct the amplifier circuit as shown in Figure 1(b) and leave R_L as an open circuit. Check that the correct bias voltages are established.

R3 **(10 points)** Apply a 0.2-Vpp 1-kHz sinusoidal signal at the input using a function generator. Record the input and output waveforms on the oscilloscope and calculate the voltage gain (A_v).

Tip: You will see that the voltage gain of the circuit in Figure 1(b) is much higher than that in Figure 1(a) and the output waveform will be badly distorted if the input amplitude is too large. To measure the voltage gain, reduce the amplitude of the sinusoidal input to remove the clipping. It is likely that the input will have to be very small (could be as small as 15-30 mVpp depending on your gain).

R4 **(10 points)** Add the 10-kΩ resistor (R_L) at the output as shown in Figure 4. Using the same input as in the previous step, record the input and output waveforms on the oscilloscope and calculate the voltage gain. Then, calculate the output resistance of the amplifier.

QUESTIONS (Q1 and Q2; 20 points)

Q1 **(10 points)** What is the role of the bypass capacitor in Figure 1(b)?

Q2 **(10 points)** Should you use the amplifiers in Fig. 1 to drive an 8-Ω speaker (R_L = 8Ω)? Would you get any gain? Clearly explain your reasoning.

Sataporn Pornpromlikit
January 2018

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-05: FREQUENCY RESPONSE OF A COMMON EMITTER AMPLIFIER

OBJECTIVES

1. To calculate the bandwidth of the amplifier circuit.
2. To measure the cut-off frequency at low frequency and high frequency.

INTRODUCTION

Lecture note on voltage gain (Av) calculation is actually a "Midband Voltage Gain" at the mid frequency band while those capacitors are considered to be short circuits. In reality, the voltage gain of an amplifier actually depends on the signal frequency; therefore, the study of frequency response of circuit is essential in order to realize the bandwidth of the circuit.

THEORY

Logarithms and Decibels

Power (Watts): $$G_{dB} = 10\log_{10}\frac{P2}{P1}$$

Voltage (Volts): $$G_{dB} = 20\log_{10}\frac{V_2}{V_1}$$

f_{3dB}: At the half-power frequencies, the resulting level is 0.707= $1/\sqrt{2}$

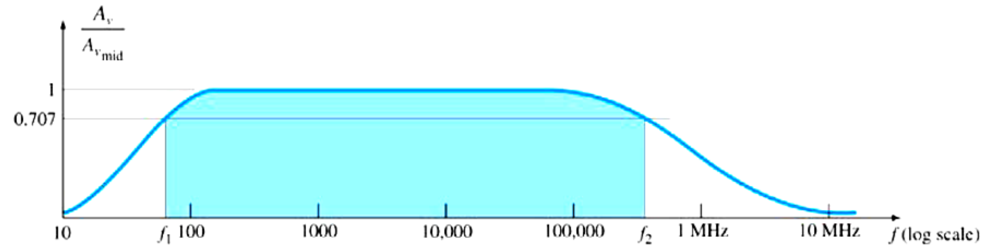


Figure 1(a). Normalized gain versus frequency plot.

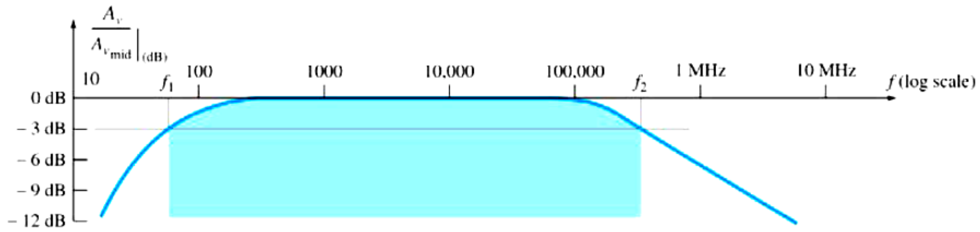


Figure 1(b). Decibel plot of normalized gain versus frequency plot of Figure 1(a).

Common-Emitter Amplifier Configuration

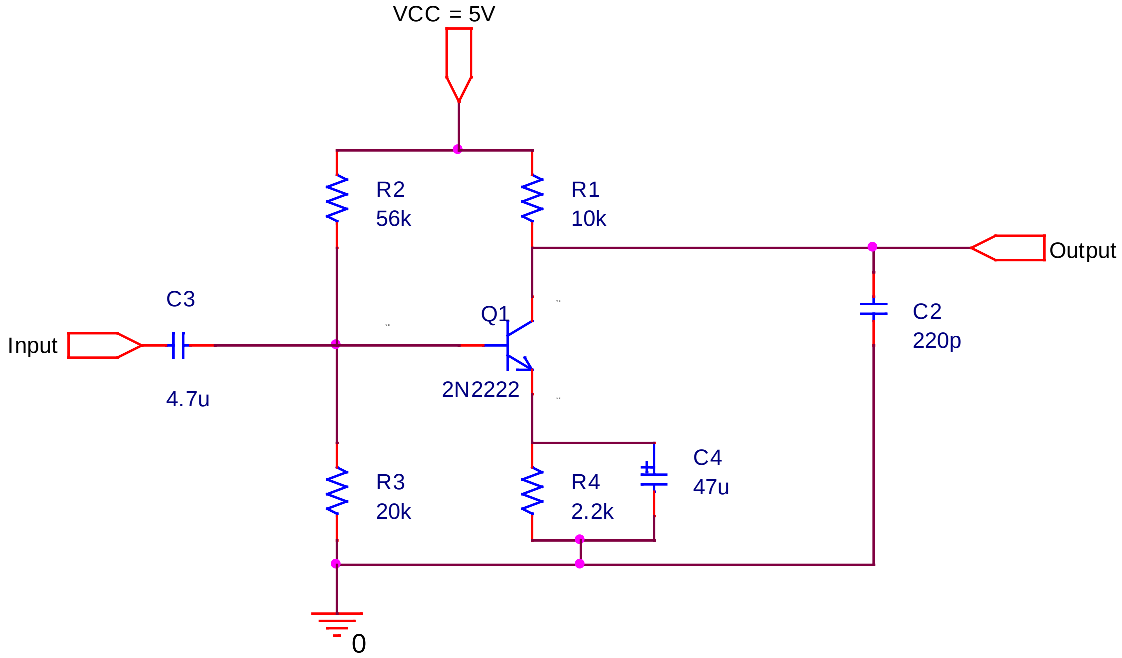


Figure 2. Common-Emitter Amplifier Configuration.

Low Frequency Response (C_E Effect):

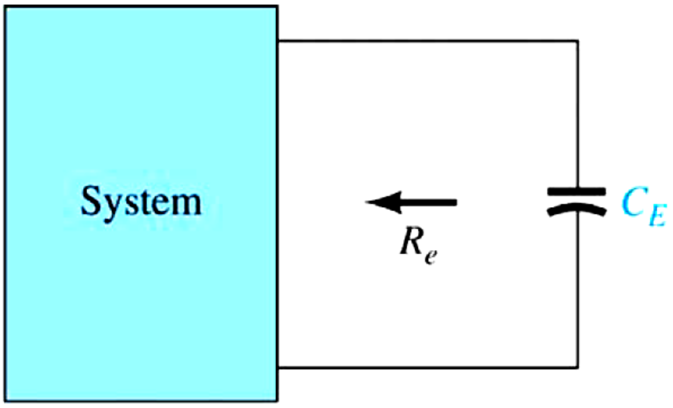


Figure 3. Low Frequency Response (C_E Effect).

$$f_{LE} = \frac{1}{2\pi . re . C_E}$$

where
$$re = 1/gm$$
$$gm = I_C / V_T$$
$$V_T = kT / q \approx 26mV$$

High Frequency Response (C_L Effect):

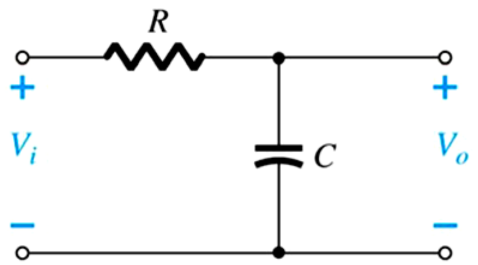


Figure 4. High Frequency Response (C_L Effect)

$$f_{HC} = \frac{1}{2\pi . R_{out} . C_L}$$

Where $\begin{aligned} R_{out} &\approx R_C \\ C_L &= 220\,pF \end{aligned}$

PRELIMINARY REPORT

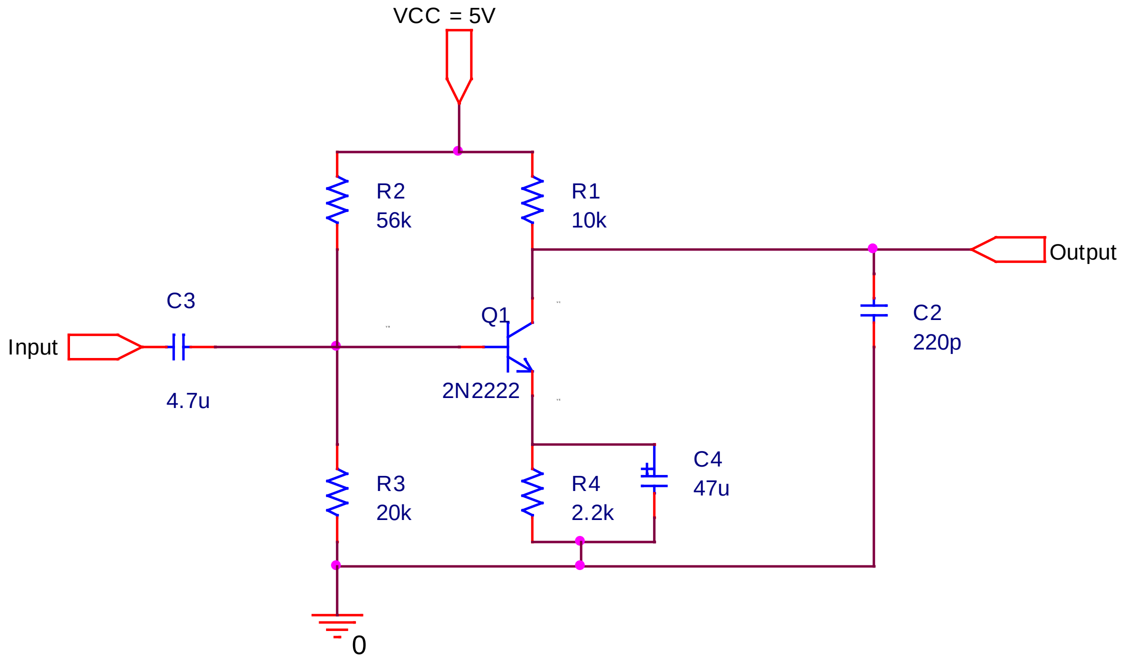


Figure 5. Common Emitter Amplifier.

1. Calculate the biasing point of the common emitter amplifier circuit and its voltage gain with no-load condition (Do not taking the capacitive load of 220 pF into your calculation).
2. Calculate the higher and lower frequencies of the given circuit.
3. Calculate the bandwidth of the given circuit.

APPARATUS

DC Power Supply
Function generator
Oscilloscope
Devices as listed in Figure 5

PROCEDURE (P1 – P4 and R1 – R5; 40 points)

Without INPUT Signal

P1 Connect the circuit as shown in Figure 5.

R1 (10 points) Measure V_{CC}, V_C, V_E, V_B using the voltmeter.

R2 (5 points) Calculate the collector current (I_C), Transconductance (g_m) and voltage gain (A_V).

Parameters	**Formula**	**Calculation**
I_C (A)	$[V_{CC}-V_C]/R_C$	
g_m (S)	I_C/V_T	
r_e (Ω)	$1/g_m$	
A_V	$-g_mR_C$	

With INPUT Signal

P2 Connect 1 kHz sinusoidal signal to the input port. Adjust the input signal until the output signal reaches $2V_{p\text{-}p}$. If the output signal is clipped, reduce the input until the clipping is removed and the output waveform looks sinusoidal.

R2 (10 points) Use the oscilloscope to display the input signal and the output signal. Sketch the input and output waveforms. Measure the actual voltage gain Av using the oscilloscope and fill in the table below.

Voltage Gain	**Calculation**	**Measurement**
A_V	$-g_mR_C$	V_{out} (p-p) / V_{in} (p-p)

Cut-off Frequency

P3 To measure the higher cut-off frequency (f_H), increase the frequency of the input signal until the output signal reaches 70.7% of the mid-band frequency value.

R3 (5 points) Record the frequency value at the higher cut-off frequency.

P4 To measure the lower cut-off frequency (f_L), reduce the frequency of the input signal until the output signal reaches 70.7% of the mid-band frequency value.

R4 (5 points) Record the frequency value at the lower cut-off frequency.

R5 (5 points) Fill in the table below.

<table>
<tr><th colspan="2">Calculation
$f_{3dB} = 1/[2\pi.R.C]$ (Hz)</th><th>Measurement
(Hz)</th></tr>
<tr><td>$f_H = 1/[2\pi.(10k\Omega).(220pF)]$</td><td></td><td></td></tr>
<tr><td>$f_L = 1/[2\pi.(r_e).(47uF)]$</td><td></td><td></td></tr>
<tr><td>Bandwidth (f_H-f_L)</td><td></td><td></td></tr>
</table>

QUESTIONS (Q1 – Q2; 20 points)

Q1 (10 points) Compare the results based on (a) Theory only/Preliminary (b) Measurements. Then, discuss the causes of the difference between the two.

Q2 (10 points) Summarize the role of capacitor into the frequency response of the amplifier.

Chiranut Sa-ngiamsak
October 2013

KHON KAEN UNIVERSITY
DEPARTMENT OF ELECTRICAL ENGINEERING

EXPERIMENT EE2-06: OP-AMP CIRCUITS AND APPLICATIONS

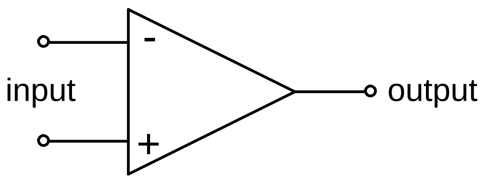
OBJECTIVES

1. To enhance the knowledge of operational-amplifier circuits.
2. To measure some characteristics of op-amp circuits.
3. To construct a Wien bridge oscillator using operational amplifier.

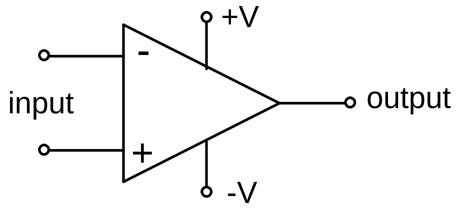
INTRODUCTION

The operational amplifier is an analog device which was developed in the 1940's for use in analog computers. This fundamental electronic building block was called an operational amplifier because originally it was intended to perform mathematical operations such as addition and subtraction. In a sense the op-amp was the first analog computer. Once hard-wired, it could perform a single computation. It could not be programmed for any other computation except by rewiring the circuit.

The standard symbol for an op-amp is shown in the figure to the left below.



(a) Standard op-amp symbol.



(b) Op-amp symbol with bipolar supply.

Figure 1. Op-amp symbols.

Bipolar Supply (Dual Supply)

The Op-Amp has a single output terminal plus two input terminals, one of which is inverted (-) and one which is not. The symbol on the right is also used. The two power terminals labeled ±V highlight the fact that the op-amp is an active device requiring an independent power source in addition to the two inputs. Notice that the op-amp requires two DC supply voltages, one at +V and one at -V. Both voltages can be delivered by using a bipolar power supply (Dual power supply) or by using a voltage divider. This use of bipolar supply voltages is to allow the amplified output voltage to swing in both positive and negative directions (as would be required in the amplification of AC signals.)

Standard IC Packages

The original op-amps used vacuum tubes, dangerously high voltages and were very expensive. Today, high quality op-amps are available as inexpensive integrated circuit packages which operate on moderate to small voltages. The 741 op-amp is available in both 8 pin and 14 pin Dual In-Line packages as well as 8 pin tin cans. The extra pins on the 14 pin chips are used for heat sinking, and so can provide greater power.

Pin Connections

In this lab we will be using an 8-pin 741 op-amp as shown below. The pins are labeled from 1 to 8 with pin 1 to the left of the small notch as shown below.

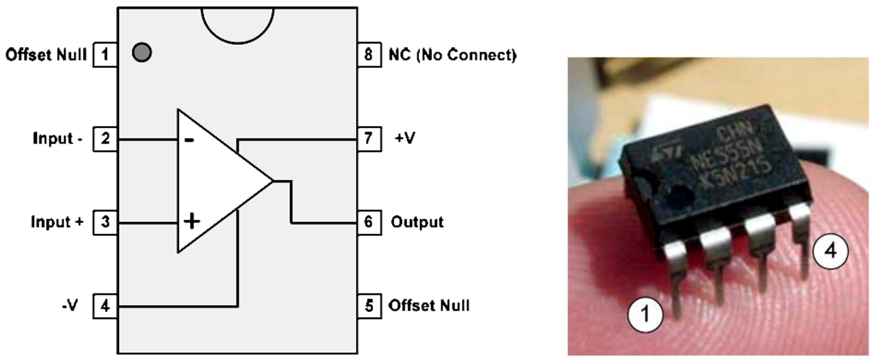


Figure 2. The 741 pin layout and IC package.

Notice that the two inputs are connected to pins 2 and 3. Pin 2 is the inverting input and pin 3 is the non-inverting input. The output is at pin 6. The bipolar power leads are connected to pins 4 and 7 as indicated. Pins 1 and 5 can be used to offset any null voltage which can arise due to variations between the transistors contained within the op-amp chip. Pin 8 is not connected (NC) to the op-amp electrically, but can be used as a heat sink.

Absolute Maximum Ratings

- The 741 chips used in this lab have approximately the following absolute maximum ratings. Do not exceed!
- Power Supply must not exceed ±18 Volts and not under ±5 Volts
- Power Dissipation must not exceed 500 mW.
- Power Consumption is a maximum of 85 mW.
- Differential input voltage must not exceed ±30 volts.

THEORY

The Ideal Op-Amp

When analyzing op-amp circuits, one usually begins by treating it as an ideal op-amp.

1. The ideal op-amp has infinite input impedance (open circuit) and so does not draw any power or current from the driving source.
2. The ideal op-amp has zero output impedance.
3. The ideal op-amp has infinite voltage gain.
4. The ideal op-amp has infinite bandwidth.

In the real world, op-amps have limitations. The practical op-amp has high (but not infinite) input impedance, low output impedance, high voltage gain and wide bandwidth. The output voltage can never exceed the bipolar supply voltages ±V. The maximum positive and negative output voltages are called the saturation limits (Maximum output swing) and tend to be very near ±V. A simplified graph of output voltage against differential input voltage is shown below. The straight portion between the saturation limits is called the linear region and its slope gives the gain of the amplifier.

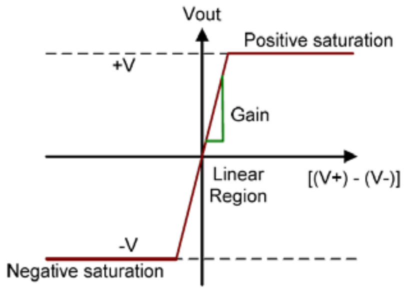


Figure 3. Maximum output swing and linear region.

Op-Amp Circuits

Below, we will look at a few of the many circuits in which op-amps have found applications. Three of the more important circuits include:

1. Inverting Amplifier
2. Non-inverting Amplifier
3. Wien bridge oscillator

1. Inverting Amplifier

The circuit below shows a simple inverting amplifier made using an Op-Amp.

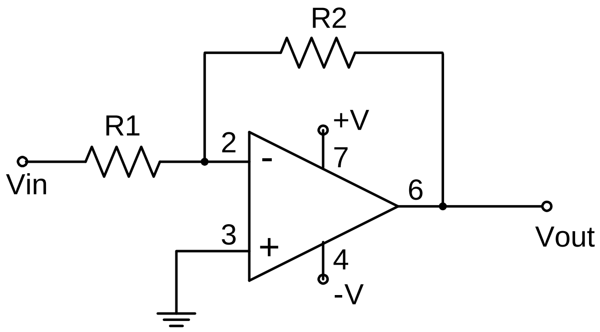


Figure 4. A simple inverting amplifier circuit.

The input voltage V_{in} is connected through a resistor R1 to the inverting input pin 2(-) on the op-amp. The non-inverting input pin 3 (+) is connected to ground. The gain A_V of an amplifier is defined as the ratio of the output to input voltages.

$$A_V = V_{out} / V_{in}$$

For the above circuit, we can assume that the same current flows through both resistors because of the extremely high input impedance and that the voltage at the inverting pin (2) is nearly the same as ground (virtual short). Thus

$$V_{out} = -(R2 / R1)\, V_{in}$$

so that the gain is:

$$A_V = -(R2 / R1)$$

Example: If R1 = 1kΩ and R2 = 10kΩ, then the gain is - 10. An input signal of 30 mV will be inverted and multiplied to result in an output signal of V_{out} = -300 mV.

2. Non-inverting Amplifier

The circuit below shows a simple design for a non-inverting amplifier. The input voltage Vin is applied directly to the non-inverting terminal of the op-amp. Negative feedback is provided by the two external resistors R1 and R2 which form a voltage divider and apply a fixed fraction of the output voltage to the inverting input terminal of the op-amp.

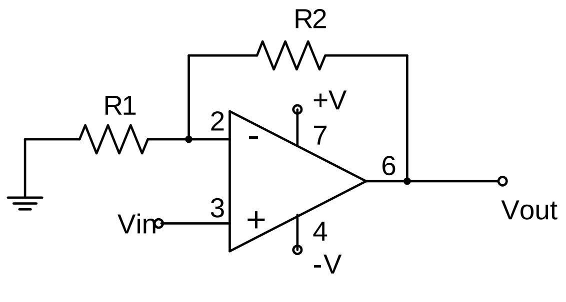


Figure 5. A simple non-inverting amplifier circuit.

Below, we will show that the gain of the amplifier is determined by the external resistors R1 and R2 according to the equation:

$$A_V = 1+(R2 / R1)$$

Example: If R2 = 10kΩ and R1 = 1kΩ, the gain is 11 so that an input voltage of V_{in} = 30 mV is multiplied to an output voltage of V_{out} = 330 mV. (The output is not-inverted for the circuit shown above.)

Derivation of the Gain for a Non-Inverting Amplifier

The negative feedback between the output terminal and the inverting input terminal is responsible for the non-inverting amplifier's properties. The gain of this amplifier is most easily calculated using the ideal op-amp assumptions. First note that because of the extremely high input impedance (resistance), effectively no current flows into either input terminal. Thus a common current ***i*** flows from the output terminal through the resistors and into the ground connection.

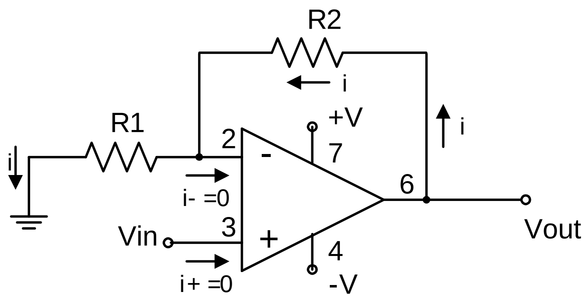


Figure 6. A simple inverting amplifier circuit with current flow.

If a positive voltage V_{in} is applied at the input, then the output voltage V_{out} will begin to increase. However, a certain fraction of the output is fed back to the inverting input via the voltage divider, causing the differential voltage input (Δv) to decrease. In equilibrium, the differential voltage input is approximately zero, since otherwise the op-amp would saturate. Thus the voltages at the two inputs are equal. This condition leads to the equation:

$$V_{in} = V_{out} [R1/(R1+R2)]$$

Since the gain is defined as the ratio V_{out}/V_{in}, we will see that $A_V = 1+(R2/R1)$.

The Virtual Short

In the above analysis, we saw that the two input terminals are forced to the same voltage even though no current flows through either terminal. This situation is called a virtual short circuit and occurs because the op-amp output (via the feedback connection) forces the inverting terminal voltage to precisely track the voltage supplied to the non-inverting terminal. The situation is analogous to two magnets on either side of a window. If one magnet is raised the other magnet will precisely track it even though there is no physical connection between them. The concept of the virtual short is extremely useful in analyzing operational amplifier circuits.

Slew Rate

An ideal op-amp has an infinite frequency response. This means that no matter how fast the input changes, the output will be able to keep up. In a real op-amp, this is not the case. If the input signal changes too fast then the output will not be able to keep up. This is defined as slewing and it results in distortion of the output waveform. Stated more formally,

$$\text{Slew Rate} = SR = \text{maximum } dv_o/dt$$

or the maximum rate at which the output can change without distorting. This can be measured by applying a high frequency square wave signal. The frequency of the waveform should be increase until the waveform becomes a triangular wave. The slope of the triangular waveform is the slew rate. (SR = $\Delta V / \Delta T$)

The Slew Rate of 741 is 0.5 Volts/μsec

3. Wien bridge oscillator

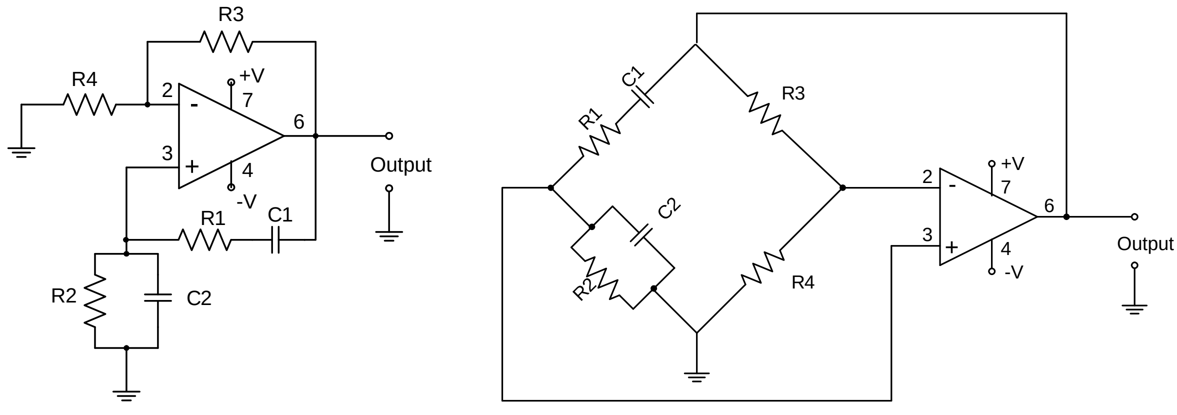


Figure 7. Wien bridge oscillator circuits

Frequency of oscillation

$$f = \frac{1}{2\pi RC} \quad Hz$$

Where R1 = R2 = R = Resistances (Ω)
C1 = C2 = C = Capacitances (F)

And R3/R4 $\geq$ 2

Figure 7 shows the Wien bridge oscillator circuits. A resistor R4 is connected to the inverting terminal (2) of the operational amplifier from the ground. Similarly, a parallel combination of a resistance R2 and a capacitor C2 is connected to the non-inverting terminal (3) of the operational amplifier from the ground. The output terminal (6) of the amplifier is fed back to inverting terminal (2) through a resistor R3. A series combination of a resistance R1 and a capacitor C1 is connected between non-inverting terminal (3) and the output of operational amplifier. To observe the output wave form, the output terminal (6) is connected to oscilloscope. The terminals (7) and (4) of the op-amp. are connected to +12 V and -12 V of the DC power supplies separately.

An oscillator consists of an amplifier and a feedback network.

1) 'Active device' i.e. Op Amp is used as an amplifier.
2) Passive components such as R-C or L-C combinations are used as feed back net work.To start the oscillation with the constant amplitude, positive feedback is not the only sufficient condition. Oscillator circuit must satisfy the following two conditions known as **Barkhausen** conditions:
   i. The first condition is that the magnitude of the loop gain ($A\beta$) = 1
      A = Amplifier gain and β = Feedback gain.
   ii. The second condition is that the phase shift around the loop must be 360° or 0°. The feedback signal does not produce any phase shift. This is the “basic principle of a Wien bridge oscillator”.

Lead-Lag circuit:

The given circuit shows the RC combination used in Wien bridge oscillator. This circuit is also known as lead-lag circuit. Here, resistor R1 and capacitor C1 are connected in the series while resistor R2and capacitor C2 are connected in parallel.

Working of lead-lag circuit:-

At high frequencies, the reactance of capacitor C1 and C2 approaches zero. This causes C1 and C2 appears short. Here, capacitor C2 shorts the resistor R2. Hence, the output voltage Vo will be zero since output is taken across R2 and C2 combination. So, at high frequencies, circuit acts as a “lag circuit.”

At low frequencies, both capacitors act as open because capacitor offers very high reactance. Again output voltage will be zero because the input signal is dropped across the R1 and C1combination. Here, the circuit acts like a “lead circuit”.

But at one particular frequency between the two extremes, the output voltage reaches to the maximum value. At this frequency only, resistance value becomes equal to capacitive reactance and gives maximum output. Hence, this particular frequency is known as resonant frequency or oscillating frequency.

The maximum output would be produced if

$$R = X_C = \frac{1}{2\pi f C}$$

If R1 = R2 = R and C1 = C2 = C

and R3/R4 $\geq$ 2

Then the resonant frequency

$$f = \frac{1}{2\pi RC} \quad Hz$$

PRELIMINARY REPORT

1. Write down the specifications for the 741 (maximum values, input/output impedance, input bias/offset, and slew rate)
2. Write down the inverting amplifier circuit which gain = -10
3. Write down the non-inverting amplifier circuit which gain = 11
4. Find out the oscillating frequency of Wien bridge oscillator in circuit Figure 10.

APPARATUS

Oscilloscope
DC power supply
Function generator
Protoboard

PROCEDURE (P1 – P4 and R1 – R4; 40 points)

P1 The pin connections for the 8 pin DIP package LM741 op-amp are provided in Figure 2.

To use the Power Supply Unit:

- Turn the Power Supply ON. Adjust the voltage of the Power Supply to 12V. This will set both positive and negative power sources respectively to +12V and -12V.
- Turn the Power Supply OFF before connecting to the circuits.
- Connect the +12V Power Supply to the +V of your circuit.
  Connect the - 12V Power Supply to the -V of your circuit.
  Connect the GND of the Power Supply to the ground of your circuit.

1. Non-inverting amplifier measurements

P2 Construct the circuit as shown in Figure 8 using an 8-pin LM 741 op-amp with R1 = 10kΩ and R2 = 100kΩ. Apply a 1 kHz sinusoidal voltage signal from the function generator to the input and use the oscilloscope to observe both input and output waveforms. Adjust the magnitude of the input signal until clipping occurs on either the positive or negative peak of the output voltage.

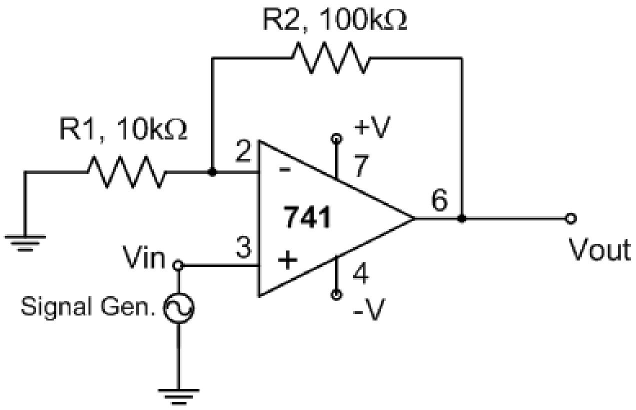


Figure 8. Measurement circuit for non-inverting amplifier.

R1 (5 points) Determine the maximum possible ac voltage swing, i.e. maximum peak to peak voltage that can be obtained at the output of the circuit without clipping. Compare this to the DC power supply voltages.

R2 (10 points) For V_{in}(p-p) =1 V at f =1 kHz, measure the amplitude of the output signal V_{out} and calculate the voltage gain of this circuit. Record the data and compare the values with preliminary calculations. Sketch the input and output waveforms of the circuit.

2. Inverting amplifier measurements

P3 Construct the circuit as shown in Figure 9 using an 8-pin LM 741 op-amp with R1 = 10 kΩ and R2 = 100 kΩ. Apply a 1 kHz sinusoidal voltage signal from the function generator to the input and use the oscilloscope to observe the shape and to measure the amplitude of the input and output waveforms.

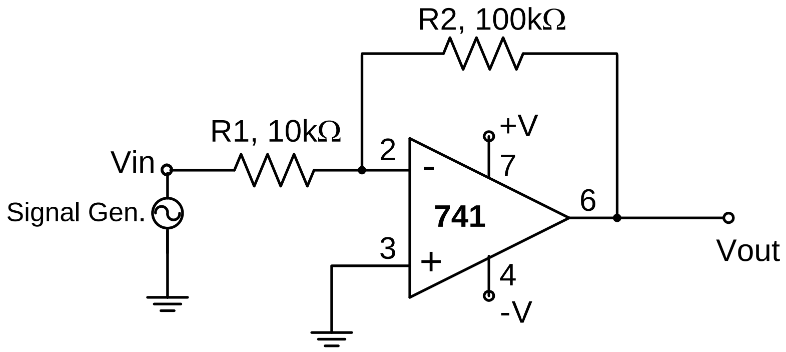


Figure 9. Measurement circuit for inverting amplifier

R3 (10 points) For V_{in}(p-p) = 1 V at f = 1 kHz measure the amplitude of the output signal (V_{out}) and calculate voltage gain of this circuit. Record data and compare the values with preliminary calculations. Sketch the input and output waveforms of the circuit.

3. Wien bridge oscillator

P4 Construct the circuit as shown in Figure 10 using an 8-pin LM 741 op-amp with R3 = 20 kΩ, R4 = 10 kΩ, R1 = R2 = R = 10 kΩ, and C1 = C2 = C = 1 μF.

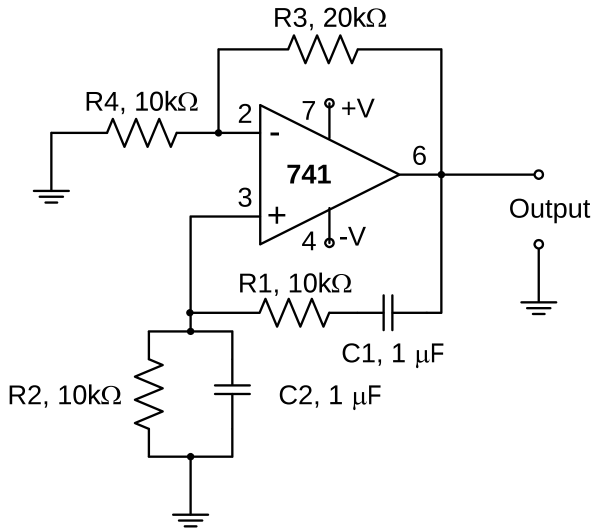


Figure 10. Measurement circuit for Wien bridge oscillator.

R4 (15 points) Use the oscilloscope to observe the shape and to measure the amplitude and the time period (T) of the generated sine wave. Fill in the table below with the results obtained from calculation and measurement.

Theoretical frequency	Measurement of output frequency	
$f=1/2\pi RC$ (Hz)	Period (T) (s)	Frequency (Hz)

QUESTIONS (Q1 – Q3; 20 points)

Consider a non-inverting amplifier with R1 = 1 kΩ and R2 = 9 kΩ, and is powered by a ±10 V bipolar supply.

Q1 (10 points) Find the gain factor in the linear region.

Q2 (5 points) What are the ideal saturation voltages?

Q3 (5 points) Find the maximum positive and negative input voltages which can be provided before the output saturates.

Wichai Premchaisavad
October 2013