Amplifying action can be provided by electromechanical devices (e.g., transformers and generators) and vacuum tubes, but most electronic systems now employ solid-state microcircuits as amplifiers. Such an integrated circuit consists of many thousands of transistors and related devices on a single tiny silicon chip. Figure 1.4 shows various types of transistors and Figure 1.5 shows an integrated circuit amplifier.

Figure 1.5 Transistor packages

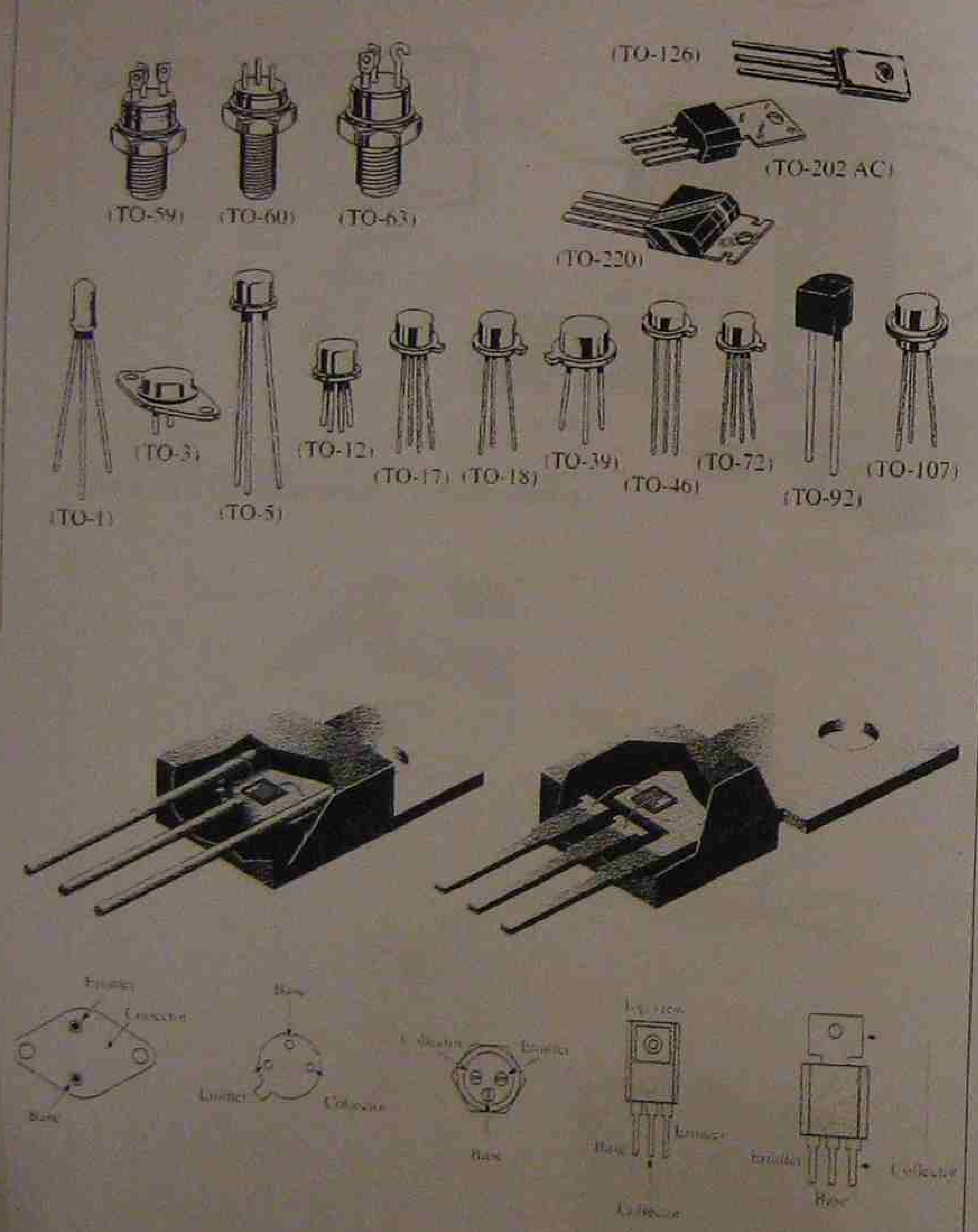


Figure 1.6 An IC amplifier (operational amplifier)

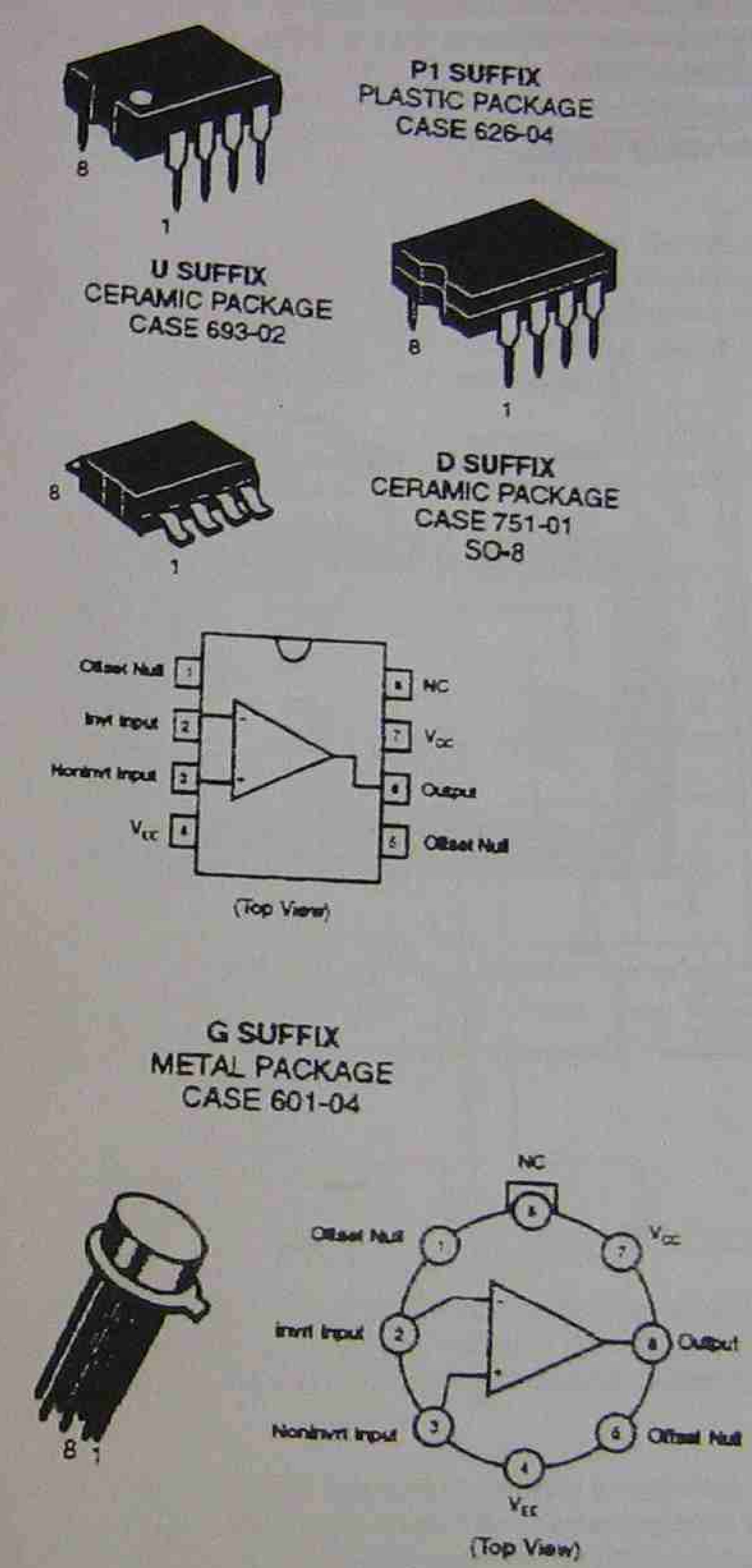
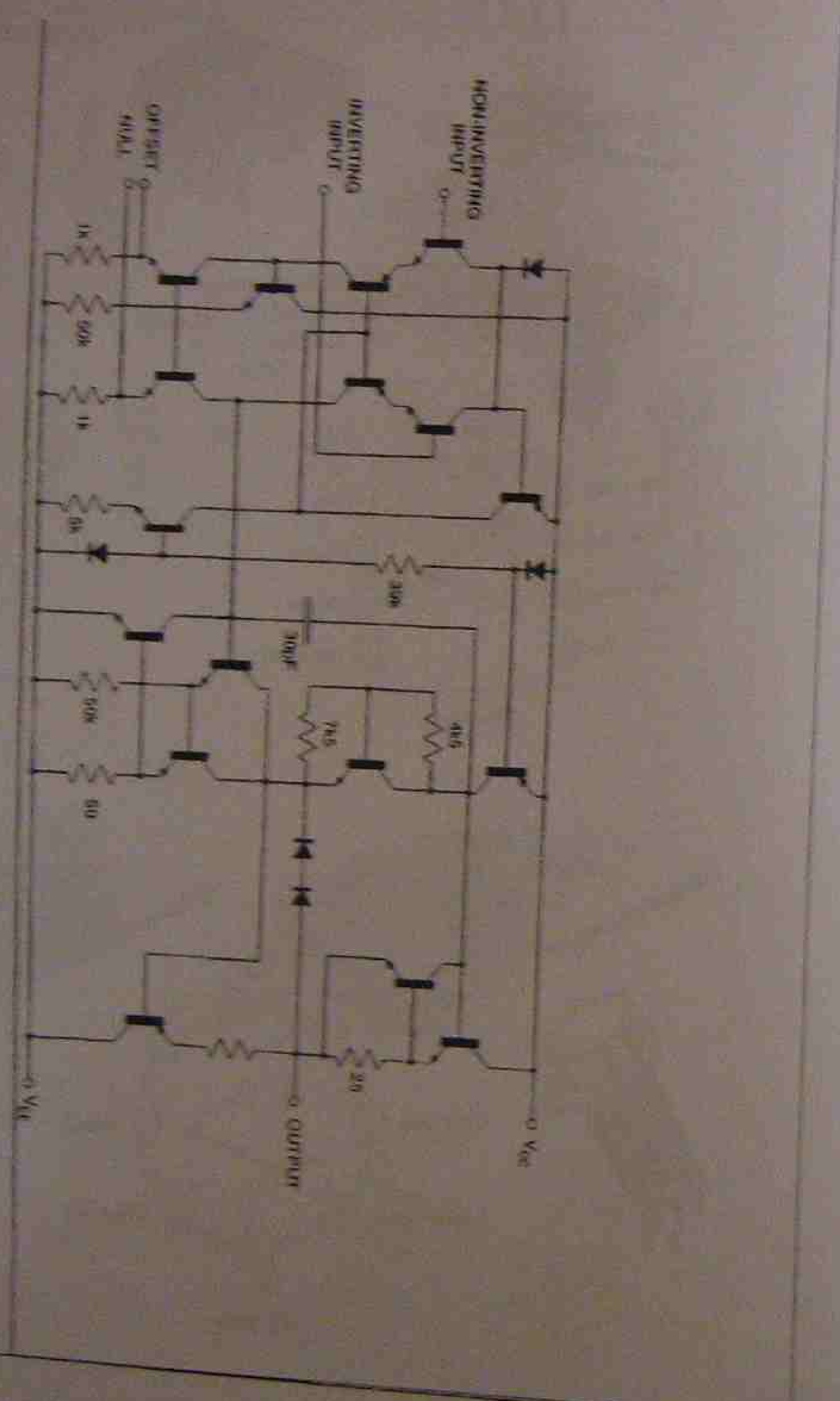


Figure 1.7 shows the internal circuit diagram of an LM741 general purpose operational amplifier. It consists of transistors, resistors, diodes and a capacitor. All discrete components are fabricated in a tiny silicon chip and typical packages are shown in Figure 5 and 6.

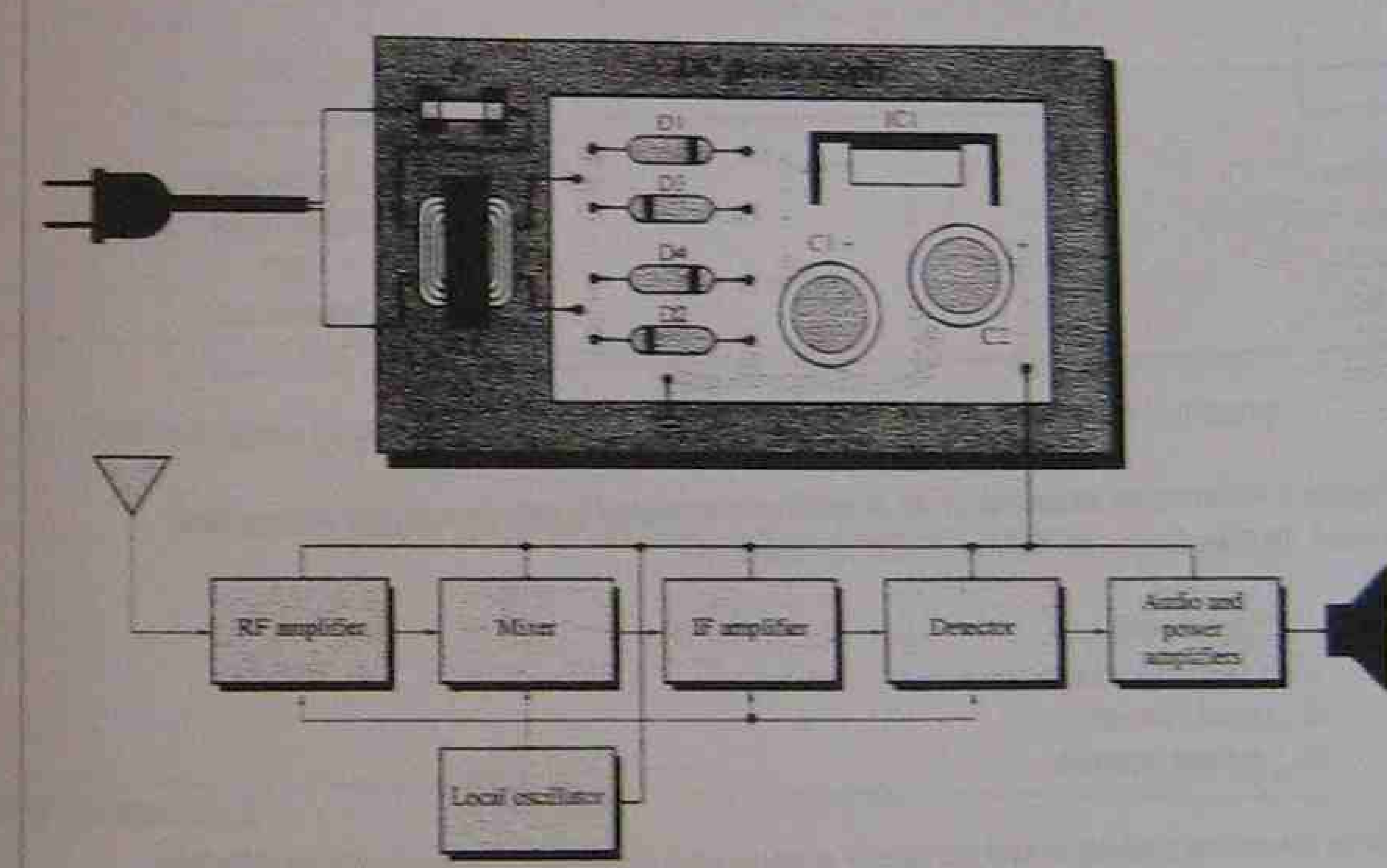
Figure 1.7 LM741 operational amplifier



A single amplifier is usually insufficient to raise the output to the desired level. In such cases the output of the first amplifier is fed into a second, whose output is fed to a third, and so on, until the output level is satisfactory. The result is cascade, or multistage amplification. Long-distance telephone, radio, television, electronic control and measuring instruments, radar, and countless other devices all depend on this basic process of amplification. The overall amplification of a multistage amplifier is the product of the gains of the individual stages.

There are various schemes for the coupling of cascading electronic amplifiers, depending upon the nature of the signal involved in the amplification process. Solid-state microcircuits have generally proved more advantageous than vacuum-tube circuits for the direct coupling of successive amplifier stages. Transformers can be used for coupling, but they are bulky and expensive.

Figure 1.8 Block diagram of a typical radio receiver.



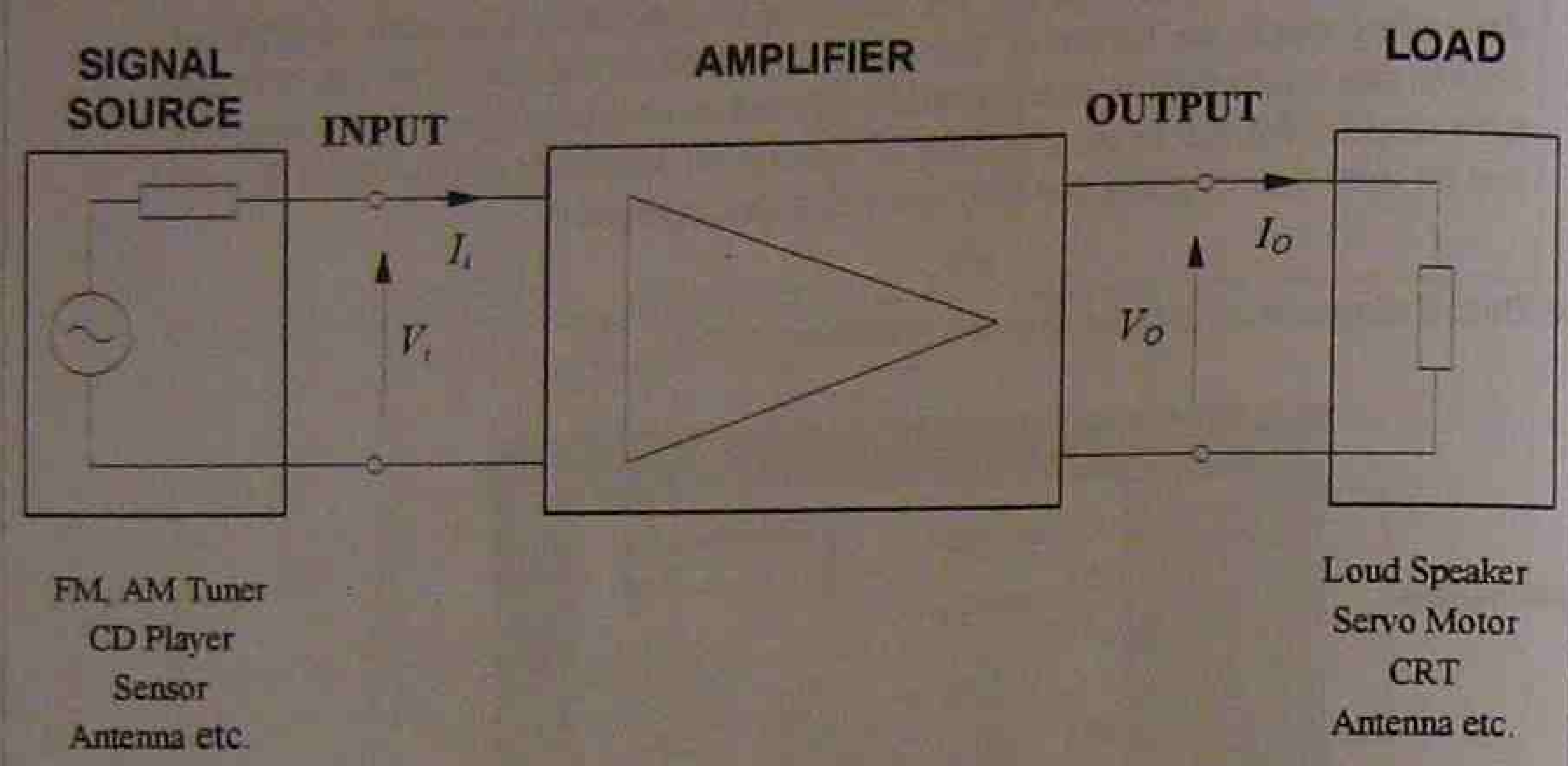
As you can seen from Figure 1.8, a radio receiver needs many amplifiers in cascade to drive a loud speaker. Also, the power supply is an important part because it supplies the DC voltage and current necessary for all other circuits in the system to operate.

An electronic amplifier can be designed to produce a magnified output signal identical in every respect to the input signal. This is linear operation. If the output is altered in shape after passing through the amplifier, amplitude distortion exists. If the amplifier does not amplify equally at all frequencies, the result is called frequency distortion, or discrimination (as in emphasizing bass or treble sounds in music recordings).

1.2 Gain

The amplifier gain is very simply calculated by dividing the output electrical

Figure 1.9



quantity (voltage, current, power) by the input electrical value.

Figure 1.9 shows an example of an amplifier connected between a signal source and a load. In this diagram, V_i, V_O, I_i and I_O represent:

V_i ; input voltage
V_O ; output voltage
I_i ; input current
I_O ; output current

These electrical values could be easily measured by using a CRO or other suitable measuring instruments.

Voltage Gain

The voltage gain (A_V) is the most frequently used gain in amplifier analysis. It is simply calculated by dividing the output voltage by the input voltage. So:

$$A_V = \frac{V_O}{V_i}$$

Drill Question 1

In an amplifier, input voltage V_i and output voltage V_o are measured as shown below. Determine the voltage gain.

(a) $V_i = 0.1\text{V}$, $V_O = 5\text{V}$

(b) $V_i = 2\text{mV}$, $V_O = 5\text{V}$

(c) $V_i = 0.1\mu\text{V}$, $V_O = 50\text{mV}$

Current Gain (A_i)

The current gain is given by dividing output current (I_O) by input current (I_i).

$$A_i = \frac{I_O}{I_i}$$

Drill Question 2

Determine the current gain for:

(a) $I_i = 0.15\text{mA}$, $I_O = 12\text{mA}$

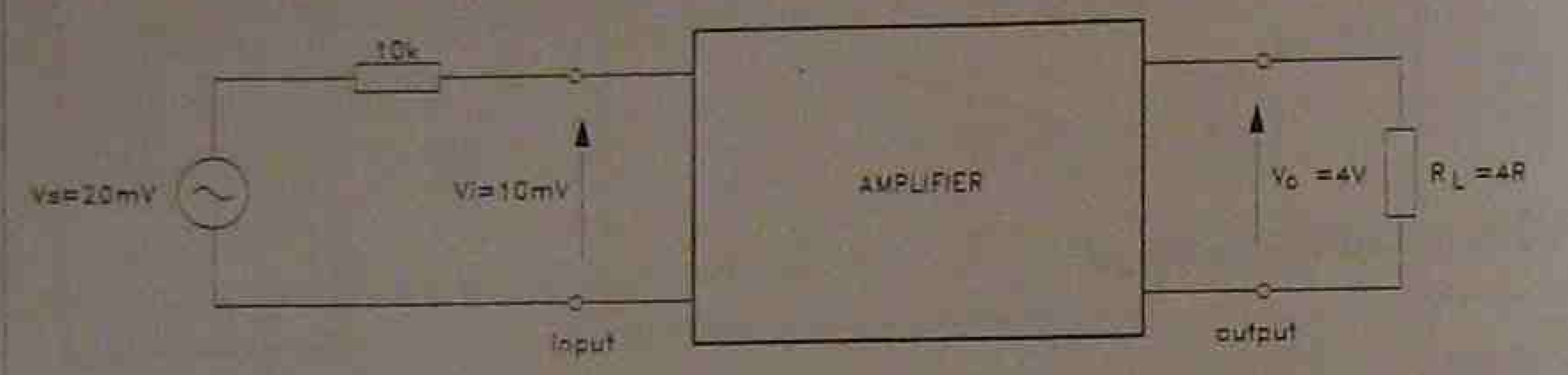
(b) $I_i = 0.15\mu\text{A}$, $I_O = 0.3\text{mA}$

Practical measurement of amplifier current, particularly input current, is very difficult in some cases. For the input current measurement, we may need to use an additional series resistor on the input line to measure the voltage drop across the series resistor. By Ohm's law, then input current is given by dividing voltage drop across the series resistor by the series resistance.

Drill Question 3

For the measurement shown in Figure 1.10, determine the voltage gain and the current gain of the amplifier.

Figure 1.10



(a) Voltage Gain

(b) Current Gain

(i) Determine the voltage drop across 10k resistor.

(ii) Calculate the input current using Ohm's law.

(iii) Calculate the output current.

(iv) Determine the current gain of the amplifier.

Power Gain (A_p)

The input power (P_i) and the out put power (P_O) of an amplifier can be easily calculated using $P(\mathrm{W}) = VI$ concept. Hence:

$$P_i = V_i \times I_i$$

$$P_O = V_O \times I_O$$

Therefore, the power gain is determined:

$$A_p = \frac{P_O}{P_i}$$

Drill Question 4

For the Figure 1.10, determine the power gain of the amplifier.

(i) Calculate the input power using the result in Drill Question 3 (ii).

(ii) Calculate the output power.

(iii) Determine the power gain.

Transresistance (R_T)

The **transresistance** of an amplifier is often used to determine an output voltage from an input current. It is sometimes called *transresistance gain* and defined:

$$R_T = \frac{V_O}{I_i} \text{ (V/I or } \Omega)$$

Transconductance (G_T)

Transconductance is a measure of the amplifier's ability to produce an output current from an input voltage. It is sometimes called *transconductance gain* and defined:

$$G_T = \frac{I_O}{V_i} \text{ (I/V or S)}$$

Drill Question 5

For the measurement in Figure 1.10, determine the transresistance and transconductance.

(a) Transresistance

(b) Transconductance

1.3 Gain in Decibels (dB)

In the previous section, the gain of an amplifier has been given as a straight ratio of two numbers. It is also quite common to convert this gain figure to **decibels** or **dB**.

Power Gain in dB

Power gain in dB is defined:

$$A_p(dB) = 10\log_{10}\frac{P_O}{P_i}$$

Voltage Gain and Current Gain in dB

Voltage gain or current gain in dB can be easily derived using power gain in dB concept.

$$A_V(dB) = 20\log_{10}\frac{V_O}{V_i}$$

$$A_i(dB) = 20\log_{10}\frac{I_O}{I_i}$$

Voltage or Current gain in dB from Power gain in dB

Generally, power gain A_P can be expressed:

$$A_p(dB) = 10\log\frac{P_2}{P_1}$$

From $P = \frac{V^2}{R}$ (W)

$$A_p(dB) = 10\log\frac{\frac{V_2^2}{R}}{\frac{V_1^2}{R}} = 10\log\frac{V_2^2}{V_1^2} = 10\log\left(\frac{V_2}{V_1}\right)^2 = 20\log\frac{V_2}{V_1}$$

Therefore, voltage gain must be:

$$A_v(dB) = 20\log\frac{V_O}{V_i}$$

Similarly, from $P = I^2R$ (W)

$$A_p(dB) = 10\log\frac{I_2^2R}{I_1^2R} = 10\log\left(\frac{I_2}{I_1}\right)^2 = 20\log\frac{I_2}{I_1}$$

Therefore, current gain A_i must be:

$$A_i(dB) = 20\log\frac{I_O}{I_i}$$

Table 1.
Frequently used numbers in dB

Direct Ratio	Power Gain (dB)	Voltage Gain (dB)
1000	30	60
100	20	40
10	10	20
2	3	6
$\sqrt{2} = 1.4142$	1.5	3
1	0	0
$\frac{1}{\sqrt{2}} = 0.707$	-1.5	-3
0.5	-3	-6
0.1	-10	-20
0.01	-20	-40
0.001	-30	-60

Drill Question 6

Express the following gains in dBs. Try with calculator and without calculator but using table 1.1.

(a) Voltage gain $A_V = 200$

(b) Voltage gain $A_V = 0.02$

(c) Current gain $A_i = 71$

(d) Power gain $A_P = 50$

Drill Question 7

For figure 1.10, determine:

(a) Voltage Gain (dB)

(b) Current Gain (dB)

(c) Power Gain (dB)

Gain Conversion from dB to Ratio

Voltage gain in direct ratio: $A_V = 10^{\left(\frac{A_V(dB)}{20}\right)}$

Current gain in direct ratio: $A_i = 10^{\left(\frac{A_i(dB)}{20}\right)}$

Power gain in direct ratio: $A_P = 10^{\left(\frac{A_P(dB)}{10}\right)}$

Drill Question 8

Convert the following gains in dBs to direct ratio.

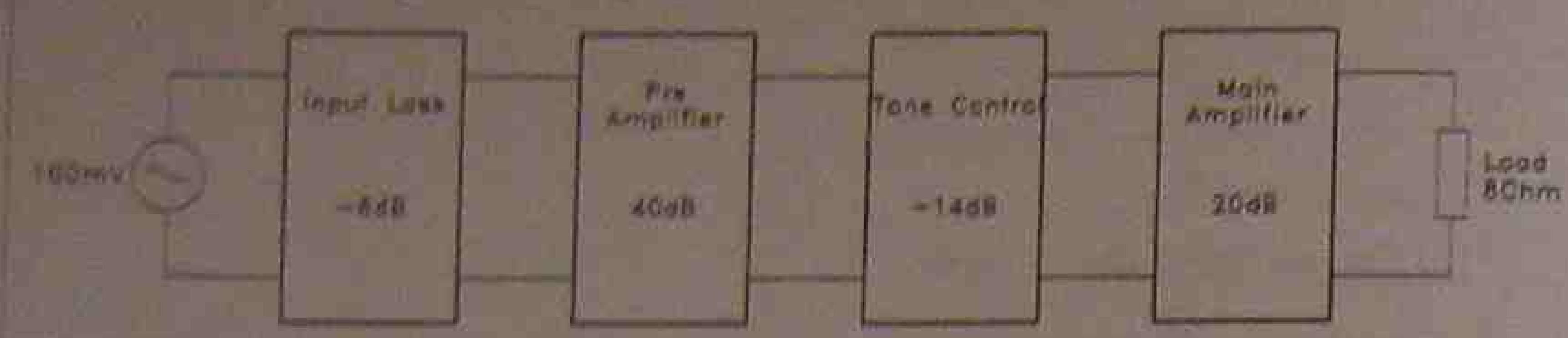
(a) Voltage gain $A_V = 50$ (dB)

(b) Power gain $A_P = 15$ (dB)

Drill Question 9

For an audio amplifier in Figure 1.11, determine:

Figure 1.11 An example of voltage gain of an audio amplifier.



(a) Total voltage gain in dB

(b) Total voltage gain in direct ratio.

(c) Input voltage of the pre amplifier.

(d) Input voltage of the main amplifier.

(e) Output voltage of the amplifier.

(f) (optional) Output power(W). Assume that input signal is sine wave.

1.4 Input and Output Resistance of the Amplifier

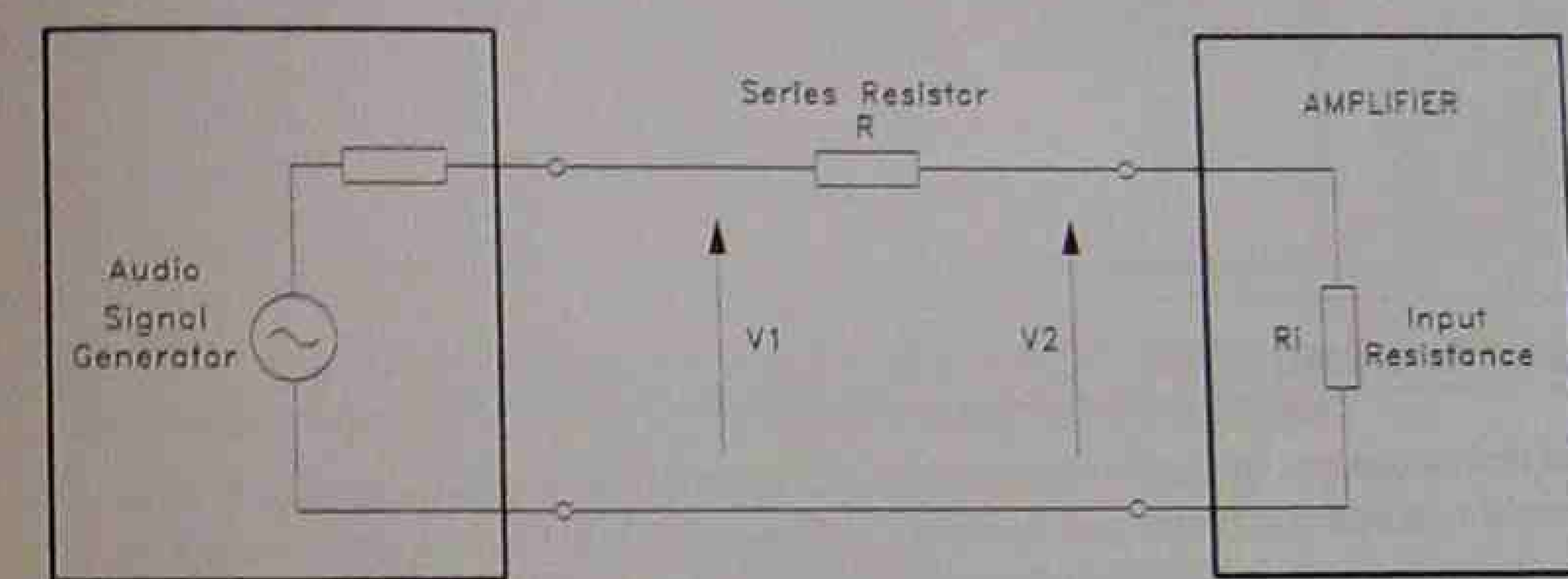
Input Resistance

The input resistance of an amplifier is defined to be a resistance which is seen from the input terminals of an amplifier. Input resistance can be simply calculated by dividing the input voltage by the input current. So:

$$R_i = \frac{V_i}{I_i}\ (\Omega)$$

However, input current measurement is practically difficult. Therefore, as shown in Figure 1.12, a series resistance is normally used for the input resistance measurement.

Figure 1.12
Input resistance measurement.



From figure 1.12, input current of the amplifier is the current through the series resistor R. Also, V_1 and V_2 can be easily measured using an oscilloscope or other suitable instruments. Since the voltage drop across R is $V_1 - V_2$, therefore the input current is:

$$I_i = \frac{V_1 - V_2}{R}$$

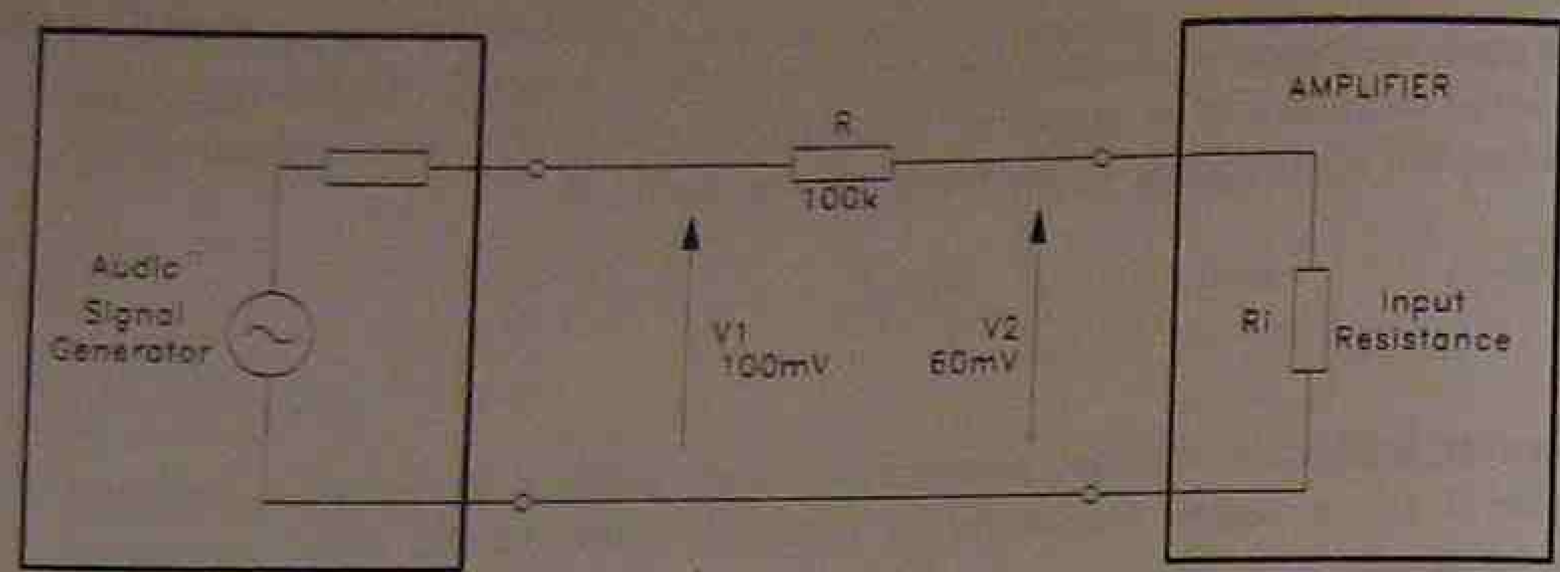
Also, V_2 is the input voltage of the amplifier V_i. Therefore, the input resistance of the amplifier can be determined:

$$R_i = \frac{V_i}{I_i} = \frac{V_2}{\frac{V_1 - V_2}{R}}$$

$$= \frac{V_2}{V_1 - V_2} \times R\ (\Omega)$$

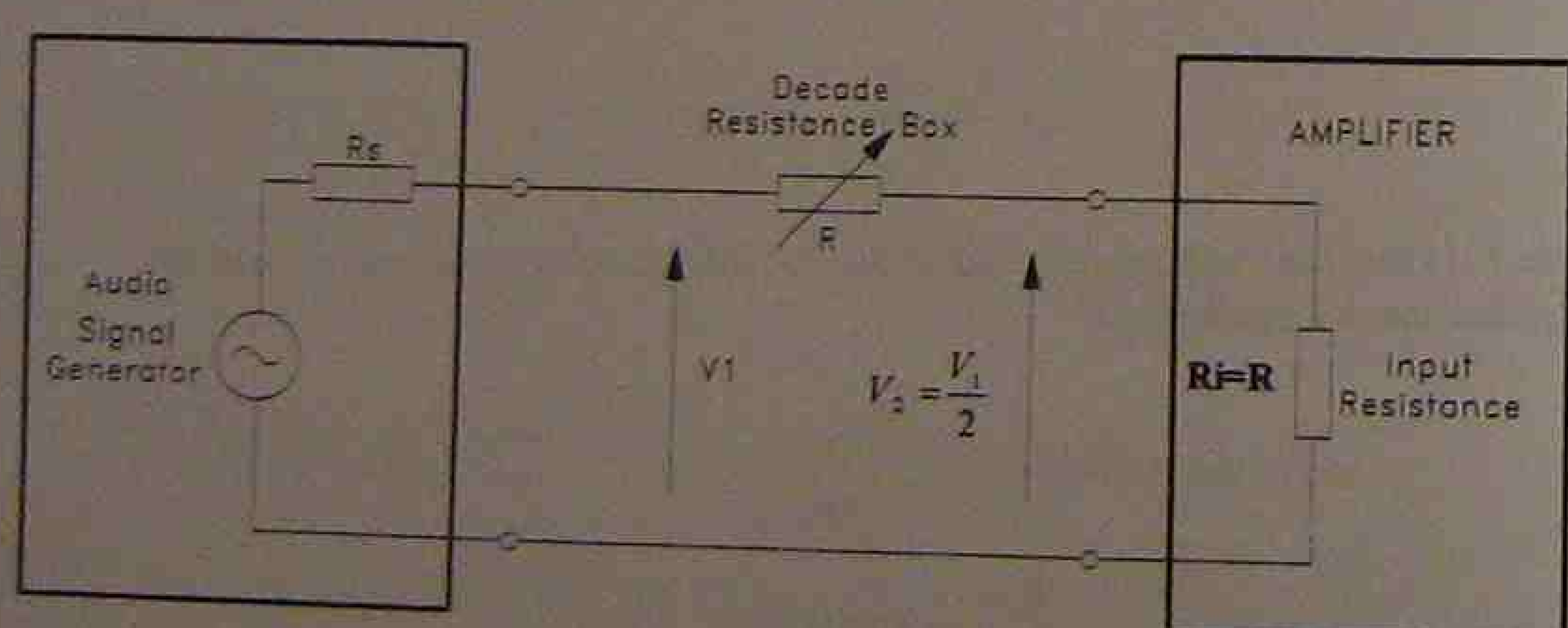
Drill Question 10

For the measurements shown below, determine the input resistance of the amplifier.



Input resistance can be easily measured by replacing the series fixed resistor R with a variable decade resistance box as shown in Figure 1.13. If V_2 is adjusted to half of V_1 by using the decade resistance box, the reading of the decade resistance box is the same value as the input resistance of the amplifier.

Figure 1.13
Input Resistance measurement using decade resistance box.

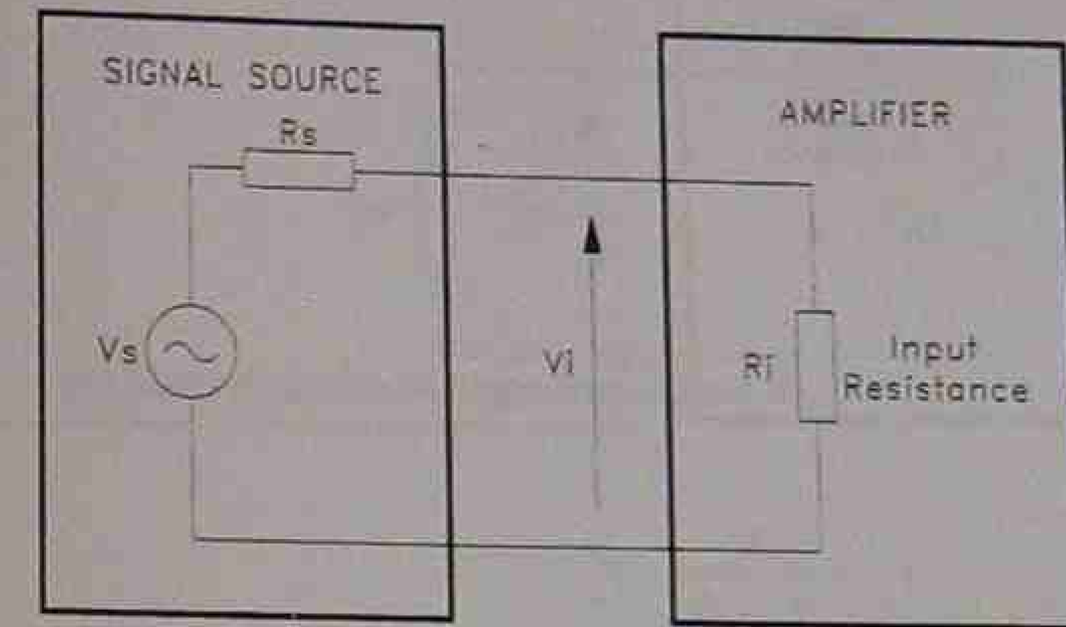


Loss due to the Input Resistance

Every signal source has an associated source resistance, R_S, like the audio signal generator in Figure 1.12. It is called internal resistance or output resistance of the signal source.

Together the source resistance and the input resistance act like a voltage divider. Figure 1.14 shows the voltage source attached to the amplifier's input.

Figure 1.14
A signal source feeding an amplifier



This means that the signal is spread across Rs and Ri. Only a fraction of the signal can get into the amplifier. In fact, using voltage divider principle:

$$V_i = \frac{R_i}{R_S + R_i} \times V_S$$

Drill Question 11

In figure 1.14, V_S = 3mV, R_S = 600Ω and R_i = 1.2kΩ. Determine:

(a) Input voltage of the amplifier.

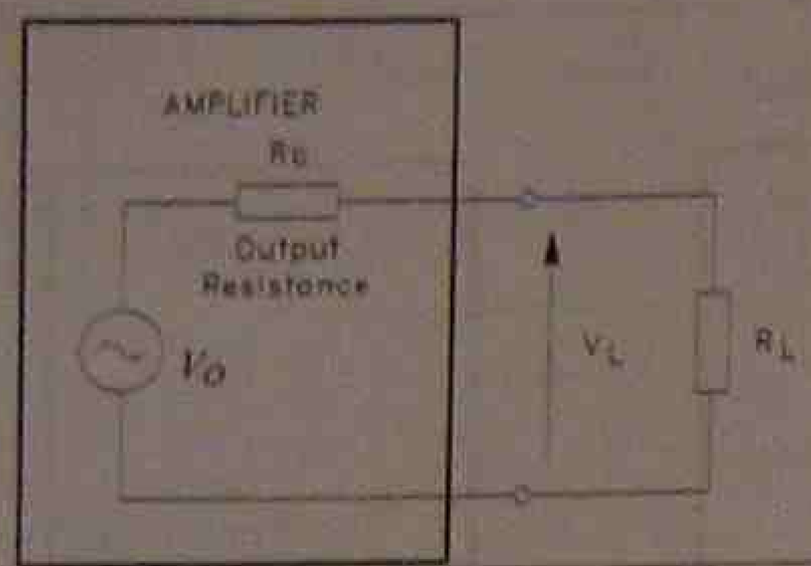
(b) Input loss in dB.

Explain how input loss can be avoided.

Output Resistance

The output of an amplifier is an amplified input signal. Therefore, like a signal source, we can easily expect an internal resistance seen from output terminals. This is called output resistance. As shown in Figure 1.15, again we have the situation of a voltage spread across two resistances, R_O and R_L.

Figure 1.15



$$V_L = \frac{R_L}{R_O + R_L} \times V_O$$

Output Resistance Measurement

From Fig. 1.15, the voltage drop across R_O is $V_O - V_L$ and current through the circuit is $\frac{V_L}{R_L}$. Therefore, output resistance R_O is determined:

$$R_O = \frac{V_{R_O}}{I_L} = \frac{V_O - V_L}{\frac{V_L}{R_L}}$$

$$= \frac{V_O - V_L}{V_L} \times R_L$$

Drill Question 12

For an amplifier, the output voltage is measured at 10V without load and 8V with 8Ω load.

(a) Draw equivalent circuit (Fig. 1.15).

(b) Determine the voltage drop across R_O.

(c) Determine the current through R_O using load voltage and load current.

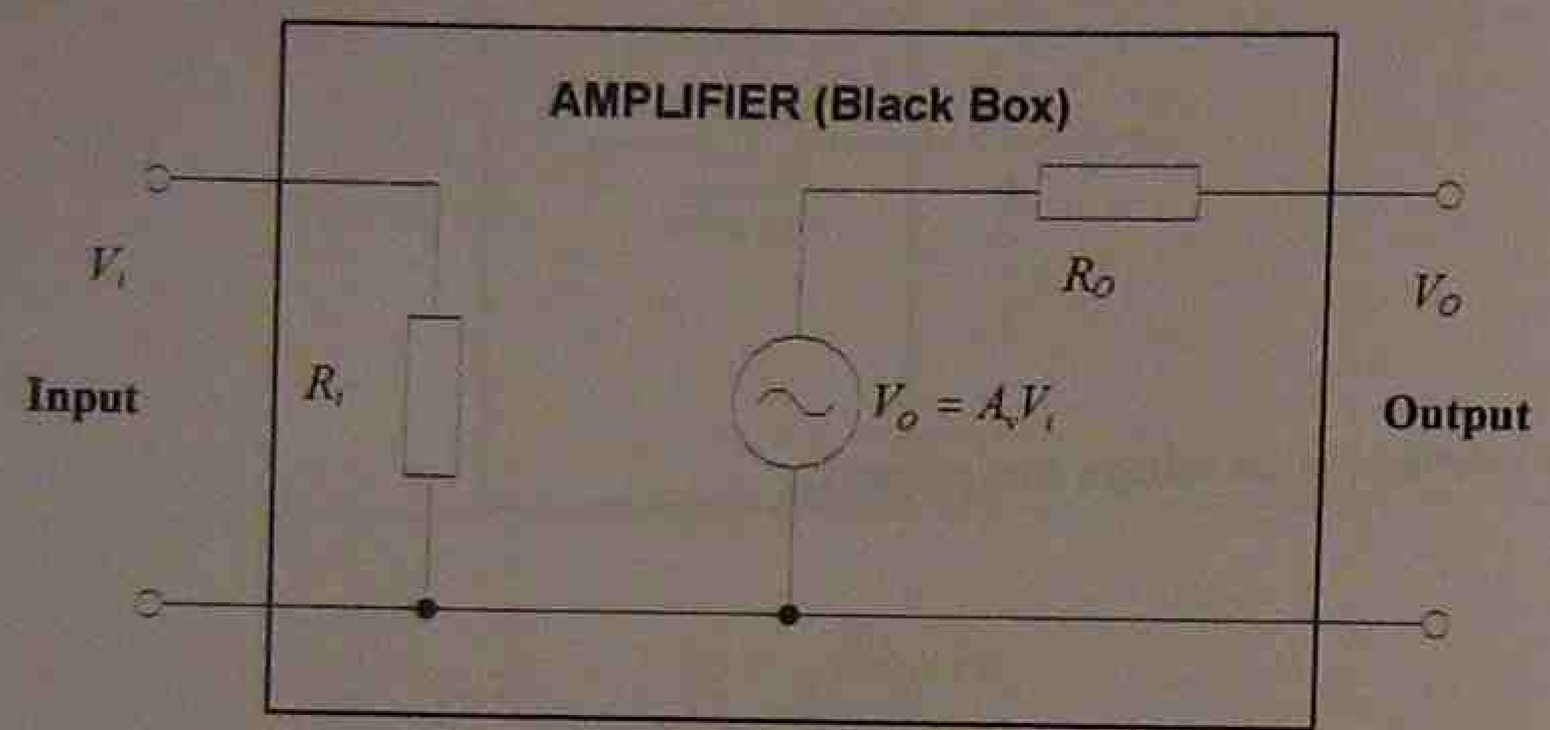
(d) Determine the value of output resistance of the amplifier.

(e) Discuss about the output resistance of an ideal amplifier.

1.5 Amplifier Equivalent Model

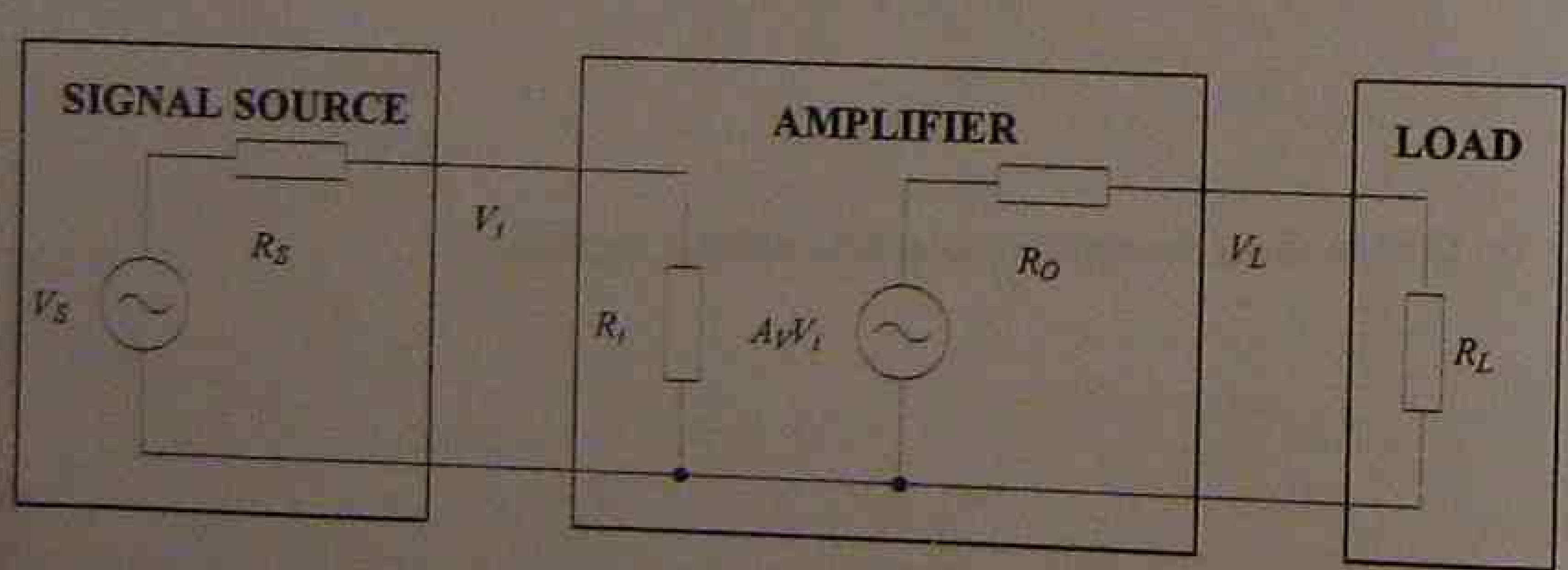
If we assume that an amplifier can be regarded as a black box, then it can be expressed with its input resistance (R_i), output resistance (R_O) and output voltage without load (V_O) regardless what electronic devices are employed to construct the amplifier.

Figure 1.16 An amplifier equivalent model.



This amplifier equivalent model is widely used to determine the exact input voltage and out voltage when signal source and load are connected to an amplifier. Figure 1.17 shows how it works.

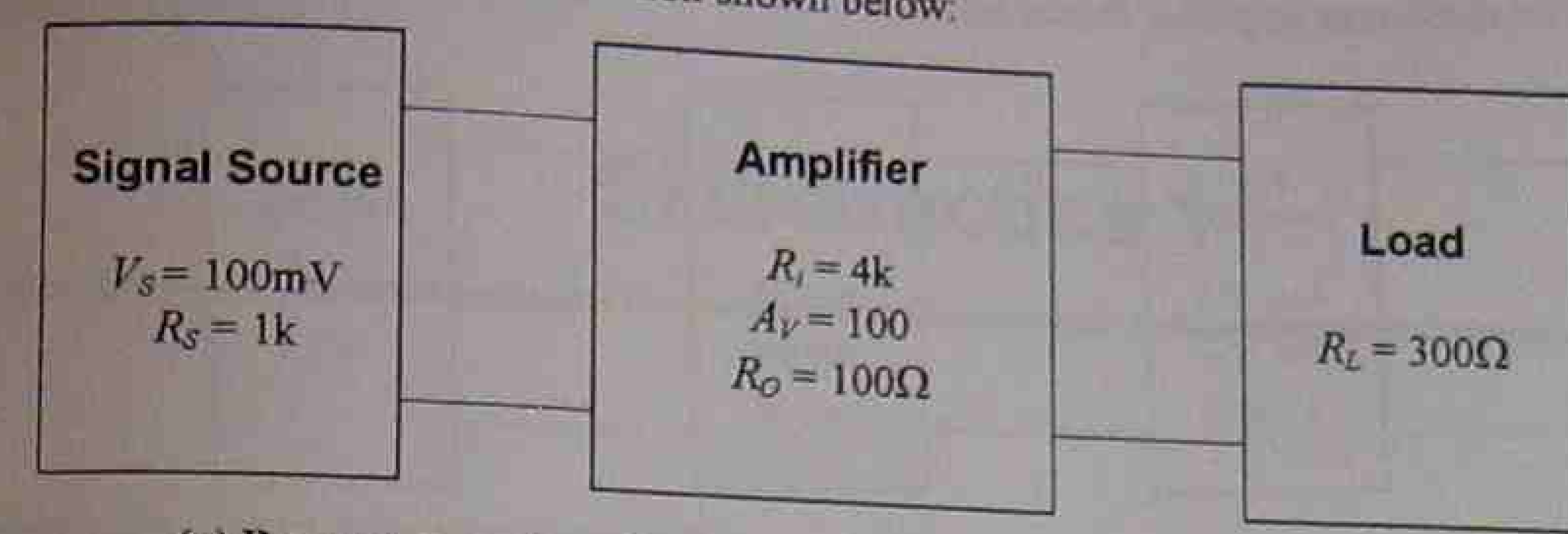
Figure 1.17 An amplifier between a signal source and a load.



$$V_i = \frac{R_i}{R_S + R_i} V_S \qquad V_L = \frac{R_L}{R_O + R_L} A_v V_i$$

Drill Question 13

For the amplifier connection shown below:



(a) Draw an equivalent circuit (Fig. 1.17).

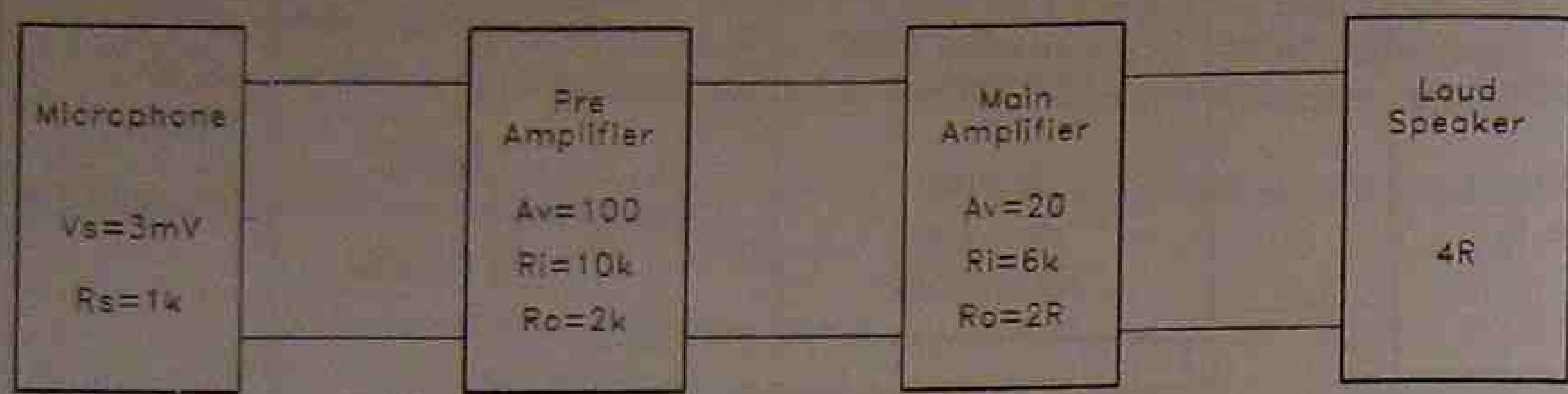
(b) Determine the input voltage of the amplifier.

(c) Determine the output voltage of the amplifier.

(d) Determine the voltage gain of the amplifier required to achieve 10V across the load.

Drill Question 14

For the multistage amplifier shown below:



(a) Draw an equivalent circuit.

(b) Calculate the input voltage of the preamplifier.

(c) Calculate the input voltage of the main amplifier.

(d) Calculate the load voltage.

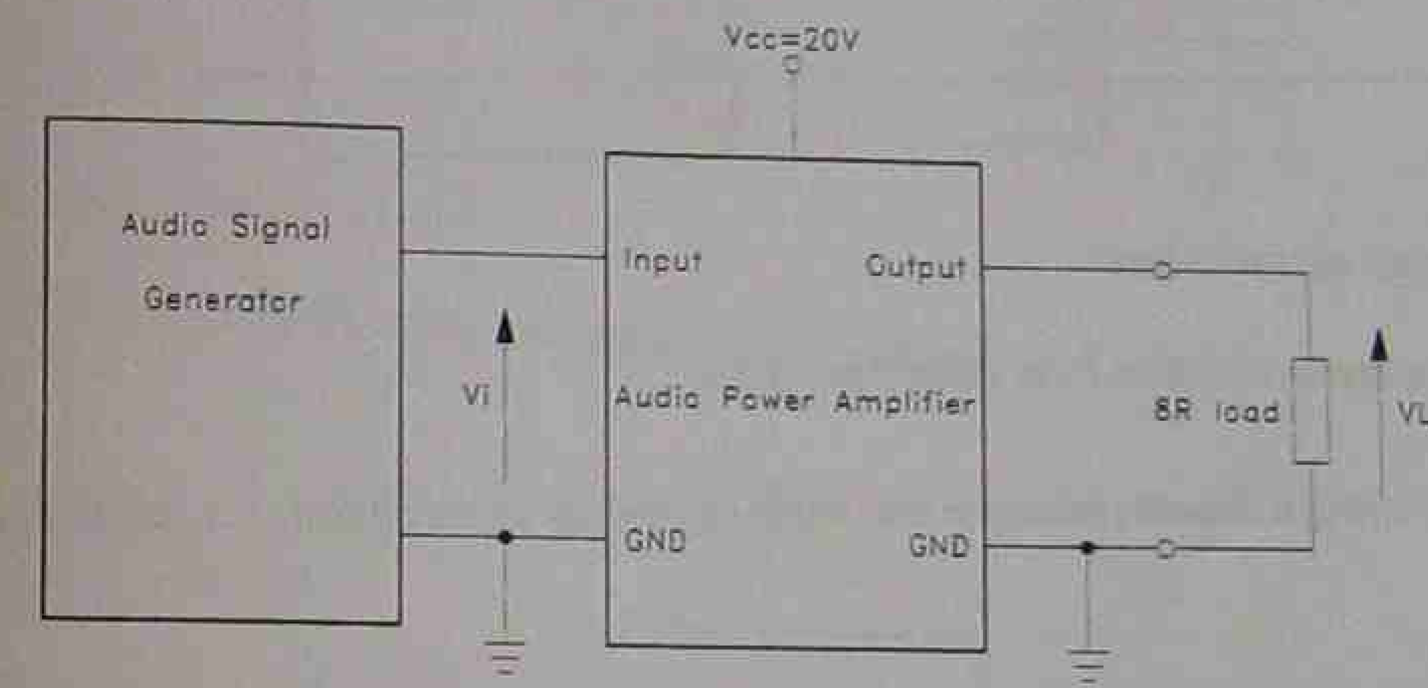
(e) Determine the output power.

Amplifiers 1

Skill Practice 1

Voltage Gain Measurement

(a) Construct the circuit shown below.



(b) Adjust the input voltage (Vi) to 100mVpp 1kHz sine wave.

(c) Observe the output waveform. If the output waveform is distorted or clipped, reduce the input voltage.

(d) Measure the output voltage (Vo_{pp}) by using an oscilloscope.

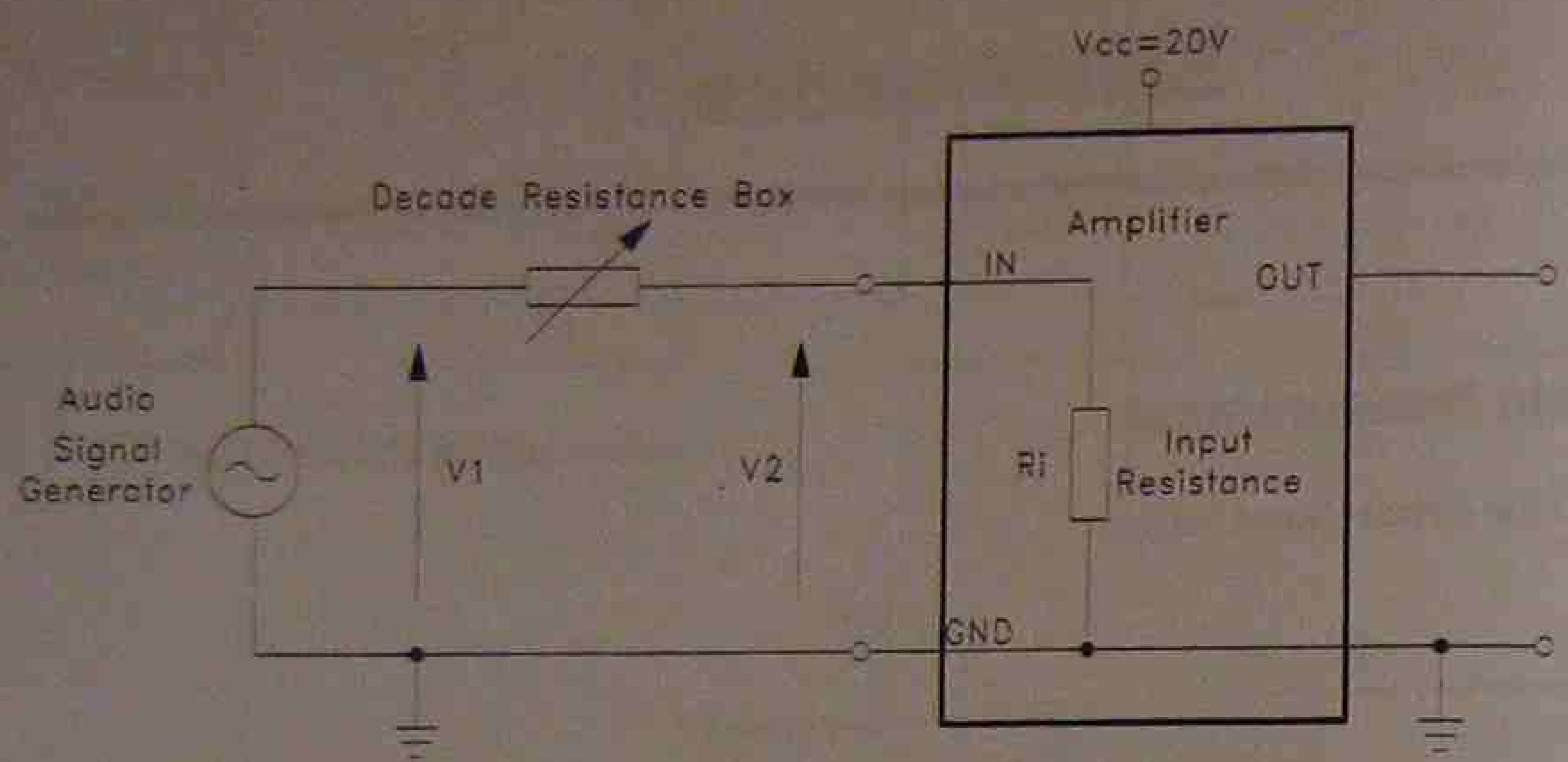
Vo =

(e) Calculate voltage gain in ratio and dB.

(f) Repeat (a)~(e) for Vi=100mVpp 100kHz.

(g) Repeat (a)~(e) for Vi=100mVpp 50Hz.

Input Resistance Measurement



(a) Construct the circuit shown below.

(b) Adjust the audio generator so that V_1 is 100mVpp, 1kHz.

(c) Adjust $V_2 = \frac{V_1}{2}$ by using a decade resistance box while observing Vo waveform. Vo should not be clipped.

(d) Read the value of decade resistance box.

Ri = R =

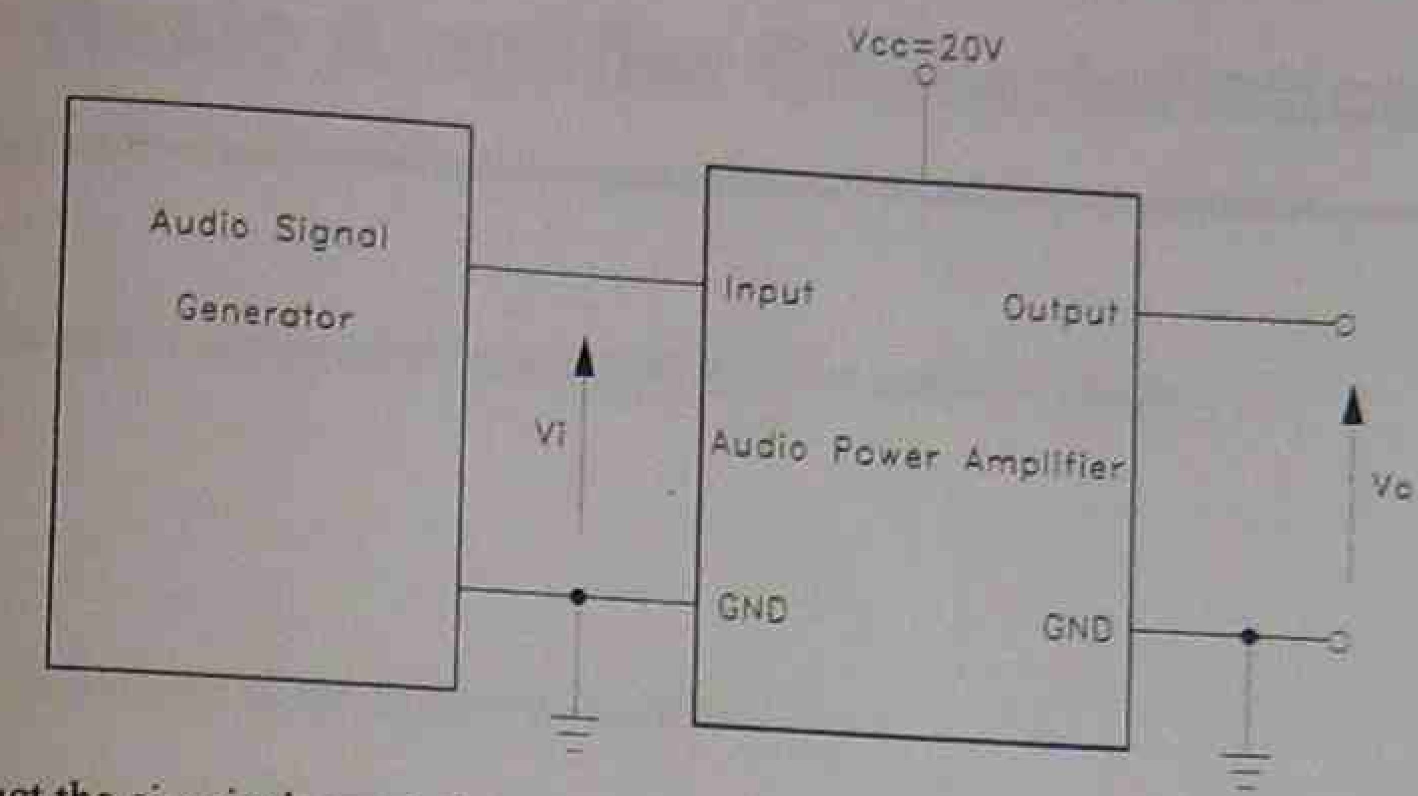
(e) Repeat (a) ~ (d) for:

100Hz input

10kHz input

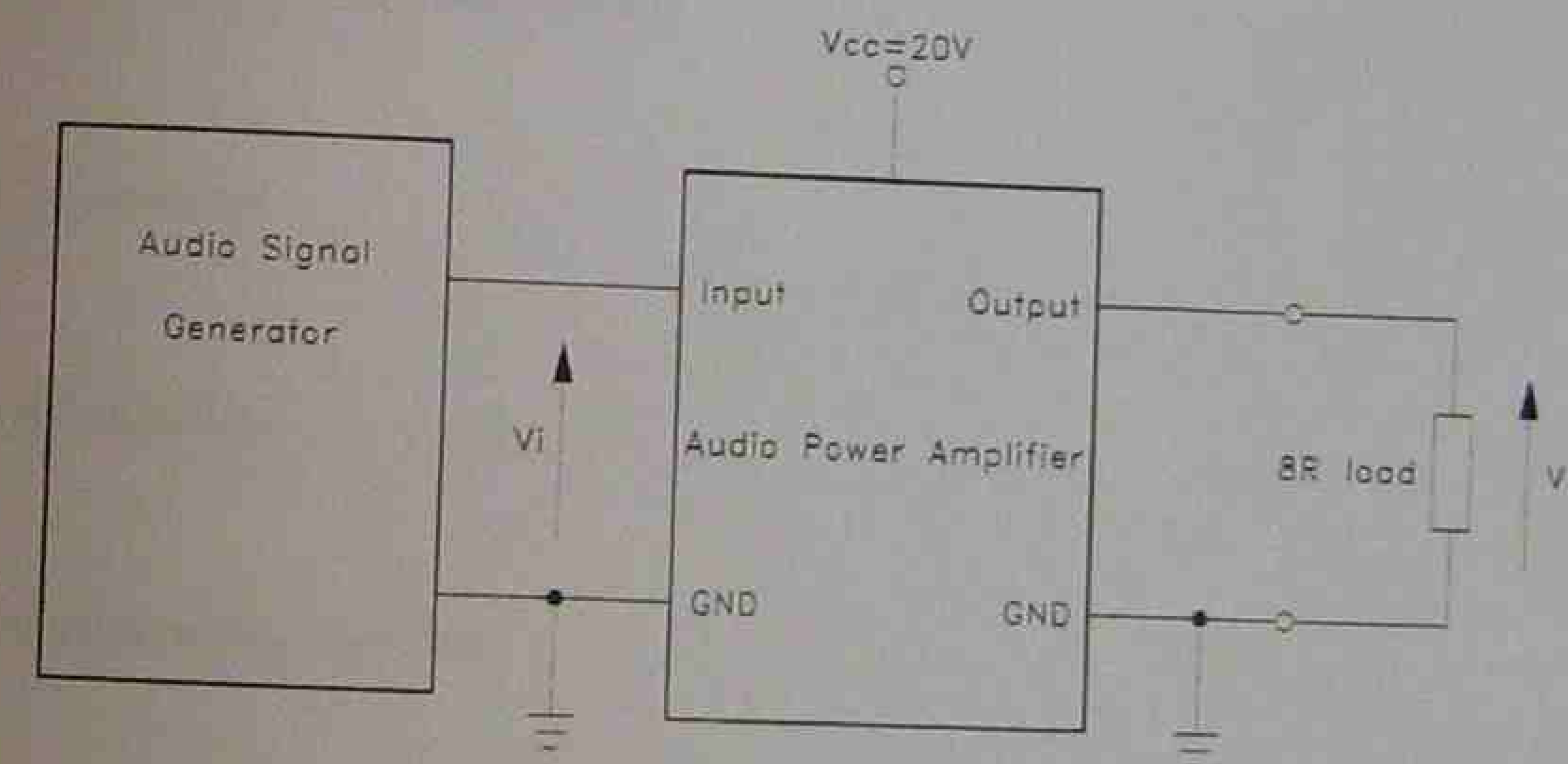
100kHz input

Output Resistance Measurement



(a) Construct the circuit shown below.

(b) Adjust audio generator for 1kHz sine wave and Vo=10Vpp.



(c) Connect 8Ω load as shown below.

(d) Measure the load voltage V_L.

(e) Calculate the output resistance of the amplifier.

$$R_o = \frac{V_o - V_L}{V_L} \times R_L$$

(f) Repeat (a) ~ (e) for 100Hz, 10kHz and 100kHz.

Review Questions 1

Basic Amplifier Characteristics

Q1. List the names of discrete electronic devices used for amplifier design.

Q2. What is the purpose of the following amplifiers?

(a) Small signal audio amplifier.

(b) Large signal audio power amplifier.

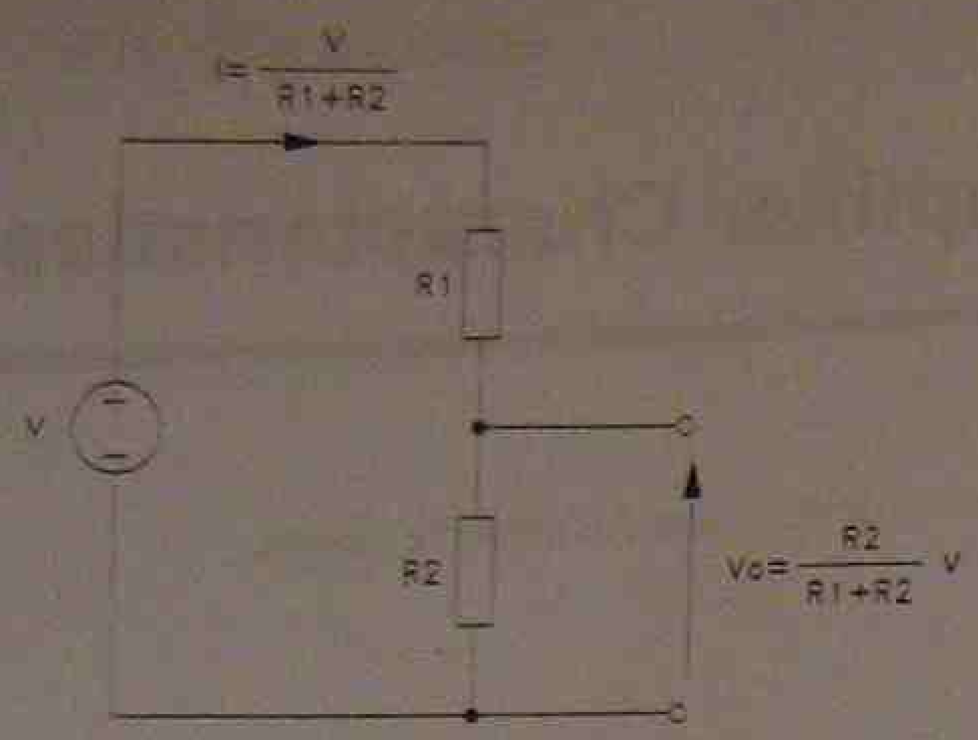
(c) DC amplifier.

(d) RF amplifier.

(e) Instrumentation amplifier.

Q3. Draw a voltage divider circuit diagram and show how to calculate the voltage drop across each of ther resistors.

Q4. For the circuit shown below, calculate the total current I and the output voltage Vo for:



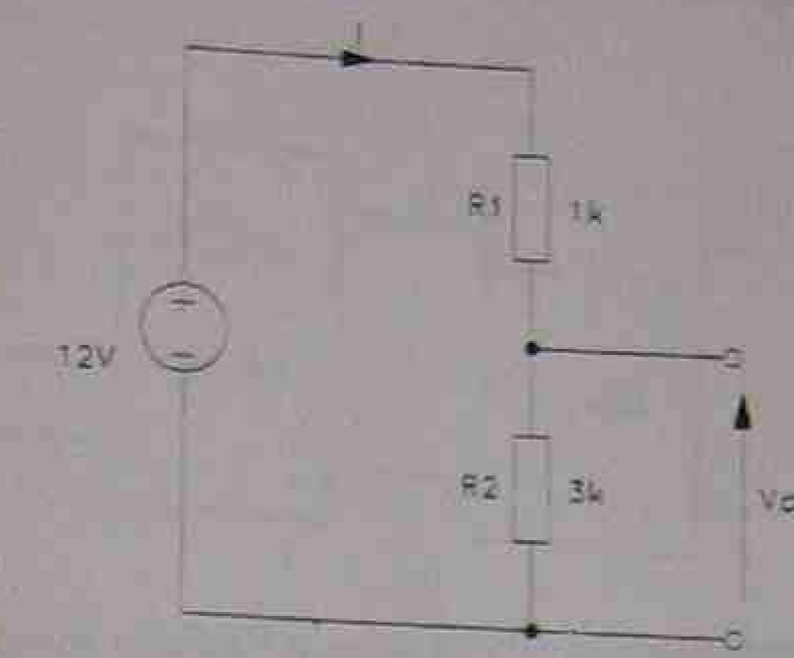
(a) V=10V, R1=600Ω, R2=1.4k

(b) V=12V, R1=4Ω, R2=8Ω

(c) V=15V, R1=0Ω, R2=2k

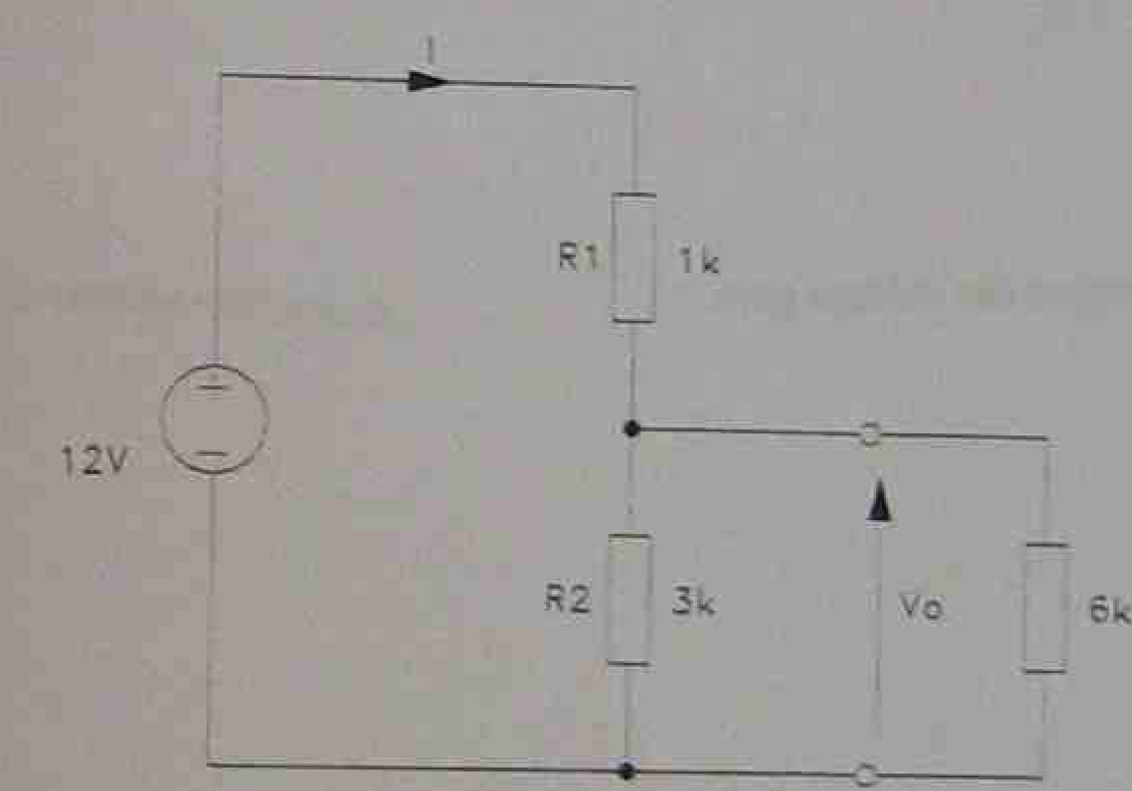
(d) V=18V, R1=2k, R2=0Ω

Q5. For a voltage divider circuit shown below:



(a) Calculate I and Vo.

(b) A 6k load is connected to the output terminal as shown below. Calculate Vo.

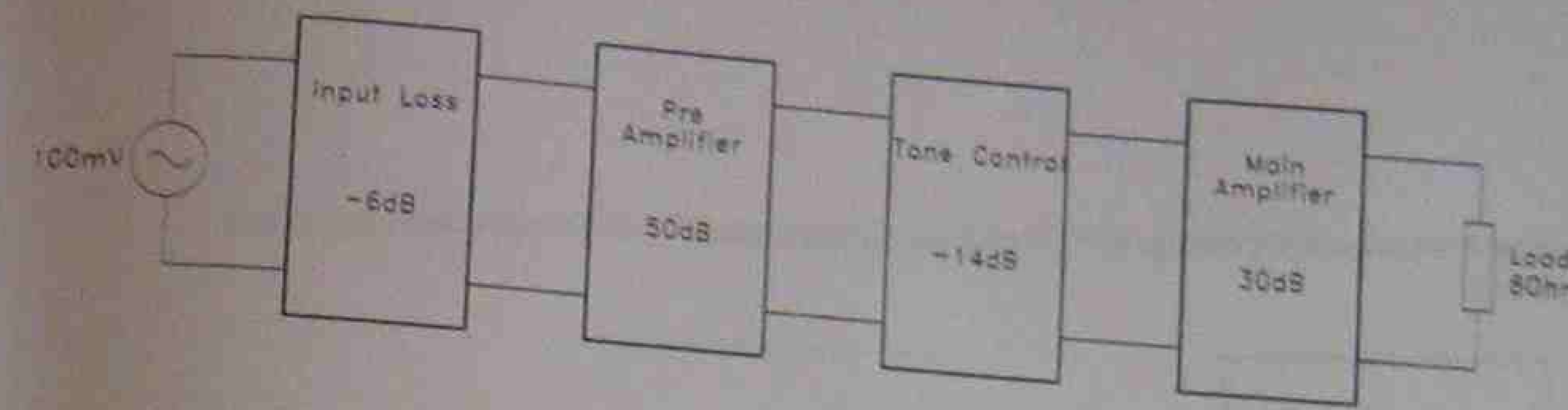


Q6. What is meant by the term 'voltage gain' in an amplifier?

Q7. In an amplifier, the output voltage is measured as 10Vpp when a 10mVpp input voltage is applied. Determine the voltage gain of the amplifier in ratio and dB.

Q8. For the circuit in Q5 (a), determine the voltage gain.

Q9. In a multi-stage electronic system, the voltage gain of each of the stages is measured shown below.

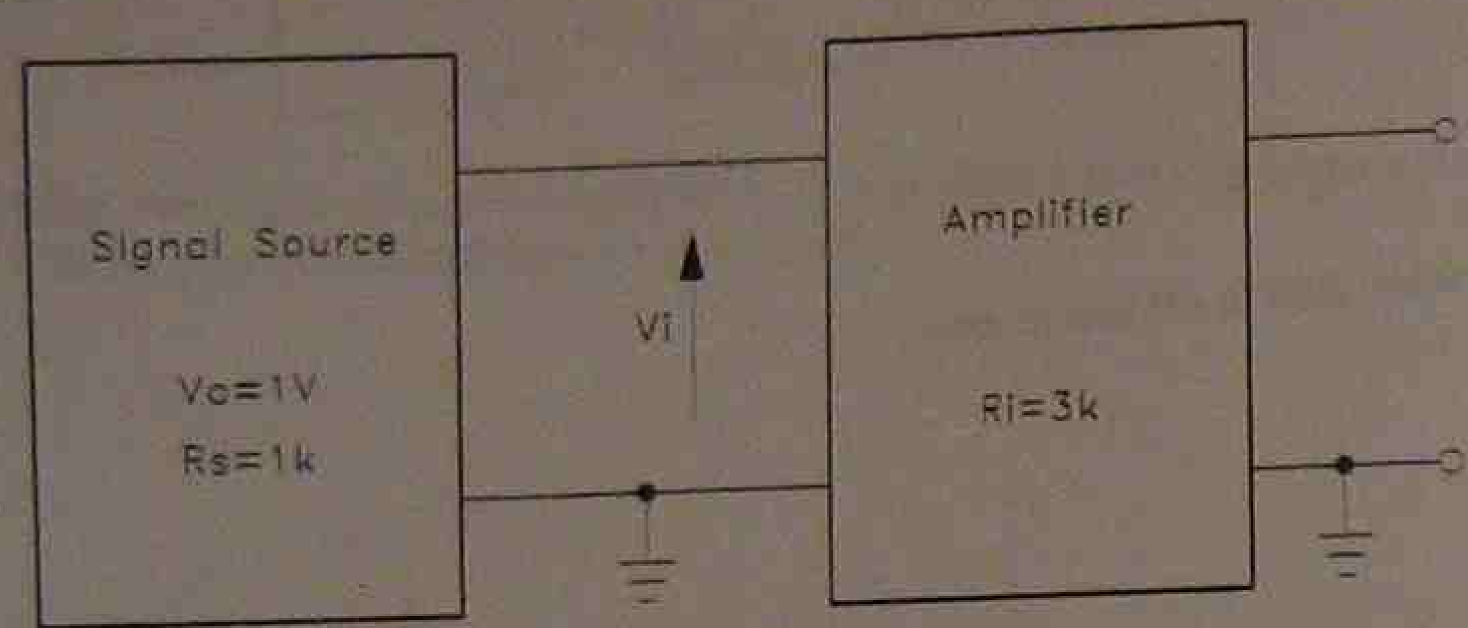


(a) Determine the total voltage gain in dB and in ratio.

(b) Calculate the output voltage and power.

0. What is the definition of the input resistance of an amplifier?

11. A signal source whose output voltage V_S=1V and internal resistance R_S=1k is connected to an ampli-
r as shown below.



(a) Determine the input voltage of the amplifier if the input resistance of the amplifier is 3k.

(i) Draw an equivalent input circuit for the source feeding the amplifier.

(ii) Calculate the amplifier input voltage.

(iii) Determine the input loss in dB.

(b) Repeat (a) for Ri=9k.

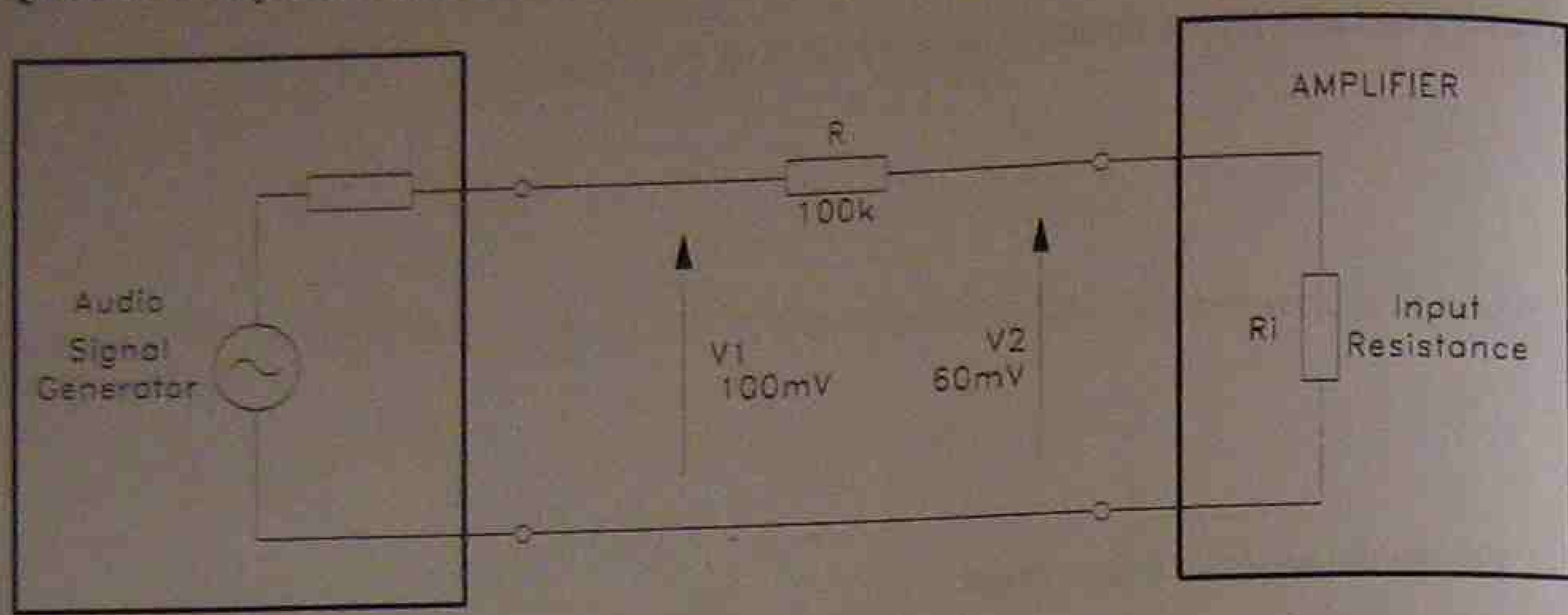
(i) Draw a equivalent input circuit for the source feeding the amplifier.

(ii) Calculate the amplifier input voltage.

(iii) Determine the input loss in dB.

(c) To avoid any loss of the signal input to an amplifier, the input resistance of the amplifier should be ____.

Q12. For an amplifier shown below, determine the input resistance of the amplifier.



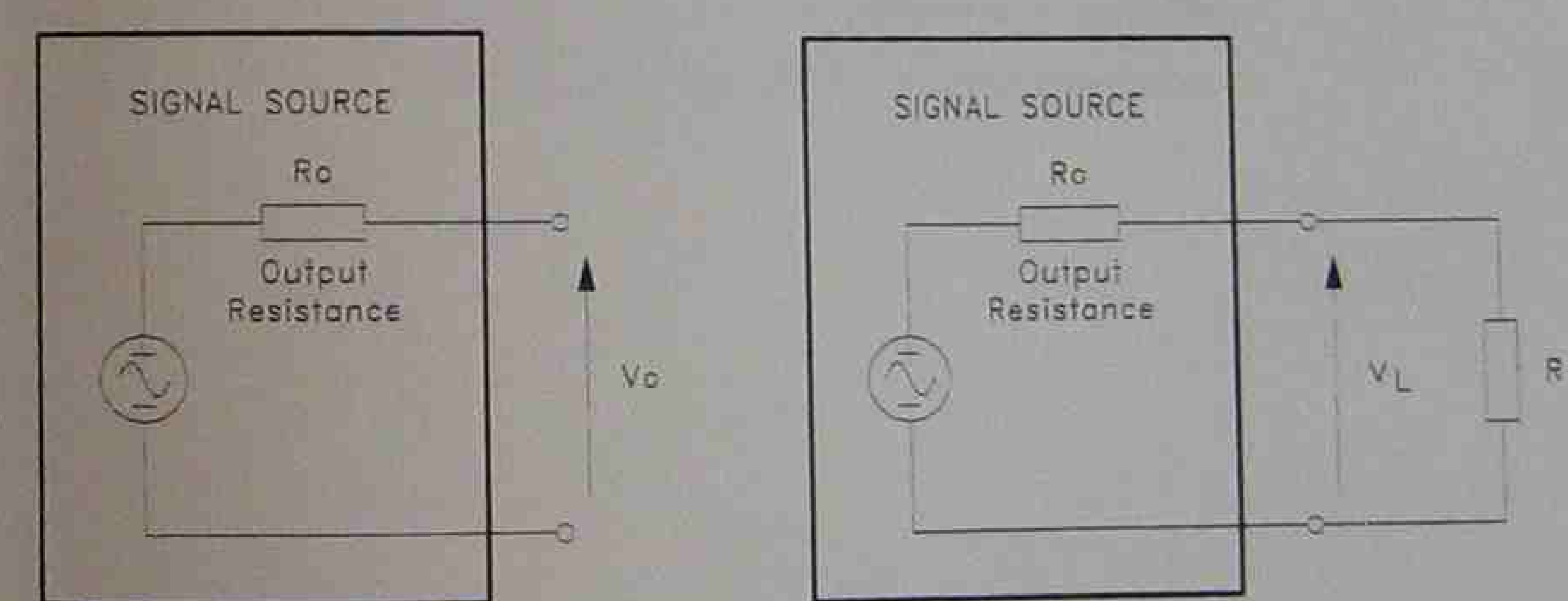
Q13. For the circuit diagram in Q12, a decade resistance box is employed instead of R. State the process of input resistance measurement.

Q14. The output voltage of an amplifier is measured at 10V without load. However, the output voltage drops to 2V when an 8Ω load is connected to output terminals.

(a) Draw the equivalent circuit for the amplifier output.

(b) Determine the output resistance of the amplifier.

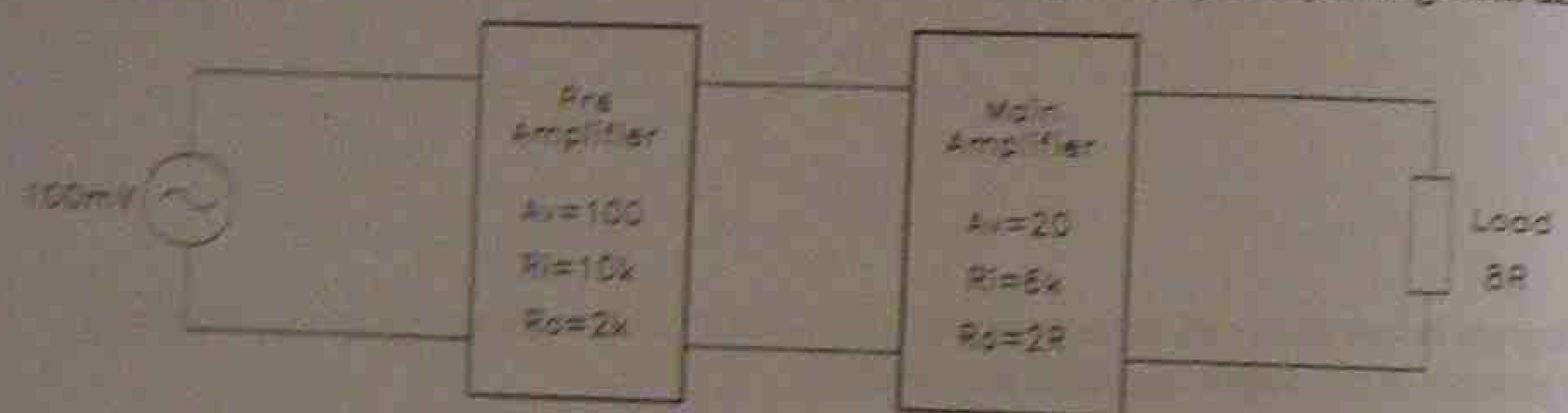
Q15. The diagram below shows an output resistance measurement method for any signal source. Briefly describe the process of the output resistance measurement.



Q16. An ideal voltage amplifier has a/an ____________ input resistance and a/an ________ output resistance.

Q17. Draw a voltage amplifier equivalent model.

Q18. Two amplifiers are connected as shown below. Calculate the output voltage and power gain in dB.



2

Frequency Response

Upon completion of this chapter, you should be able to:

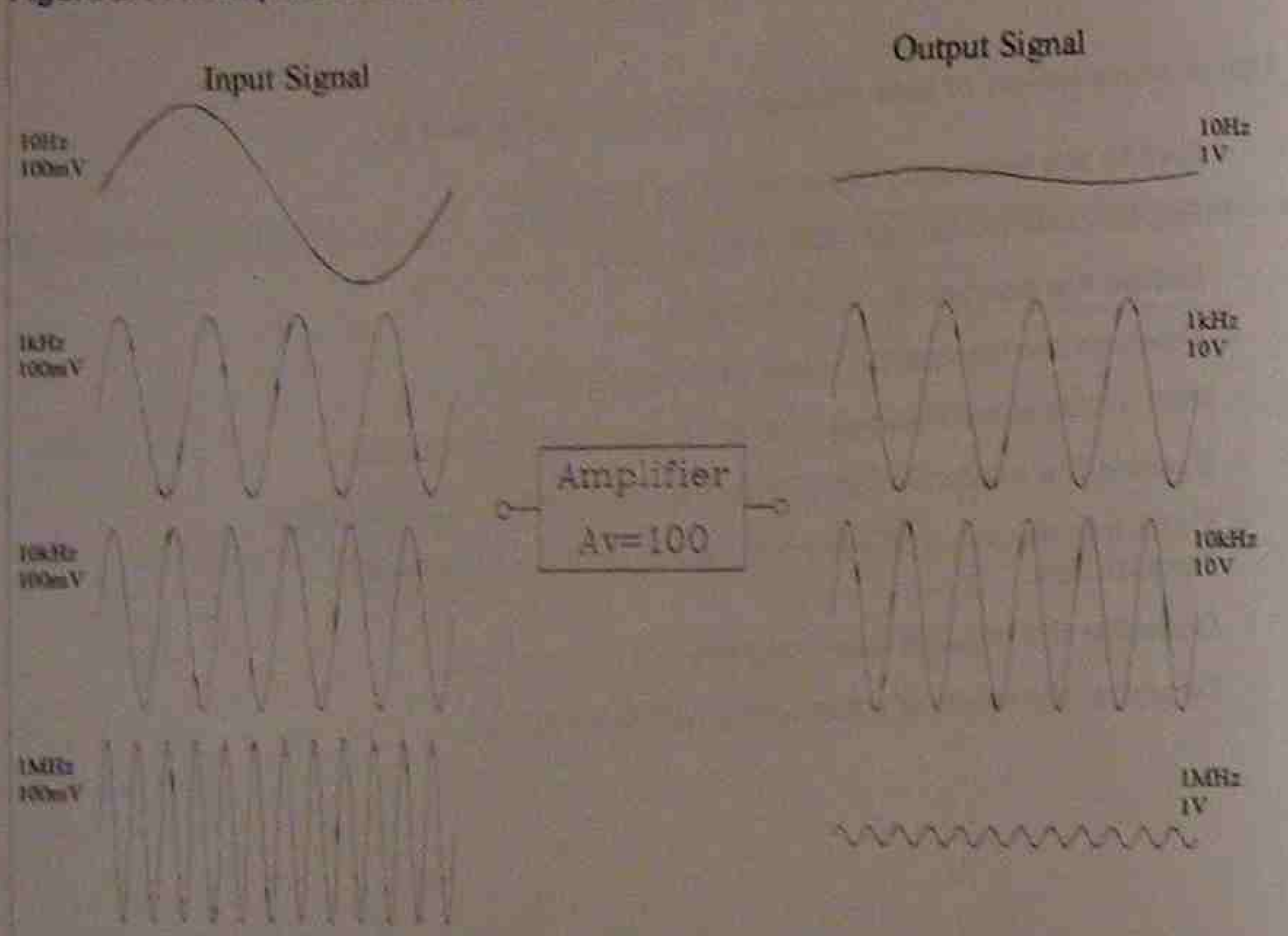
- Define the frequency response of an amplifier.
- Explain half-power (or -3dB) frequencies.
- Define the bandwidth of an amplifier.
- Measure the frequency response of an amplifier.
- Plot a frequency response curve on semi-log graph paper.
- Explain the major factors of frequency limitation of an amplifier.
- Calculate the value of coupling capacitor to limit the lower frequencies.
- Describe the square wave response of an amplifier.
- Describe the slope of a frequency response curve.

2.1 Introduction

No amplifier is perfect. A perfect amplifier would treat all small signals in the same way. In the real world, an amplifier will amplify some signals less than other signals.

The amplifier discriminates against small signals of certain frequency. It is impossible for a real amplifier to amplify signals of all frequencies by the same amount. In general, all amplifiers perform as shown in Figure 2.1.

Figure 2.1 An amplifier handling a variety signals.



In Figure 2.1, several signals are applied to the input of an amplifier. Each of these signals has exactly the same amplitude but each has a different frequency. You can see what happens to each of these signals at the output of the amplifier. At the output, note the signals 10Hz and 1MHz come out with a voltage gain of 10 while the signals 1kHz and 10kHz have come through with voltage gain of 100. Later you will do practical work in which you will observe this for yourself.

2.2 Frequency Response

Frequency response of an electrical circuit is defined as the variation in the output voltage (or current) over a specified range of frequencies. However, the frequency response of an amplifier is normally represented as the variation in the voltage gain (or current gain) over a specified range of frequencies.

Table 2.1 shows measurement results of someone's attempt to determine a particular amplifier's frequency response. These results are seen plotted on the graph in Figure 2.2.

Table 2.1

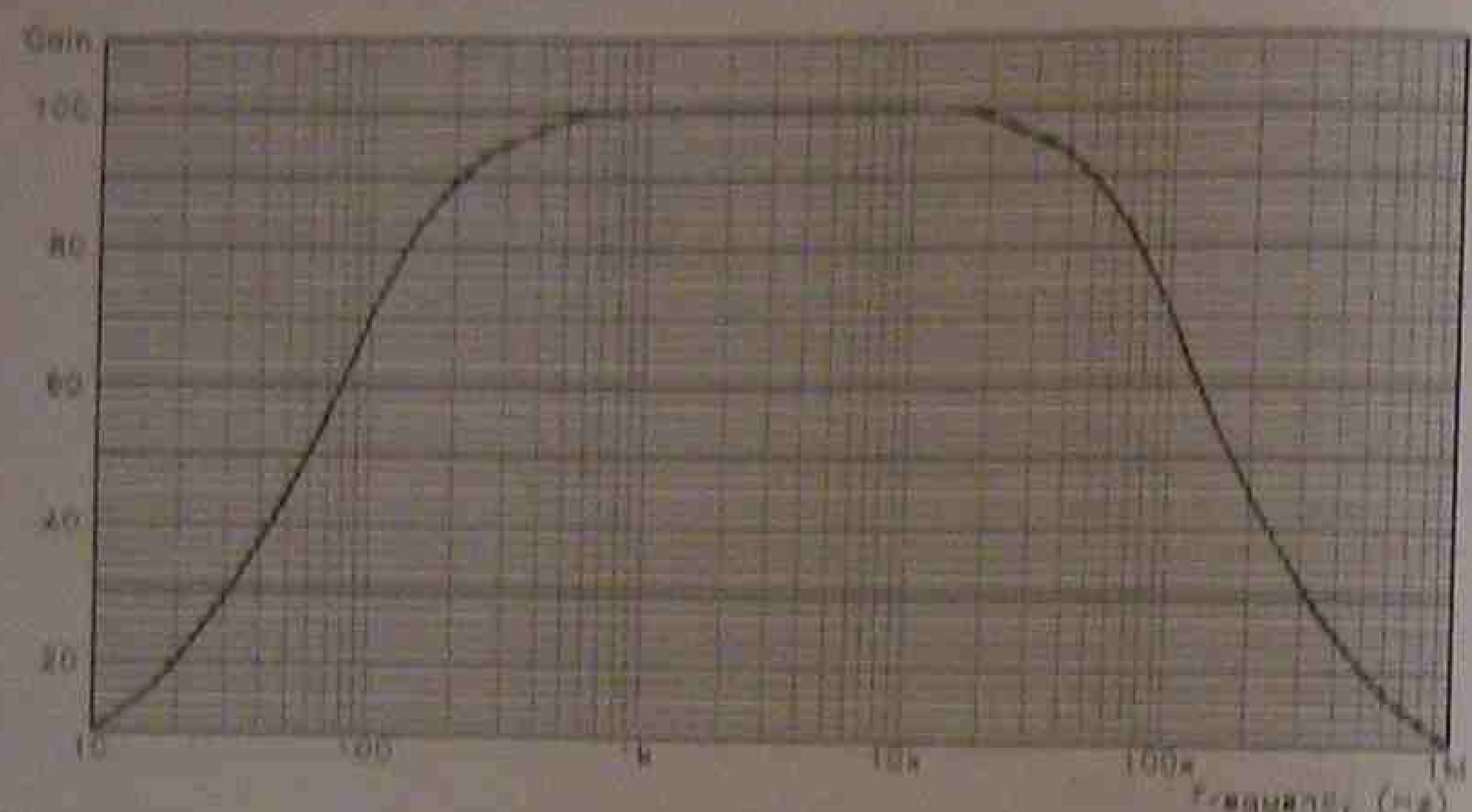
Frequency (Hz)	V_i (mV)	V_O (V)	Voltage Gain	Voltage Gain (dB)	10Ω load power(W)	power(dB) 1mW=0dB
10	100	1				
20	100	1.96				
40	100	3.71				
100	100	7.07				
200	100	8.94				
400	100	9.28				
1k	100	9.95				
2k	100	10				
4k	100	10				
10k	100	9.95				
20k	100	9.81				
40k	100	9.28				
100k	100	7.07				
200k	100	4.47				
400k	100	2.43				
1M	100	1				

Drill Question 1

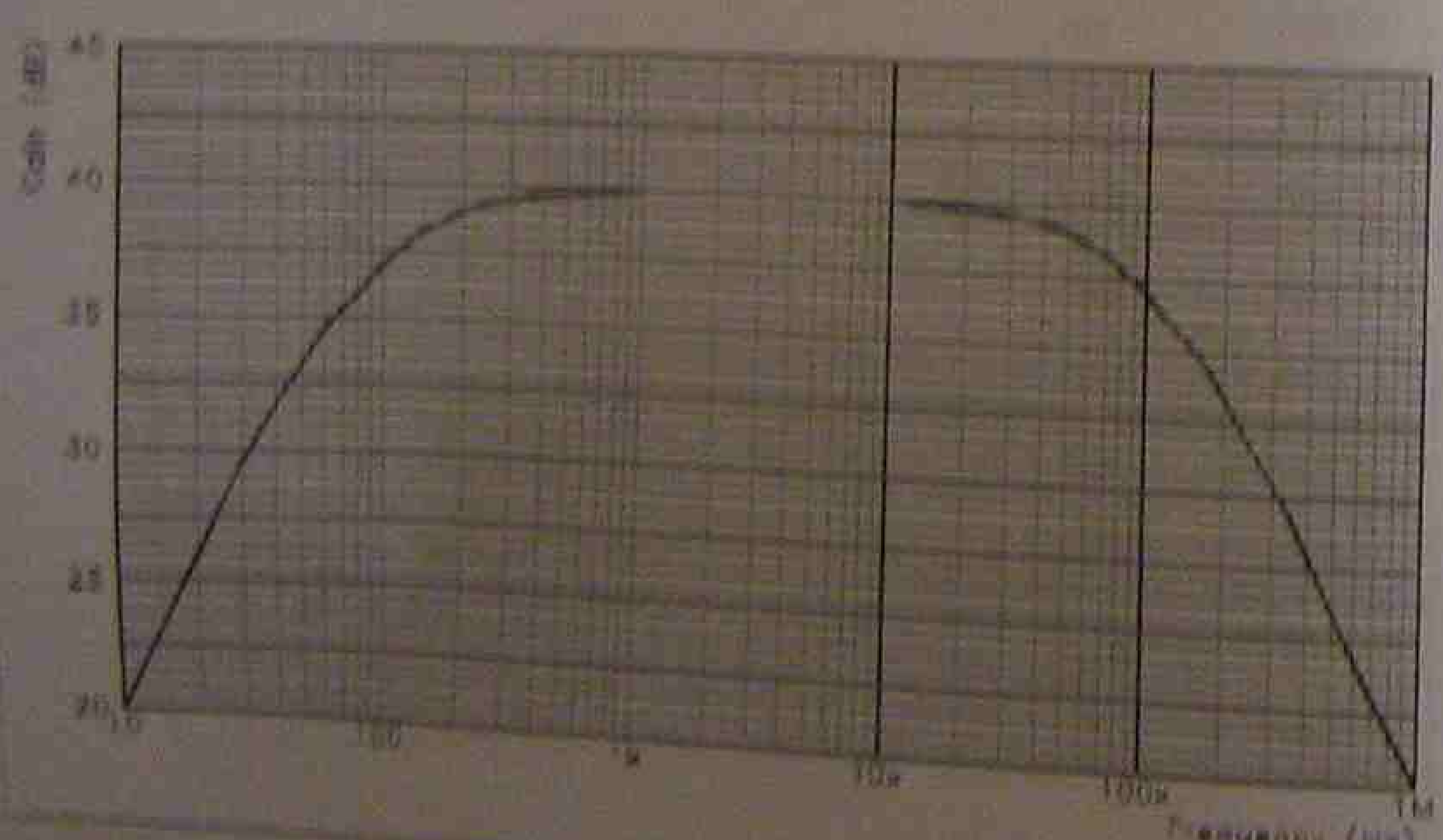
For Table 2.1, calculate voltage gain in direct ratio and in dB for the all measurements. Compare your calculation with frequency response plot in Figure 2.2. Also, calculate output power to the 10Ω load and show this output power in dB (1mW=0dB) on the graph in (b).

Figure 2.2 Frequency response curve

(a) Vertical axis in direct ratio



(b) Vertical axis in dB



Frequency response curves are normally plotted on semi-logarithmic graph paper as shown in Figure 2.2. For the horizontal axis, there is no zero at the extreme left hand side of the scale; that is, there is no zero origin. Also the scale is divided into equal steps that are actually a factor of 10 apart. For example, 10Hz and 100Hz are one decade apart and so are 100Hz and 1KHz. Therefore, using logarithmic scales helps us to create a more compact graph.

Drill Question 2

In Figure 2.2, determine voltage gain of the amplifier in both direct ratio and dB for the frequencies:

- 30Hz
- 80Hz
- 500Hz
- 6kHz
- 600kHz

2.3 Bandwidth

Drill Question 3

From table 2.1 and drill question 1, mid-band voltage gain is measured at 40dB and mid-band output power is measured at 10W. Determine:

- **Half output power (5w) frequencies.**
- **Voltage gain (dB) for half power frequencies.**
- **Voltage gain difference between mid-band frequencies and half power frequencies.**
- **Power gain difference between mid-band frequencies and half power frequencies.**

Half-power (or -3dB) Frequencies

If you have completed Drill Question 1~3, you noticed that at 100Hz and at 100kHz the power is half that developed at the mid-band frequencies. We used to call these two frequencies the **half-power frequencies**.

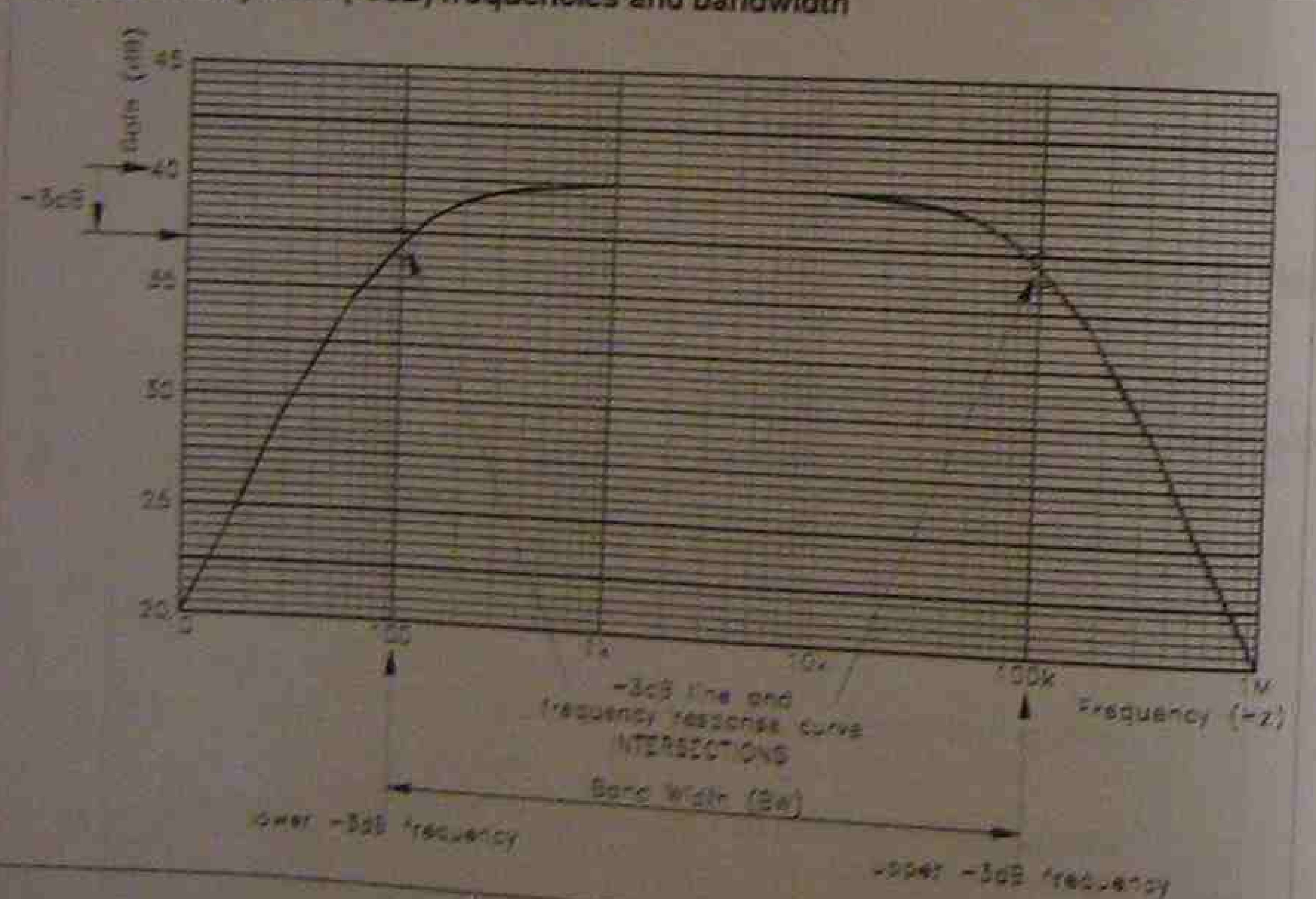
You also noticed that, regardless of voltage gain or power gain, there is a 3dB gain difference between mid-band frequencies and half-power frequencies. Because of this 3dB difference, half-power frequencies are sometimes called -**3dB frequencies**.

Bandwidth

There are two half-power frequencies as shown in Figure 2.3.

- The lower half-power (-3dB) frequency, $f_{L_{-3dB}}$. This is the lower frequency limit.
- The upper half-power (-3dB) frequency $f_{U_{-3dB}}$. This is the upper frequency limit.

Figure 2.3 Half-power (-3dB) frequencies and bandwidth



Bandwidth (BW) is determined as

$$BW = f_{U_{-3dB}} - f_{L_{-3dB}}$$

Drill Question 4

From Figure 2.3, determine the bandwidth.

Drill Question 5

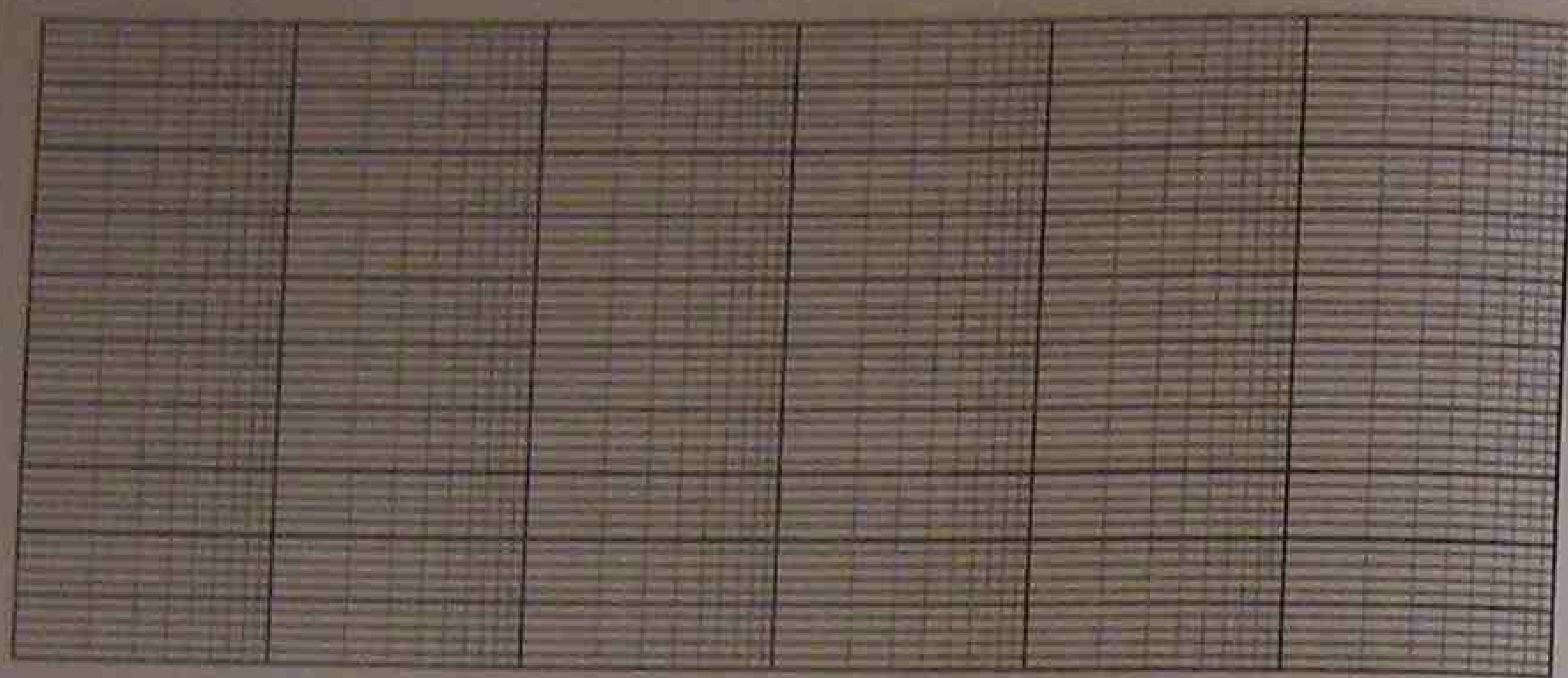
For a frequency response measurement result shown below:

Input voltage is 100mV for all frequencies

Frequency (Hz)	Output voltage (mV)	Gain (dB)
10	15.6	
20	62.4	
40	242.5	
100	842.3	
200	987.4	
1k	1000	
10k	1000	
20k	987.4	
40k	842.3	
100k	242.5	
200k	62.4	
1M	15.6	

(A) Calculate voltage gain in dB for all measured frequencies.

(B) Plot the frequency response curve.



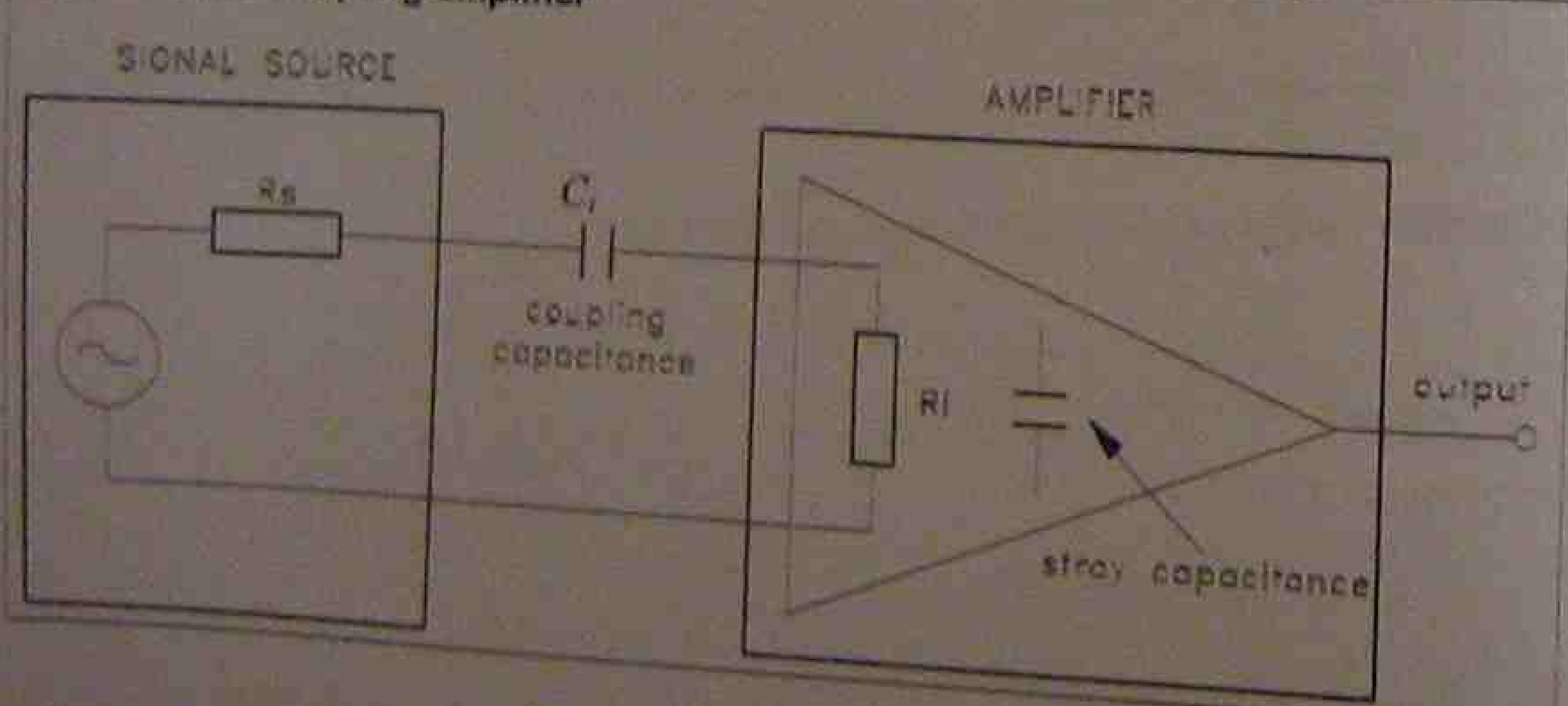
(C) Determine the half-power frequencies

(D) Determine the bandwidth of the amplifier.

2.4 Bandwidth and Capacitance

Ideally, an amplifier would be capable of amplifying an infinite number of frequencies: starting at 0Hz to ∞Hz. However, due to the reactance effect, mainly capacitive reactance, there is a frequency limitation. Figure 2.4 shows a typical RC coupled amplifier.

Figure 2.4 RC coupling amplifier



Lower Frequency Limit

In figure 2.4, the coupling capacitor is placed between the signal source and amplifier to avoid DC current flow between the two devices. The reactance of the capacitor, $X_C = \frac{1}{2\pi f C}$, increases as the frequency decreases. More of the signal source voltage is across C_i, therefore, and less is across R_i in the amplifier input. As a series circuit, R_S, C_i and R_i divide the ac signal voltage. The lower half-power frequency is defined:

$$f_{L_{-3dB}} = \frac{1}{2\pi(R_S + R_i)C}$$

Higher Frequency Limit

In an amplifier, the higher frequency limit is mainly due to the shunting effect of the stray, distributed capacitors. The total parallel capacitance is typically 10–40pF. Even that small amount of C can bypass high frequencies since the reactance of a capacitor decreases as the frequency increases.

Drill Question 6

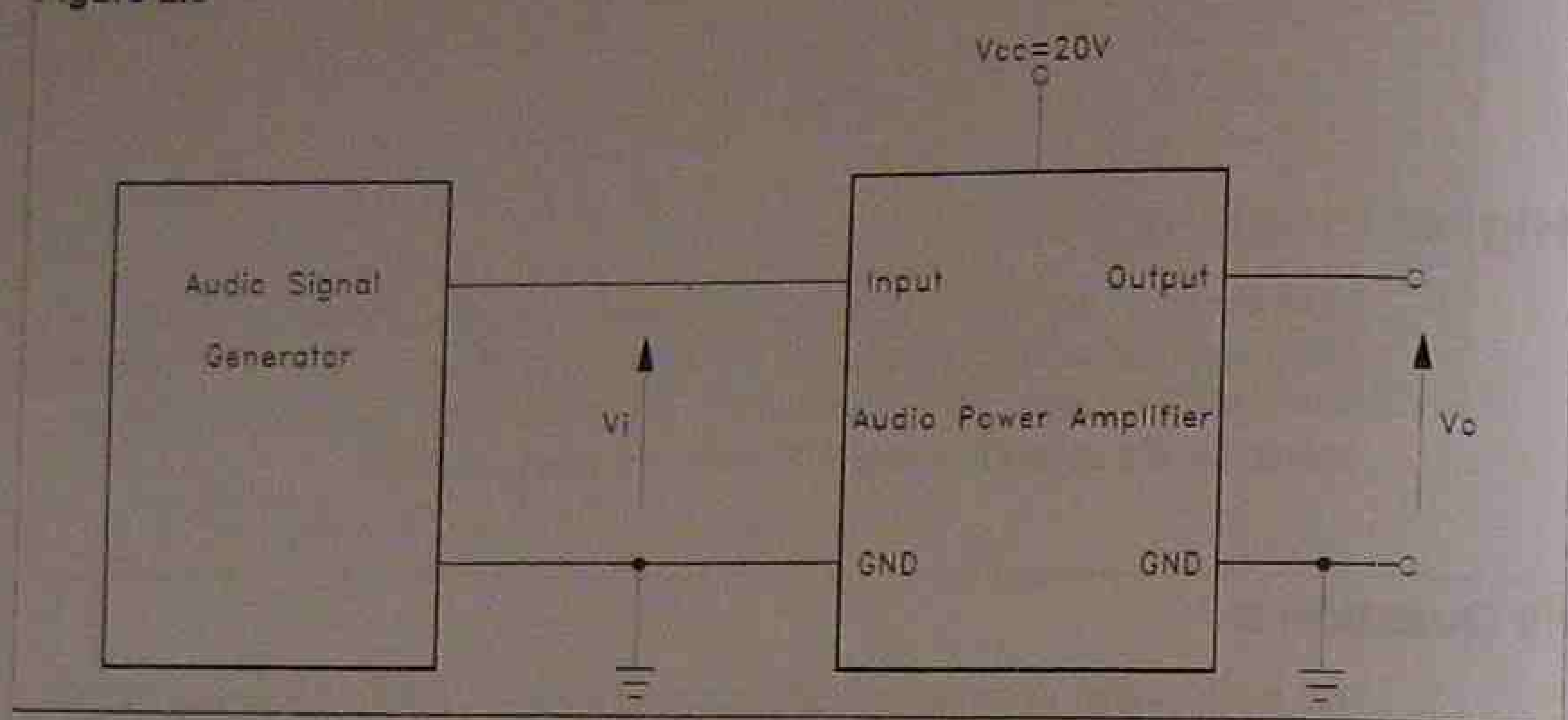
An audio signal generator whose source resistance is 600Ω is connected to an amplifier which has a 3kΩ input resistance. A 1μF coupling capacitor is employed between the signal source and the amplifier. Determine the lower half-power frequency

An amplifier has an input impedance of 10kΩ. Calculate the value of coupling capacitor required for 40Hz lower -3dB frequency.

Skill Practice 2

1. Construct the circuit shown below

Figure 2.5



2. Connect the oscilloscope CH1 to the input of the amplifier and CH2 to the output of the amplifier.

3. Measure the input and output voltages for the frequencies shown in Table 2.2. Recommended input voltage is 100mVpp. However, output voltage waveform should not be clipped or distorted.

4. Calculate the voltage gain in direct ratio and dB for all the measurements.

5. Determine the mid-band voltage in direct ratio.

6. Calculate the voltage gain in direct ratio for the half-power frequencies.

7. Predict the approximate half-power frequencies using table 2.2 and measure the exact half-power frequencies.

Lower half-power frequency =

Upper half-power frequency =

Table 2.2

Frequency (Hz)	Figure 2.5				Figure 2.6			
	Input voltage (mv)	Output voltage (V)	Voltage Gain		Input voltage (mv)	Output voltage (V)	Voltage Gain	
			ratio	dB			ratio	dB
10								
20								
40								
80								
100								
200								
400								
800								
1k								
2k								
4k								
8k								
10k								
20k								
40k								
80k								
100k								
200k								
400k								
800k								
1M								

8. Plot the frequency response using Table 2.2. Clearly indicate two half-power frequencies.

Voltage gain in direct ratio

Voltage gain in dB

9. Determine:

(A) Mid-band voltage gain.

(B) Bandwidth of the amplifier.

10 Apply a 100mVpp square wave for the frequencies shown below and sketch the output waveforms.

100Hz

1kHz

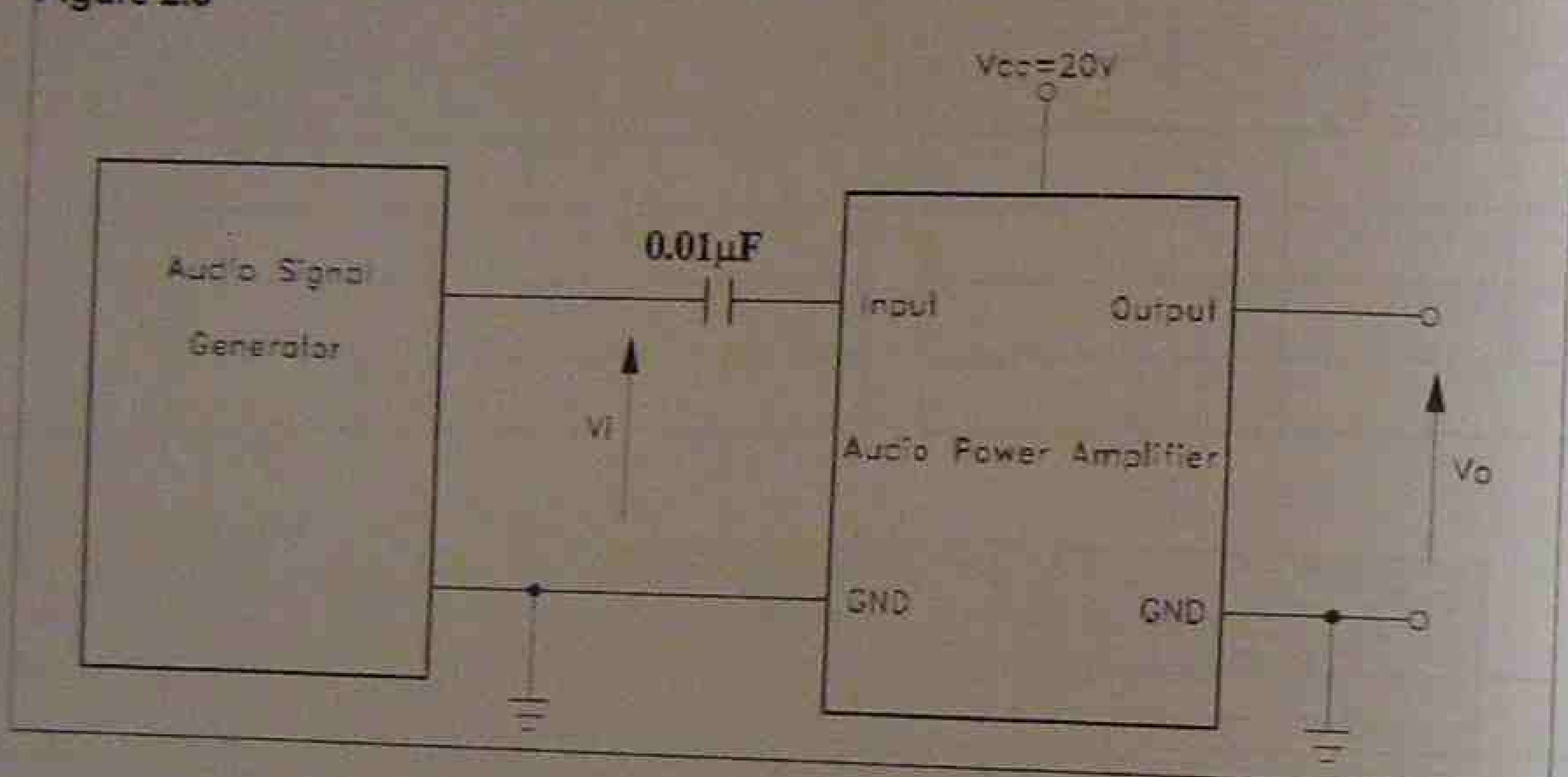
10kHz

100kHz

11. Explain how square wave response could be used to estimate the frequency response of the amplifier.

12. Connect a 0.01μF capacitor as shown in Figure 2.6 and repeat **1~10**. Use page 2-12 graph to plot the frequency response.

Figure 2.6



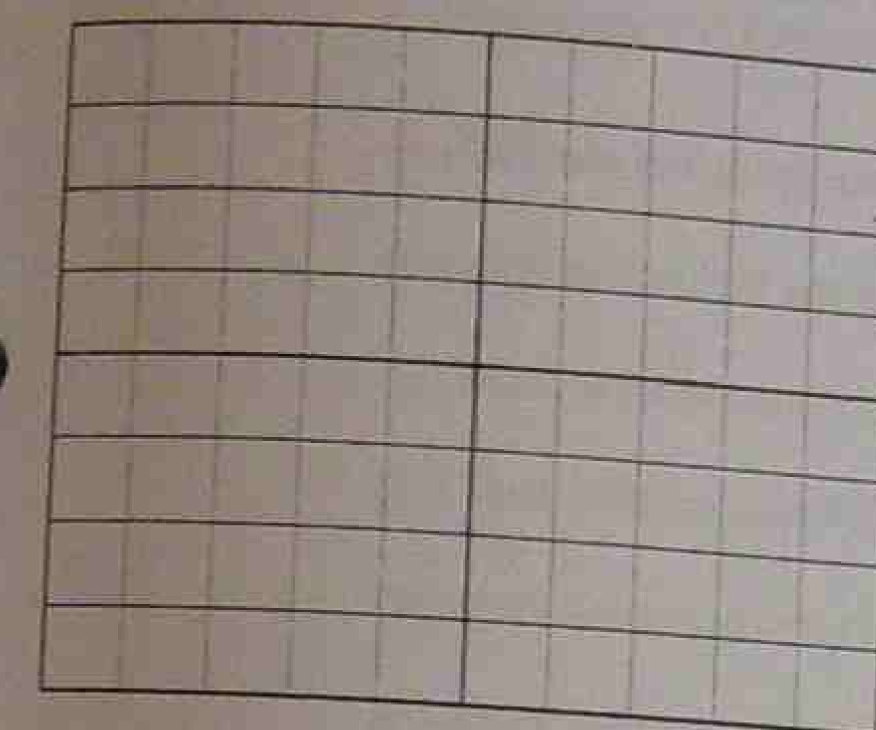
(A) half power frequencies are:

(C) Calculate the lower half-power frequency using previous input resistance measurement and compare this theoretical calculation with your measurement result. If there is any difference, discuss why.

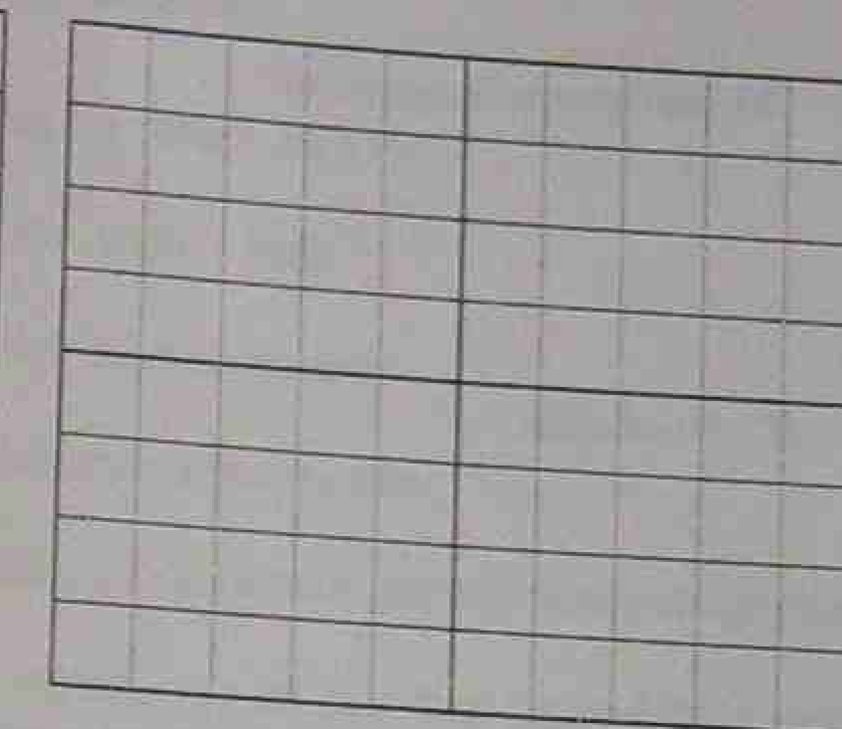
(C) Determine the bandwidth.

(D) Square wave response.

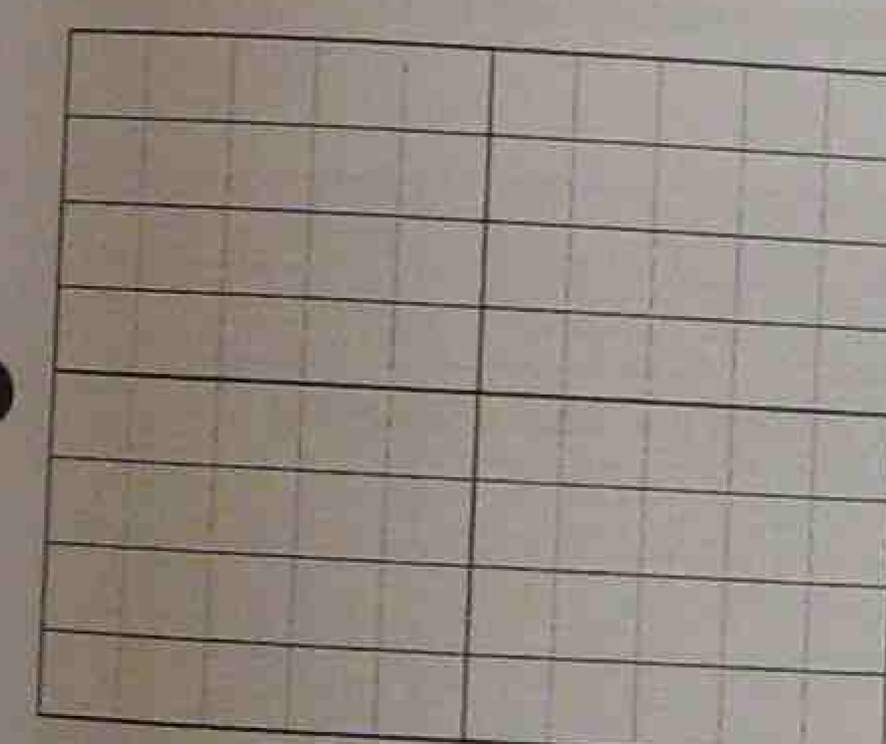
100Hz



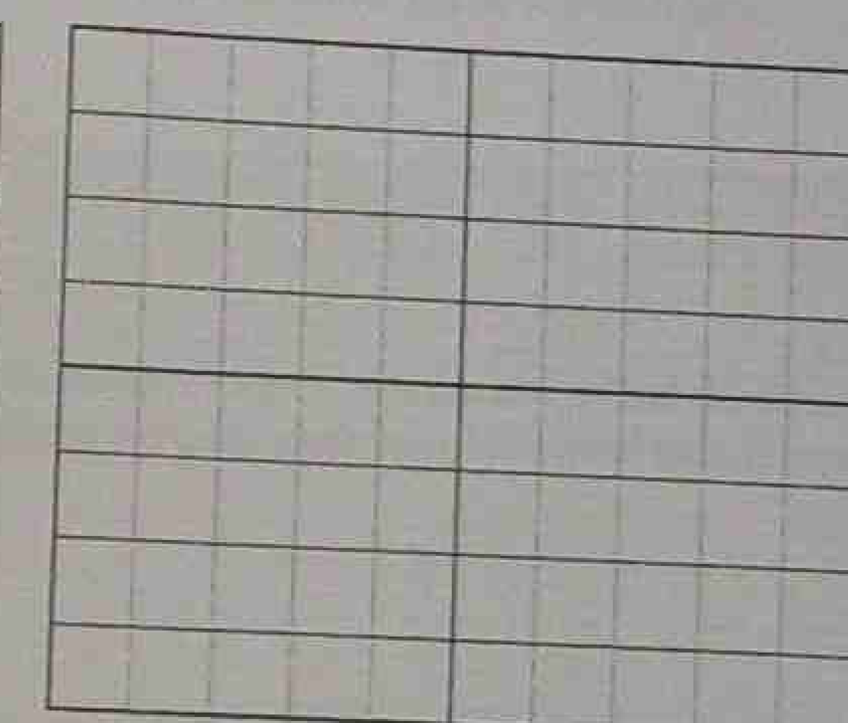
1kHz



10kHz



100kHz



Review Questions

1. Describe:

(A) Frequency response

(B) Half-power frequencies

(C) Bandwidth

2. Explain what is the main cause of frequency limitation of an amplifier.

3. An amplifier has a 100kΩ input resistance. Calculate the value of coupling capacitor to limit the lower frequency to 340Hz.

4. Draw a frequency response curve for:

- **Mid-band gain = 40dB**
- **Lower half-power frequency = 200Hz**
- **Upper half-power frequency = 10kHz**
- **Attenuation slope of frequencies below the lower half-power frequency = 6dB/oct or 20dB/dec**
- **Attenuation slope of frequencies above the upper half-power frequency = -12dB/oct or -40dB/dec.**

(A) Indicate horizontal scales for 10Hz ~ 100kHz.

(B) Indicate vertical scales for -50dB ~ 50dB

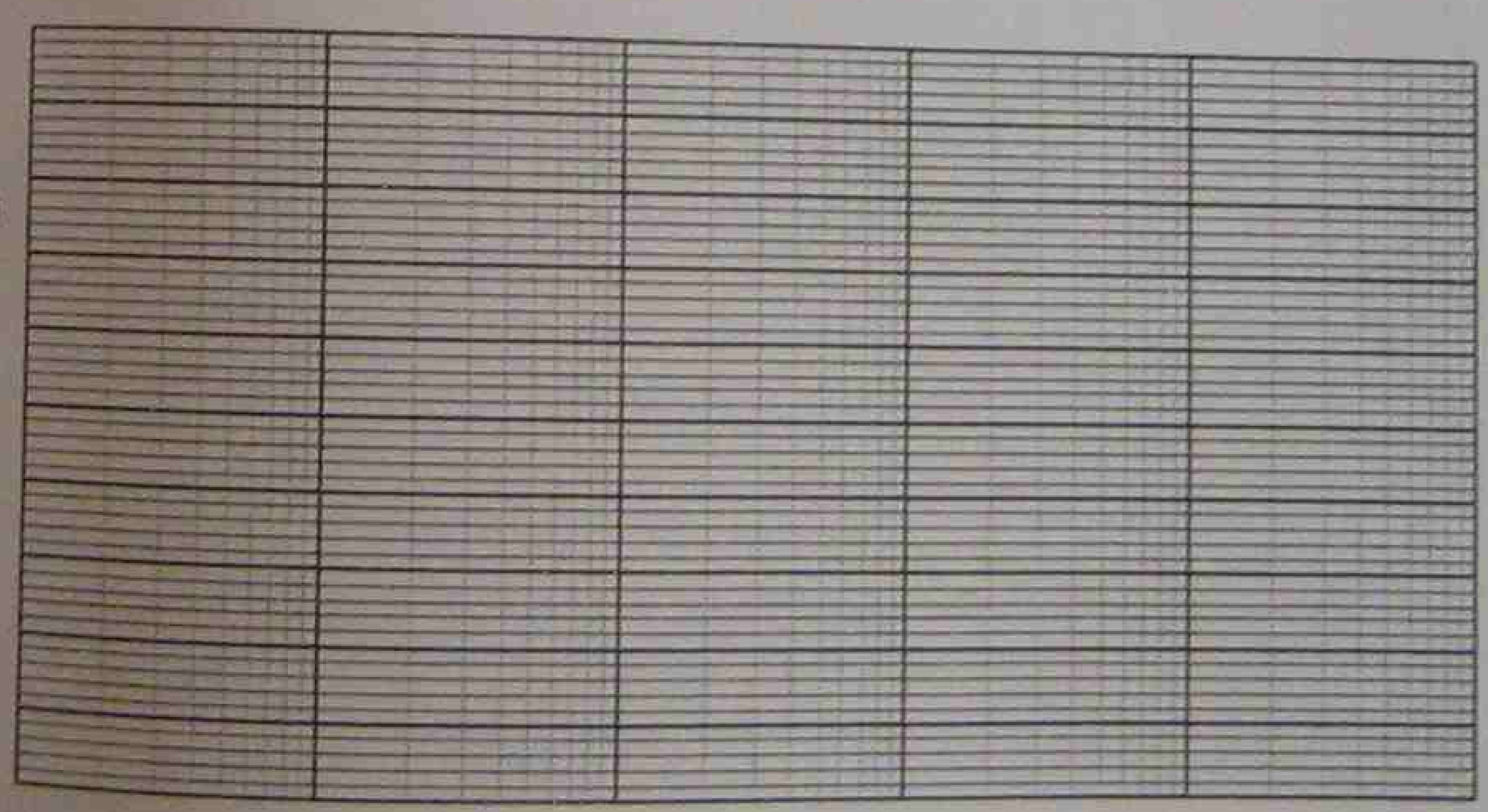
(C) Mark voltage gain for mid-band frequencies.

(D) Mark voltage gain for half-power frequencies.

(E) Mark voltage gain for 20Hz. (20dB less than the mid-band voltage gain)

(F) Mark voltage gain for 100kHz. (40 dB less than the mid-band voltage gain)

(G) Plot frequency response curve on the graph paper shown below.



5. From question 4, determine the voltage gain at:

(A) 10Hz

(B) 50Hz

(C) 40kHz

3

Operational Amplifiers

Upon completion of this chapter, you should be able to:

- Describe the brief history of the op-amp.
- Explain an op-amp symbol.
- Identify the names of terminal in op-amp symbols.
- Know how to connect the power supply for op-amps.
- Explain the difference between the inverting input and non-inverting input of an op-amp.
- Describe an op-amp equivalent model.
- Describe the major characteristic difference between the ideal op-amp and the practical op-amp.
- Explain what a voltage comparator is.
- Design a simple voltage comparator.
- List the applications of a voltage comparator.
- Construct op-amp application circuits.
- Describe PWM (pulse width modulation).

3.1 What is an Operational Amplifier (op-amp)?

Early operational amplifiers (op-amps) were used to perform mathematical operations such as addition, subtraction, integration, and differentiation in analogue computers - hence the term operational. These early devices were constructed with vacuum tubes and worked with high voltages. Today's op-amps are built as a single integrated circuits (the first practical version was τηε μA709 by Fairchild Semiconductor in 1965) that use low voltages and are reliable and inexpensive. Figure 1.6 and 1.7 show typical op-amp IC packages and an internal circuit diagram.

Symbol and Terminals

Figure 3.1 Op-amp symbol

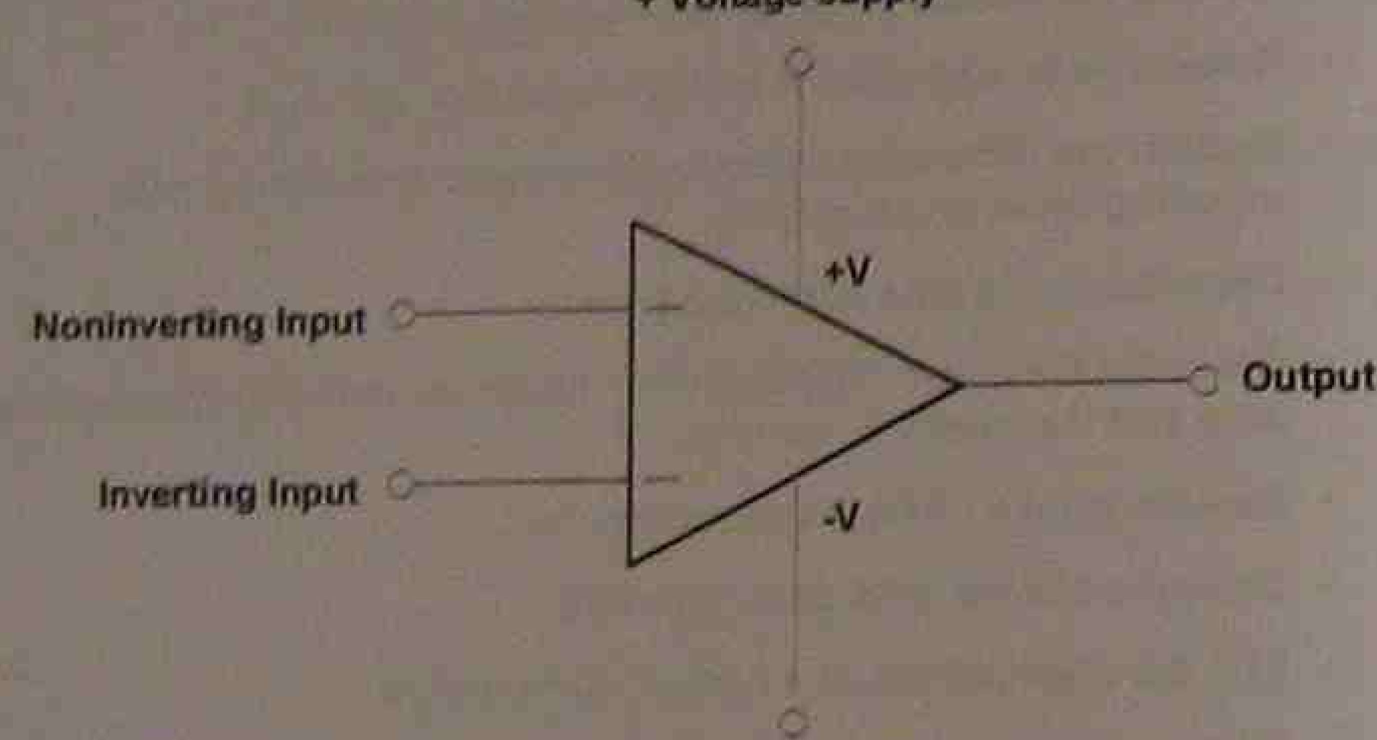
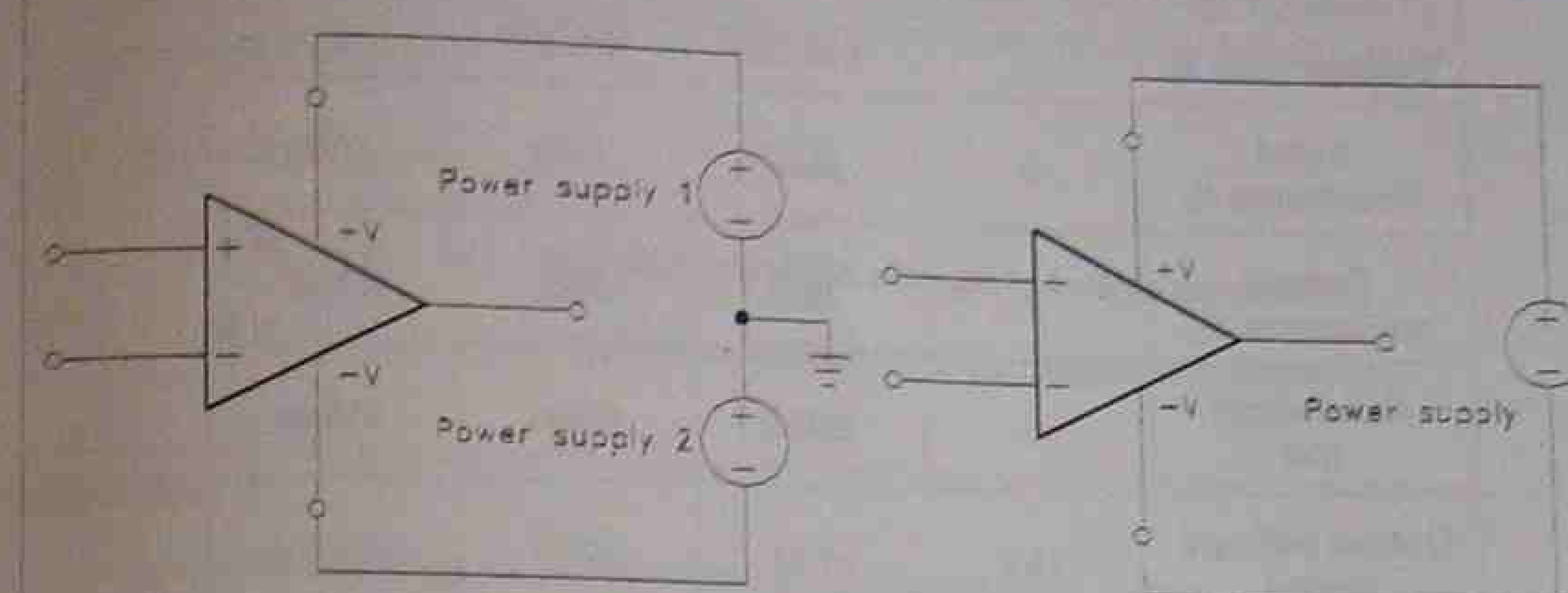


Figure 3.1 shows the standard op-amp symbol. Firstly, it has two input terminals, called the noninverting (+) input and the inverting (-) input. It amplifies the voltage difference between noninverting input and inverting input. The input names are from the polarity (phase) difference between input terminal and output terminal. If the + input voltage is greater than the - input voltage, the polarity of the output voltage is positive. If the - input voltage is greater than the + input voltage, the polarity of the output voltage is negative. Therefore there is no voltage polarity difference between the + input terminal and the output terminal and the + input terminal is named **noninverting input**. However, the output voltage polarity is the opposite of the input voltage polarity - hence the - input terminal is named **inverting input**.

The typical op-amp operates with two dc supply voltages as shown in Figure 3.2. One positive and the other negative, hence we need two power supplies instead

of one. Usually these voltage terminals are left off the schematic symbol for simplicity but are always understood to be there.

Figure 3.2 Typical op-amp with two power supplies.

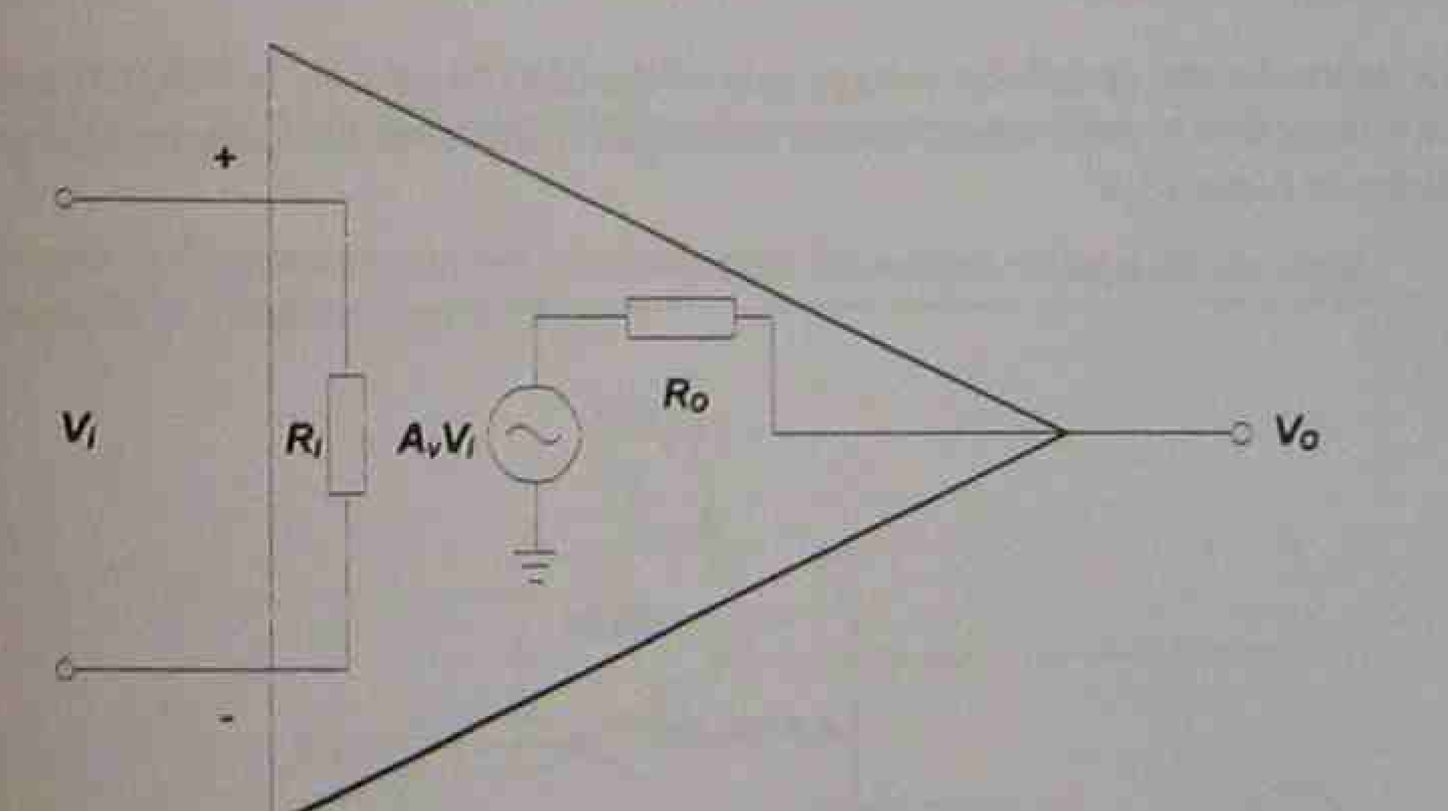


(a) CORRECT power supply wiring (b) WRONG power supply wiring

Comparison of the Ideal and the Practical op-amp

As seen from chapter 1, the op-amp can be expressed with an amplifier equivalent model since it is an amplifier.

Figure 3.3 A simple op-amp equivalent model.



V_i: Voltage difference between + input terminal and - input terminal (differential input voltage)
R_i: resistance seen from two input terminals (differential input resistance)
A_V: open loop voltage gain (differential mode voltage gain)
R_O: output resistance

Table 3.1 Basic characteristic comparison between ideal op-amp and practical op-amp (±15V supplied voltage, R_L = 2kΩ)

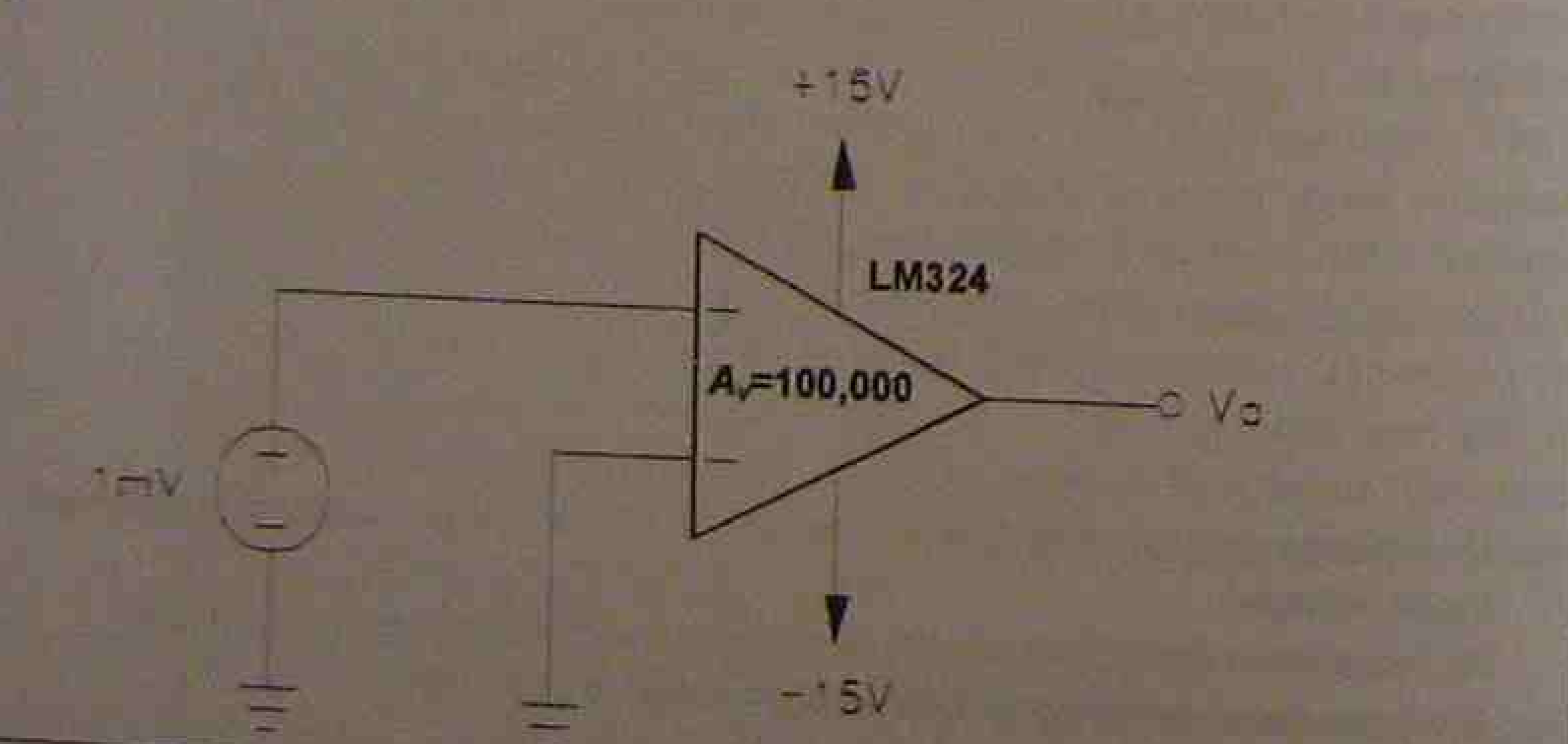
Characteristics	Ideal op-amp	Practical op-amp		
		LM741	LM 324	LF351
Voltage gain A_v	∞	200,000	100,000	100,000
Input resistance R_i	∞	2MΩ	1MΩ	1TΩ
Output resistance R_O	0	75Ω	75Ω	270Ω
Bandwidth BW	∞	2MHz	1MHz	4MHz
Output voltage swing	±15V	±13V	±13V	±13.5V

3.2 Comparators

Operational amplifiers are frequently used to compare the amplitude of one voltage with another. In this application, the op-amp is used in the open-loop configuration, with the input voltage on one input and a reference voltage on the other.

For example, the open loop voltage gain of the LM324 op-amp is 100,000 and 1mV is applied to the + input terminal and negative input terminal is grounded as shown in Figure 3.4.

Figure 3.4



The output voltage you would expect is:

$$V_O = A_V V_i = 100{,}000 \times 1mV = 100V$$

We know that this is impossible because supply voltages go to only 15V. Therefore, the op-amp output voltage will be saturated. In an ideal situation V_O can reach 15V, but in a practical circuit V_O can get to about 13V with 2kΩ load to the output terminal.

For an ideal op-amp:

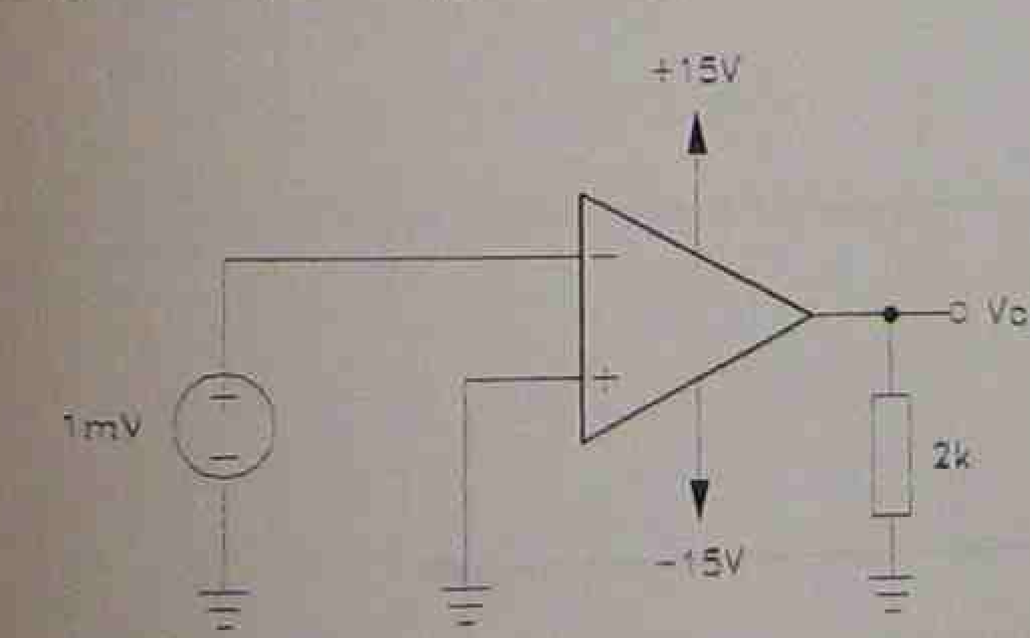
- If $V_{i+} > V_{i-}$, V_o = positive supplied voltage (positive saturation)
- If $V_{i+} < V_{i-}$, V_o = negative supplied voltage (negative saturation)

Drill Question 1

For the circuit diagram 3.4, calculate the minimum input voltage to saturate the op-amp.

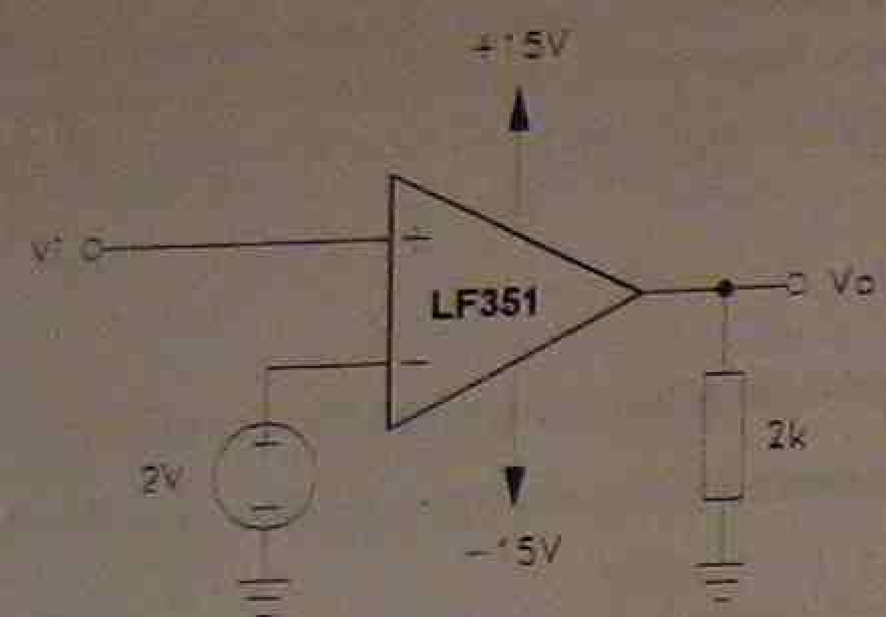
Drill Question 2

For the circuit diagram shown below, determine the output voltage for an ideal op-amp, LM741, LM324, and LF351.



Drill Question 3

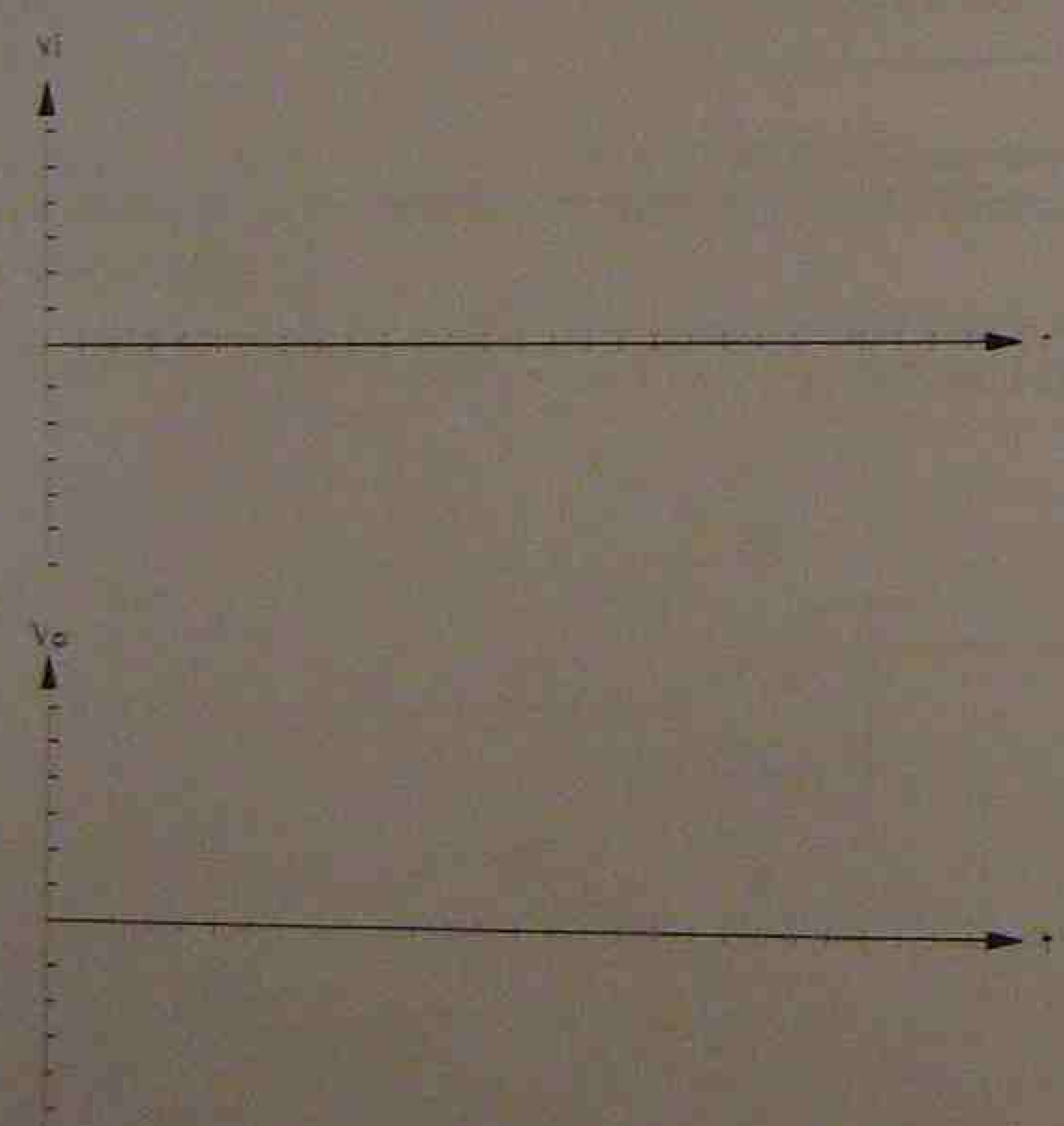
For the circuit diagram shown below, determine the output voltages for the input voltages shown in table below.



V_i (V)	-10	-5	0	1.999	2.001	5	10
V_o (V)							

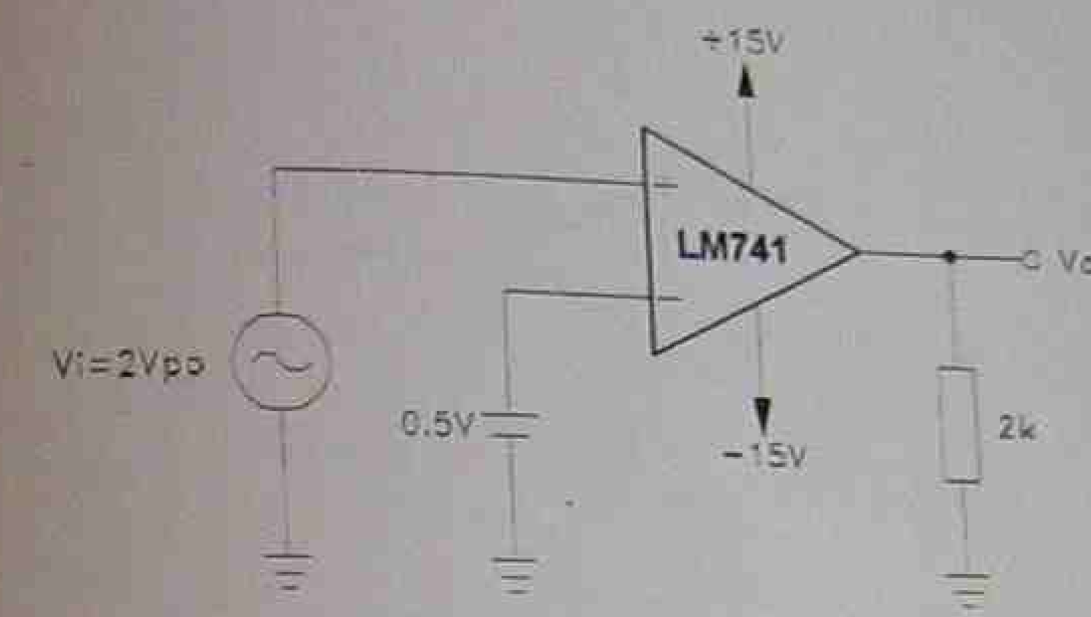
Drill Question 4

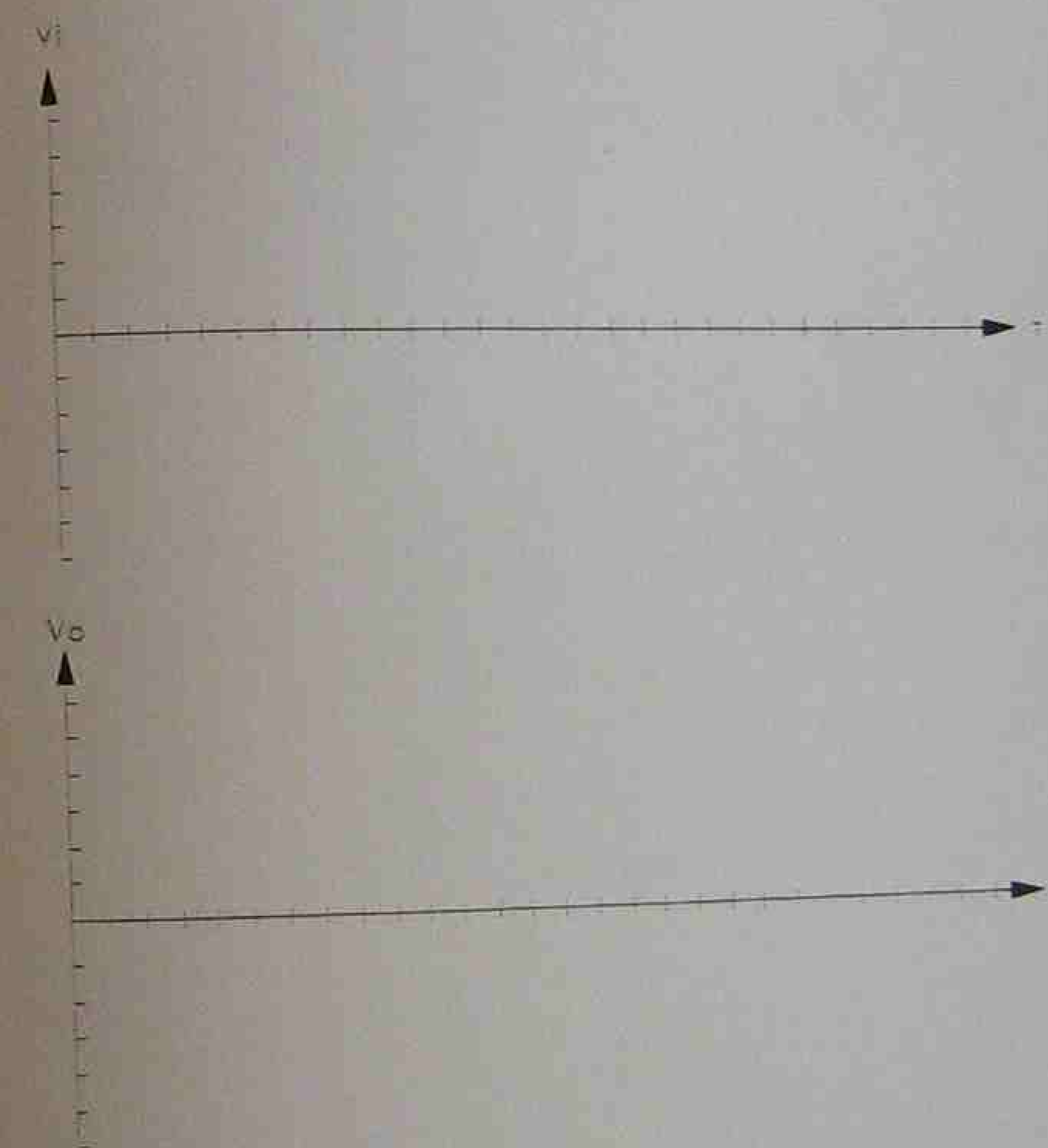
For the circuit diagram in Figure 3.4, the 1mV dc voltage source is replaced with 2Vpp 1kHz ac signal. Draw both input and output waveforms.



Drill Question 5

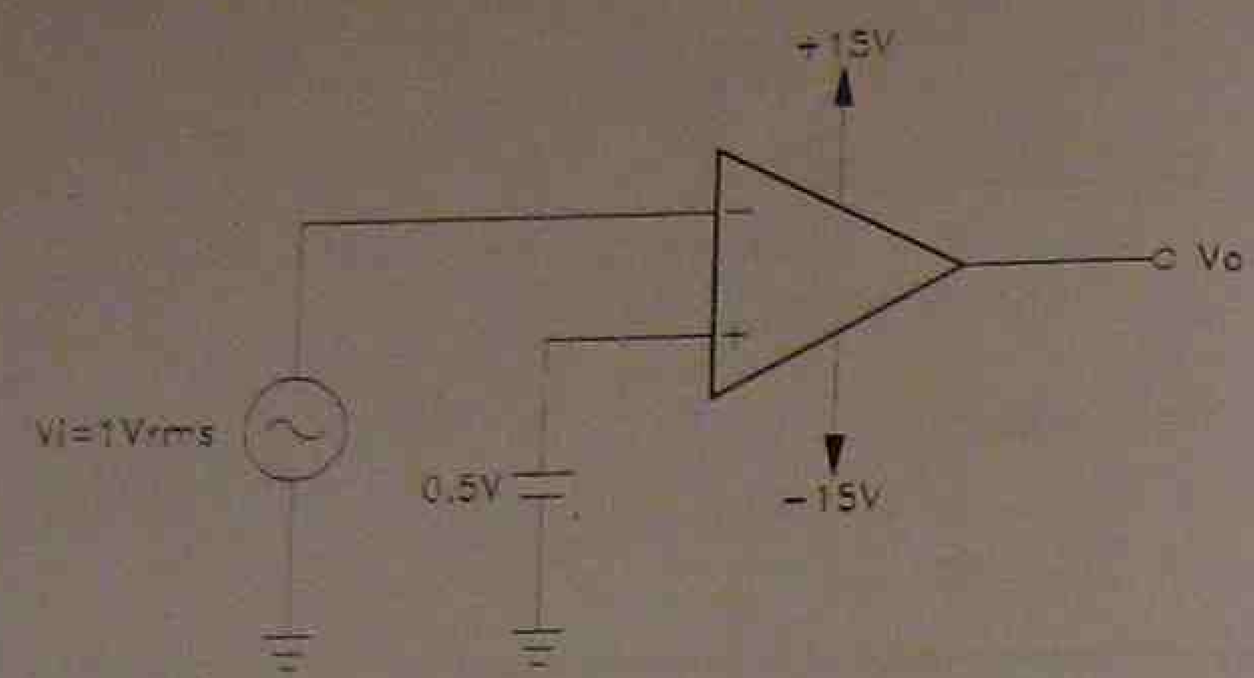
For the circuit diagram shown below, draw both input and output waveforms.

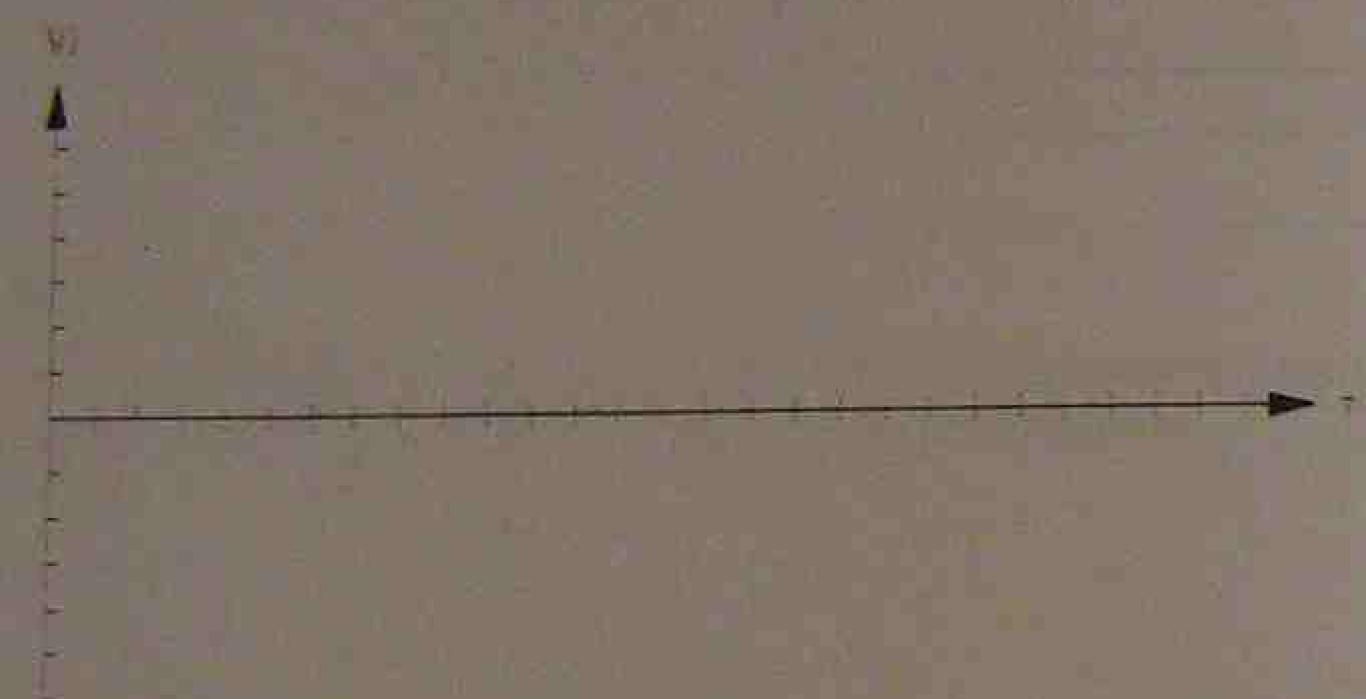


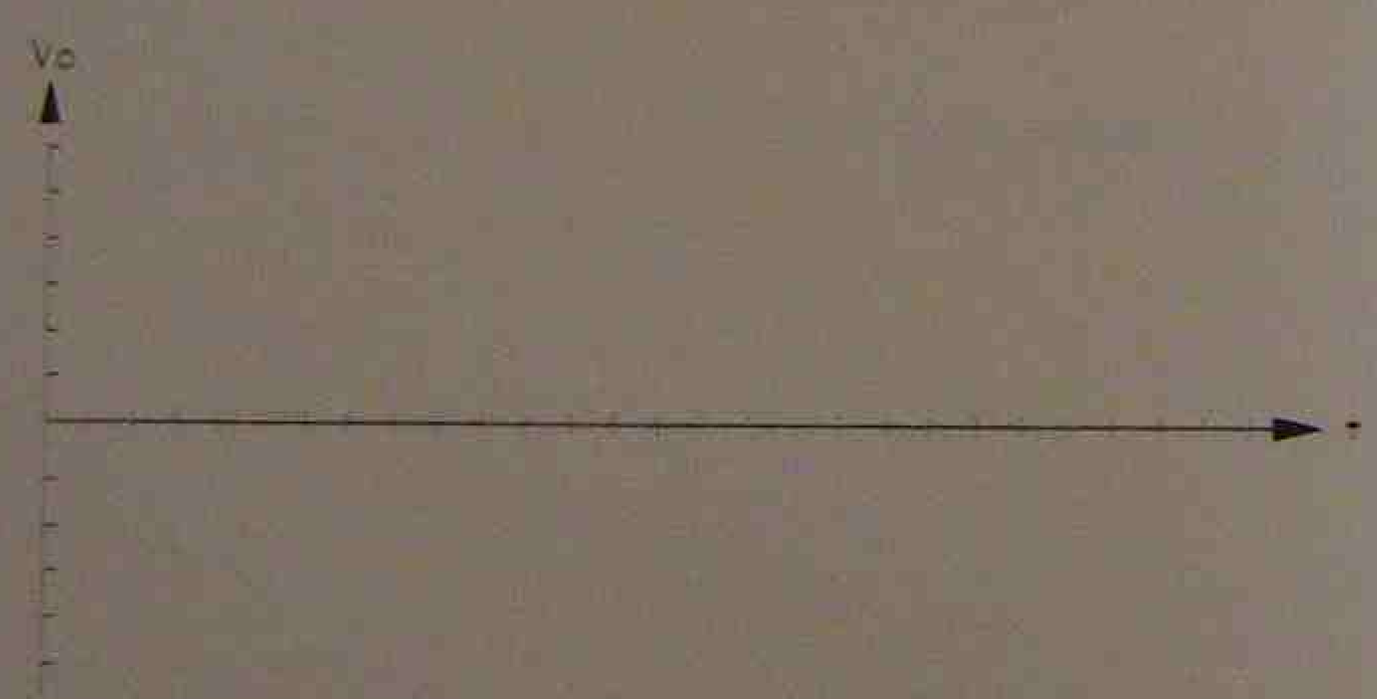


Drill Question 6

For the circuit diagram shown below, draw both input and output waveforms.

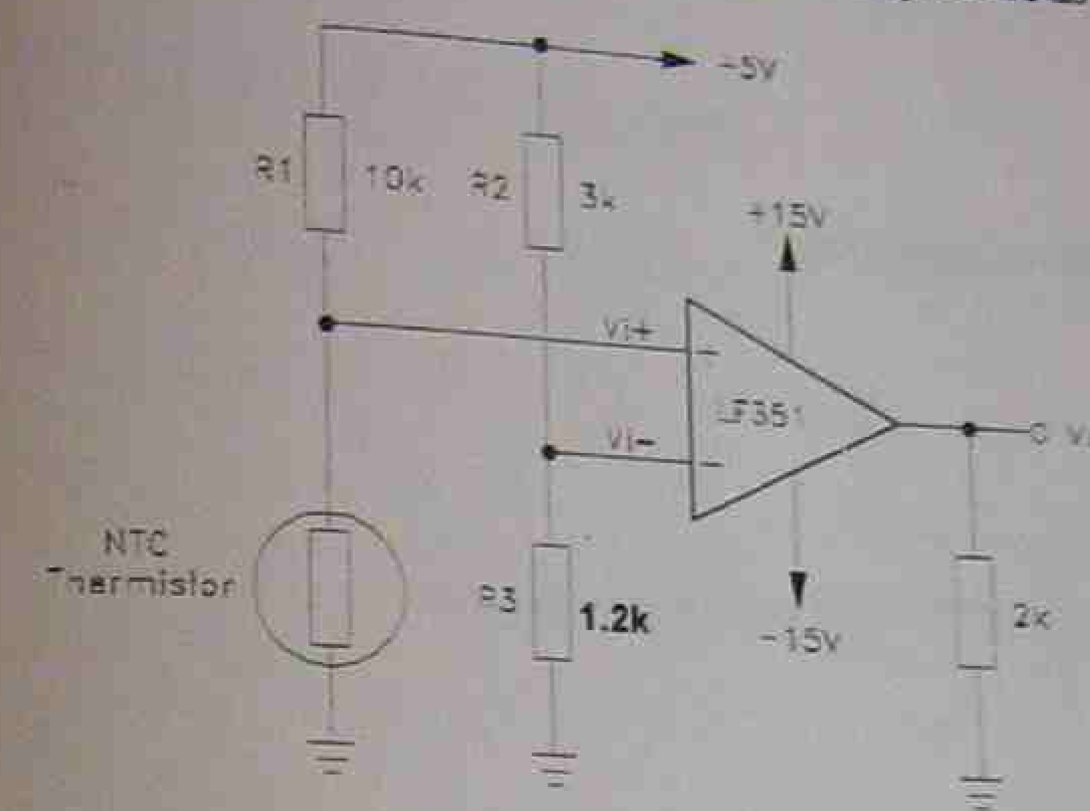






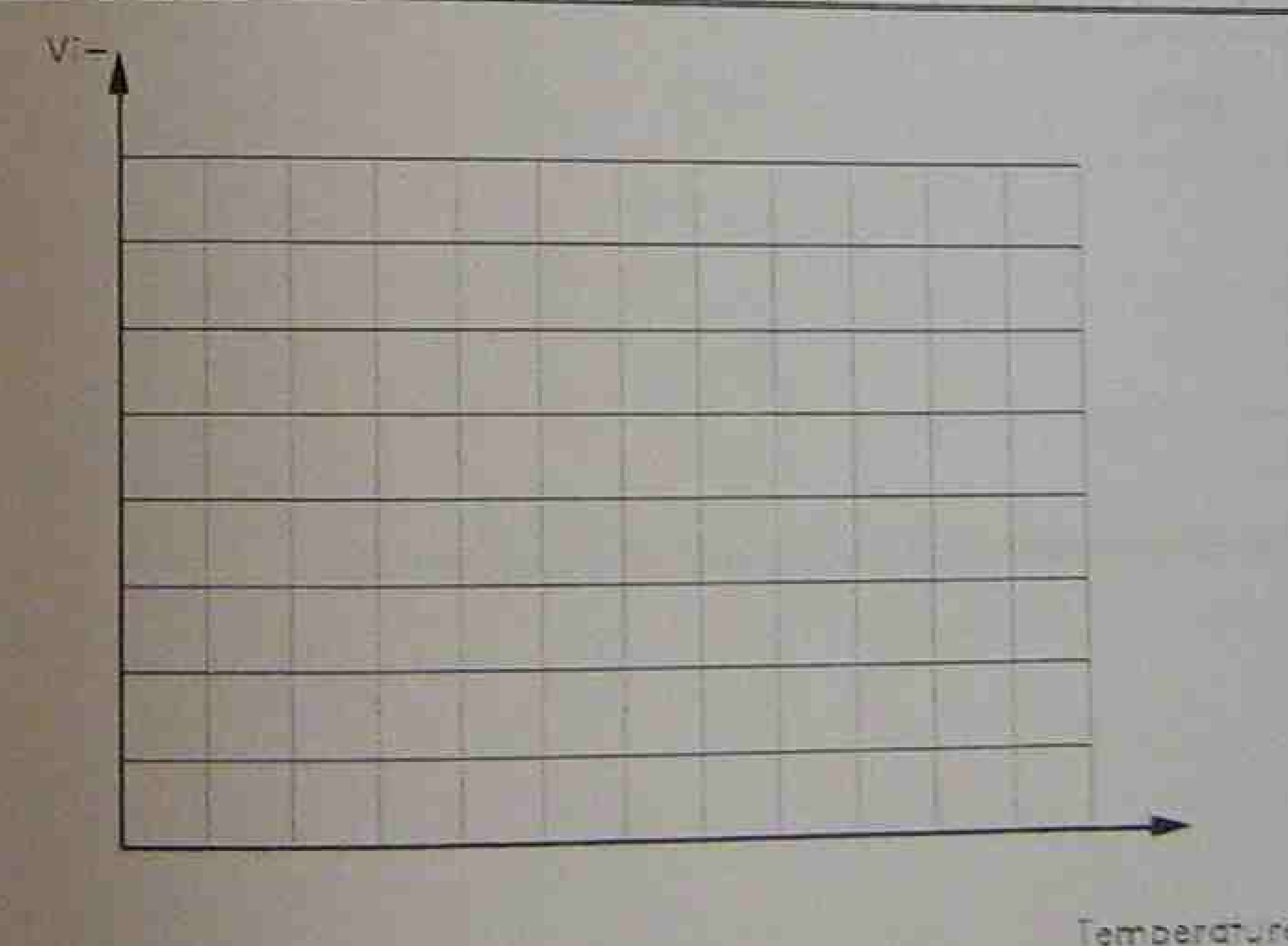
Drill Question 7

An NTC thermistor is employed to control the temperature as shown below.



(a) The resistance of the NTC thermistor is measured as shown below. Calculate V_{i+} for each of the temperature shown in table below. Also, draw the V_{i+} - temperature graph.

Tempera ture (°C)	20	25	30	35	40	45	50	55	60	65	70	75	80
R (kΩ)	9.5	8.2	6.8	5.7	4.9	4.3	3.7	3.3	3	2.7	2.5	2.3	2.2
V_{i+}													



(b) Calculate inverting input voltage V_{i-}.

(c) Determine the transition temperature.

(d) Determine the value of R3 for 70°C transition temperature.

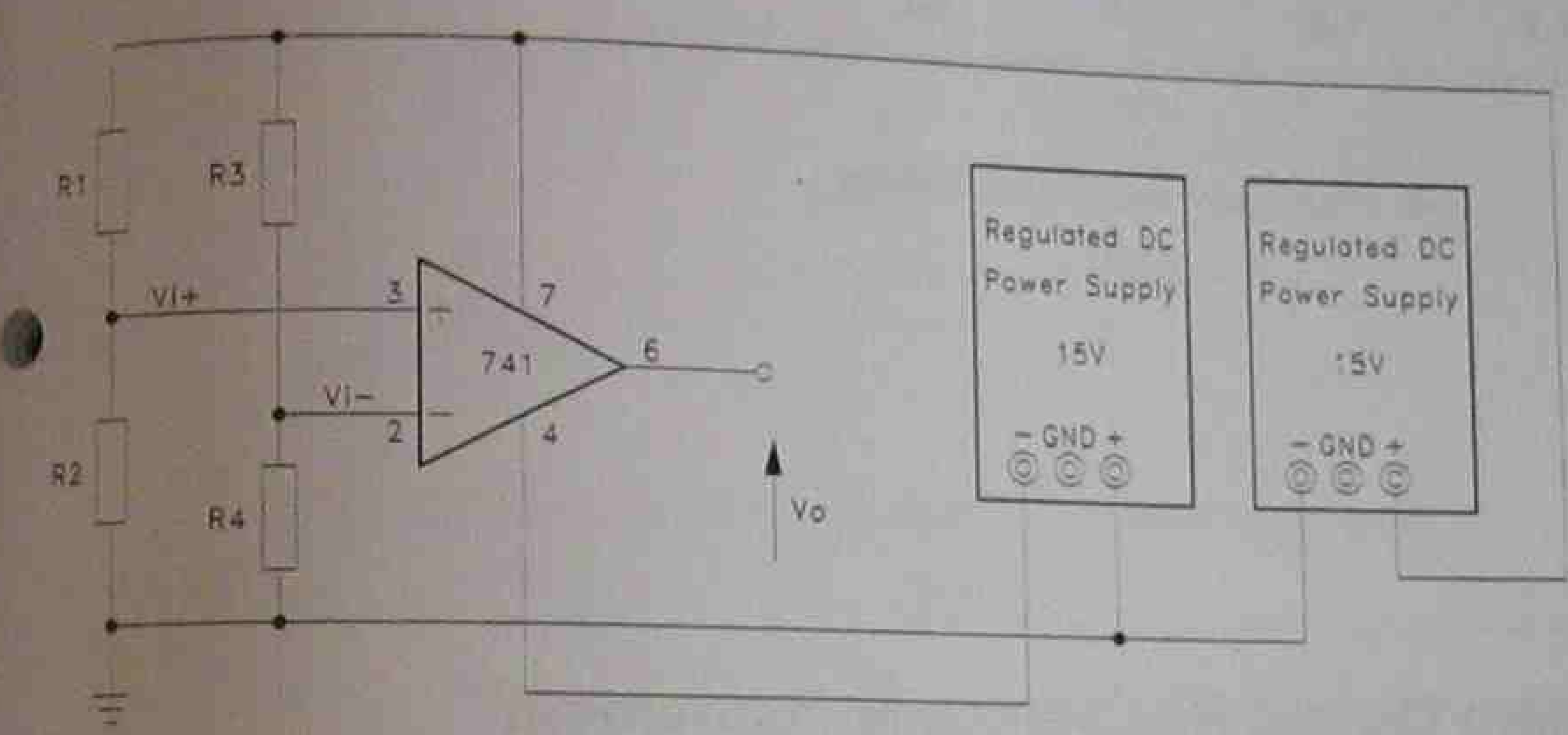
(e) (optional) Show a relay interface circuit to the output terminal to operate a 240V heating element.

(f) (optional) Is the relay interface circuit above safe? If not, explain why.

Skill Practice 3

OP amp Voltage Comparator

1. Construct the circuit shown below.



2. Calculate the V_{i+}, V_{i-} and determine the output voltage V_o for the R1 ~ R4 values shown below.

R1(k)	R2(k)	R3(k)	R4(k)	V_{i+}(V)	V_{i-}(V)	V_o(V)
1.2	2.2	2.2	1.2			
2.2	1.2	1.2	2.2			
1.2	1.2	2.2	2.2			

3. Measure the V_{i+}, V_{i-} output voltage V_o for the R1 ~ R4 values shown below.

R1(k)	R2(k)	R3(k)	R4(k)	V_{i+}(V)	V_{i-}(V)	V_o(V)
1.2	2.2	2.2	1.2			
2.2	1.2	1.2	2.2			
1.2	1.2	2.2	2.2			

4. Explain how the circuit is working.

OP amp Output Resistance Measurement

For R1=1.2k, R2=2.2k, R3=2.2k, R4=1.2k:

1. Measure the output voltage without load.

V_o =

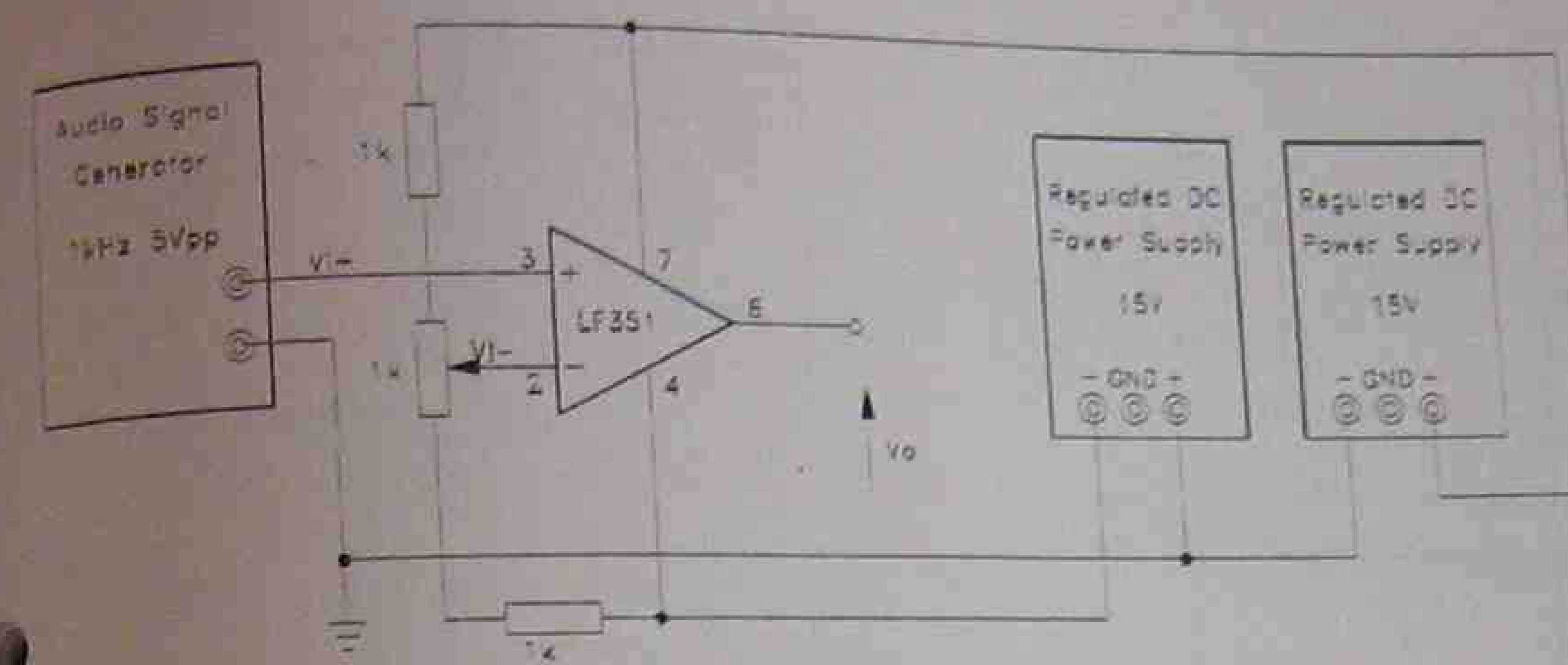
2. Connect 1.2k load between the output terminal (pin 6) and ground. Measure the output voltage (V_L) and calculate the output resistance using the equation in page 1-20.

Ro =

3. Compare with manufacturer's specification.

Pulse Width Modulation

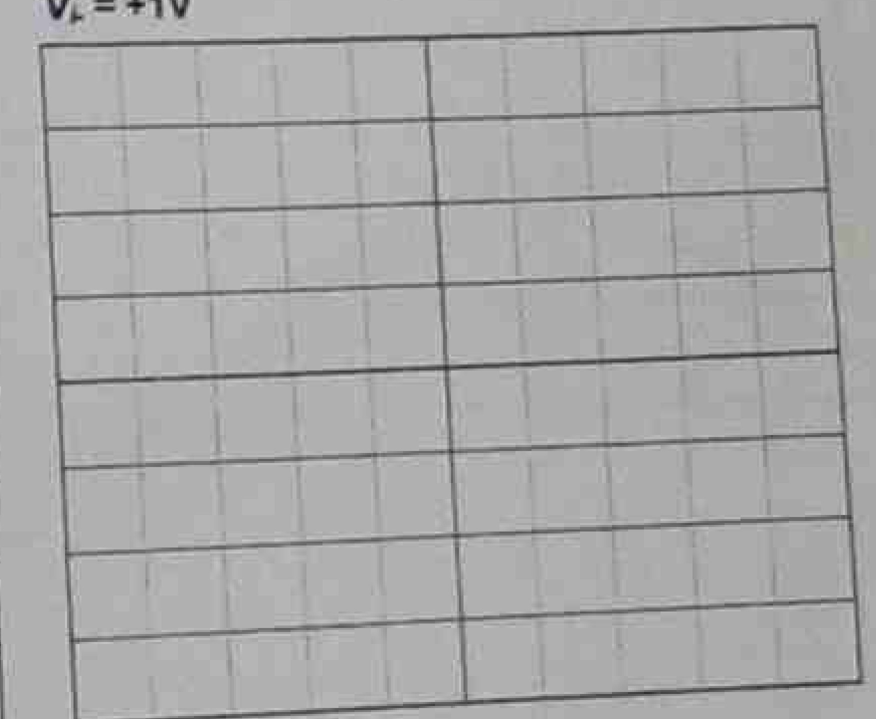
1. Construct the circuit shown below.



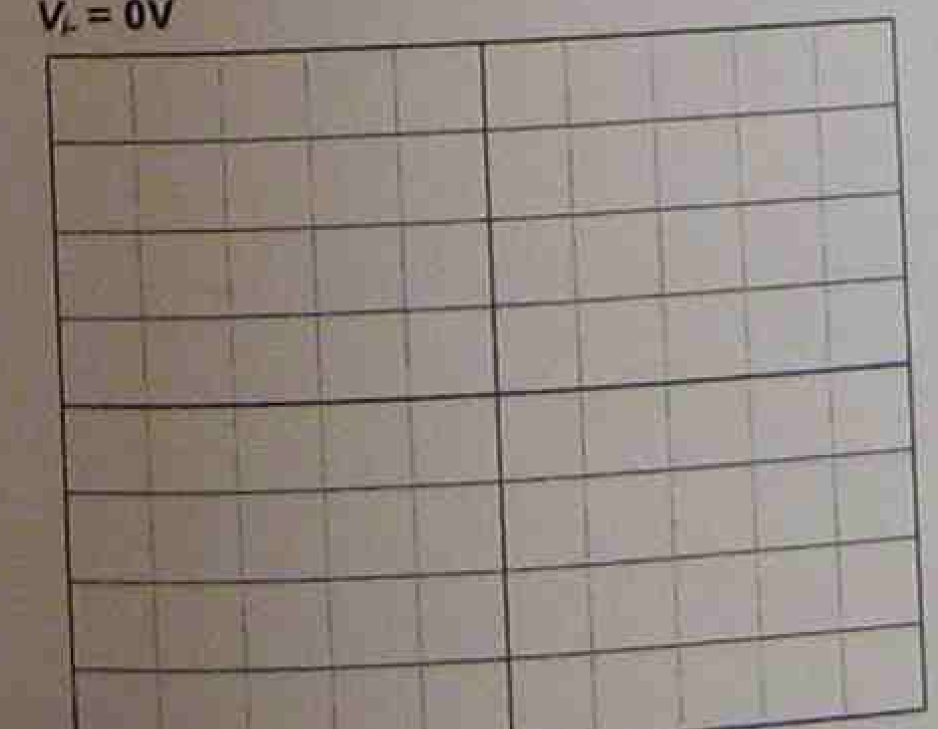
2. Sketch the V_{i+} and V_O waveforms for each of the V_{i-} shown below.

V_{i-} = +2V

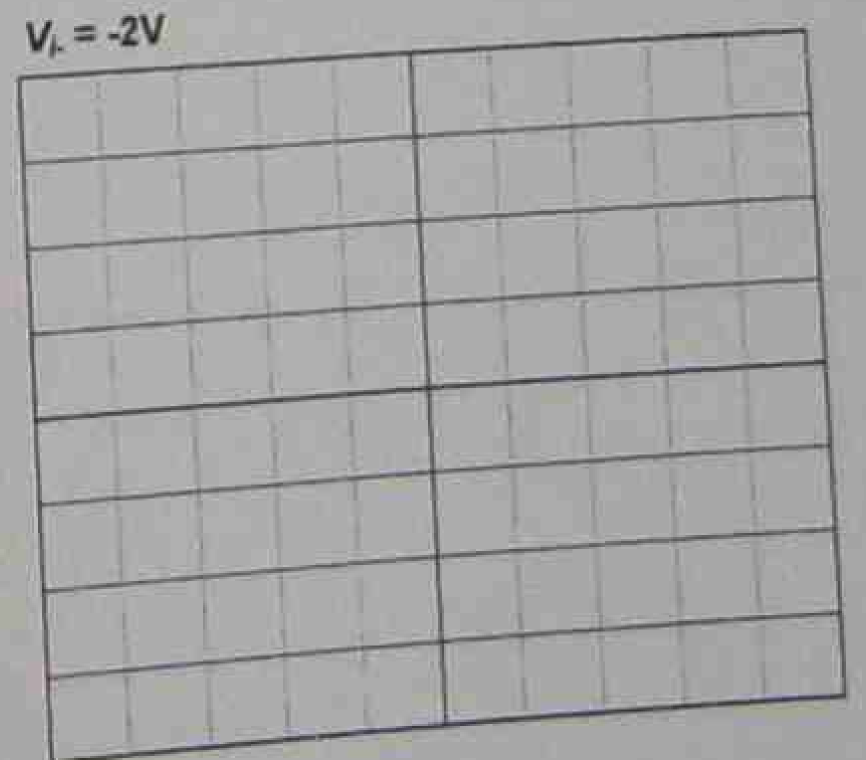
V_{i-} = +1V



V_{i-} = 0V

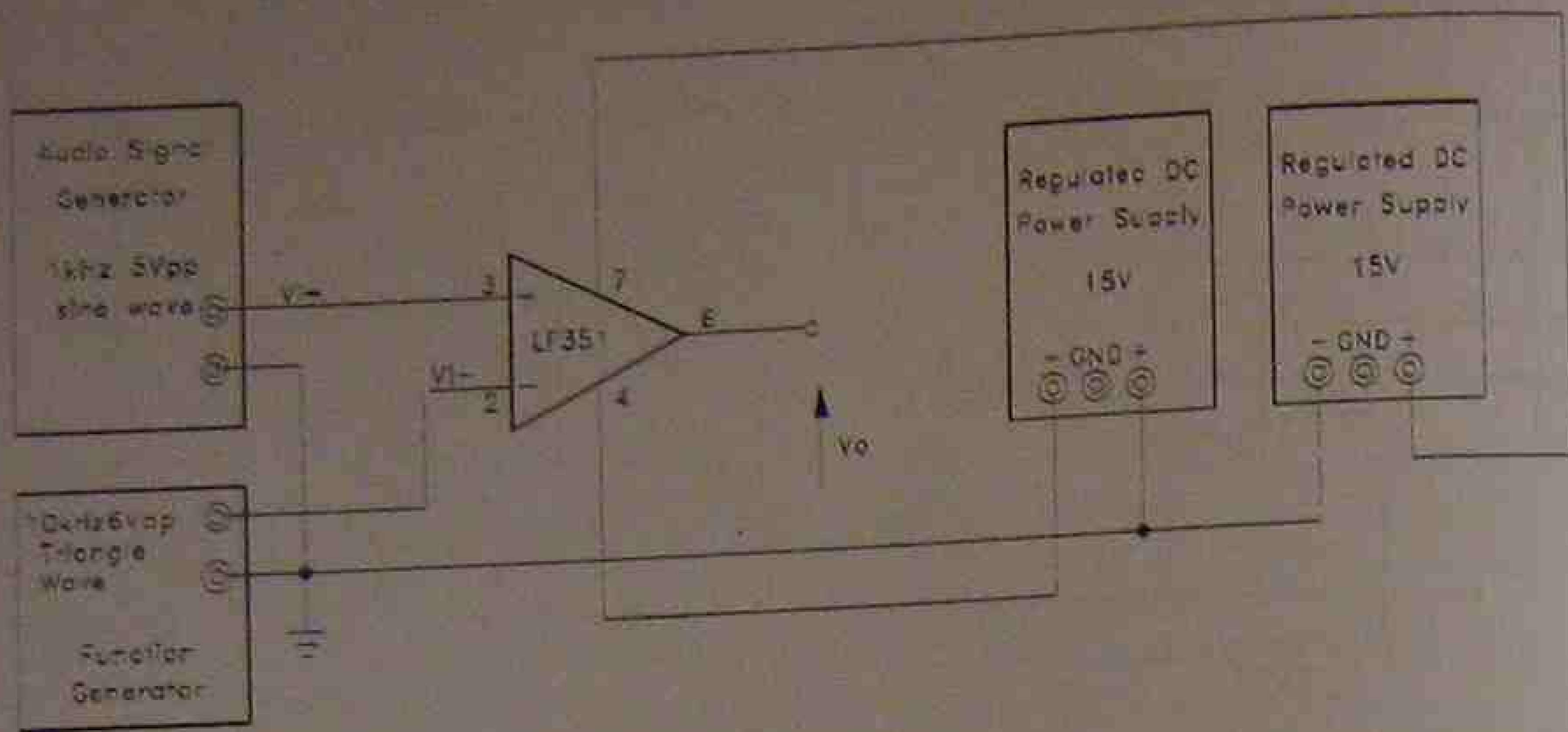


V_{i-} = -2V

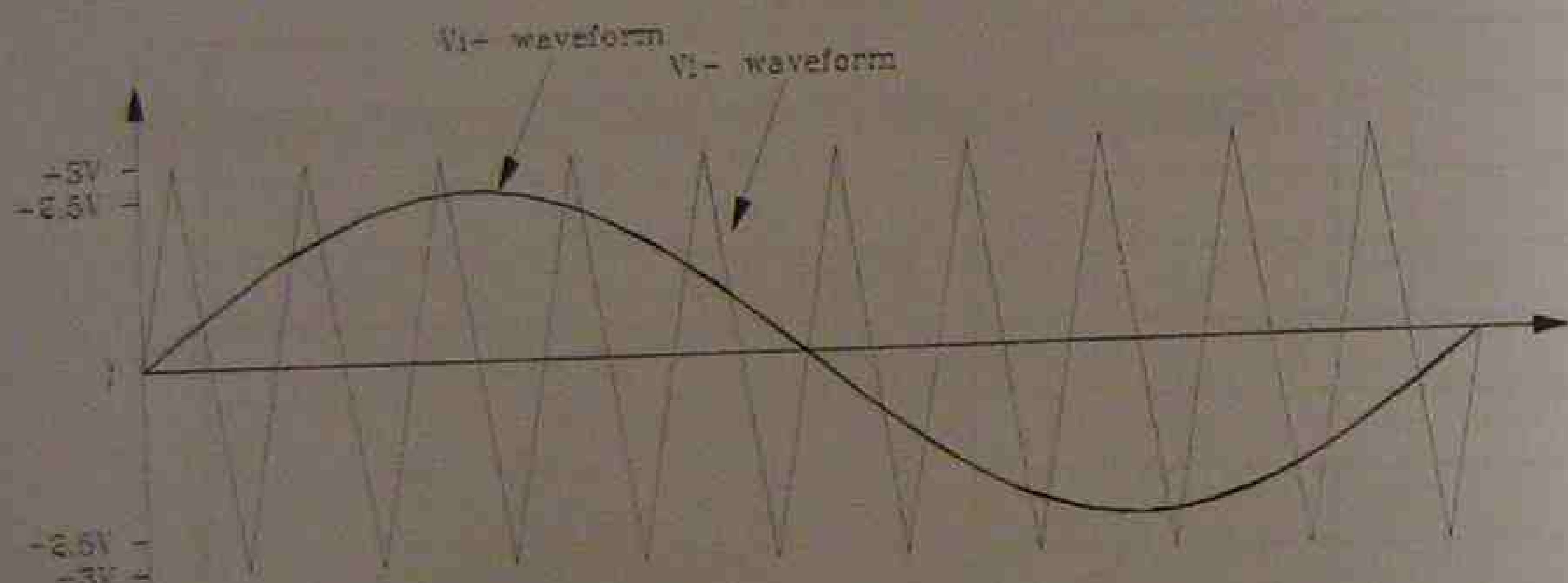


Ho Choi

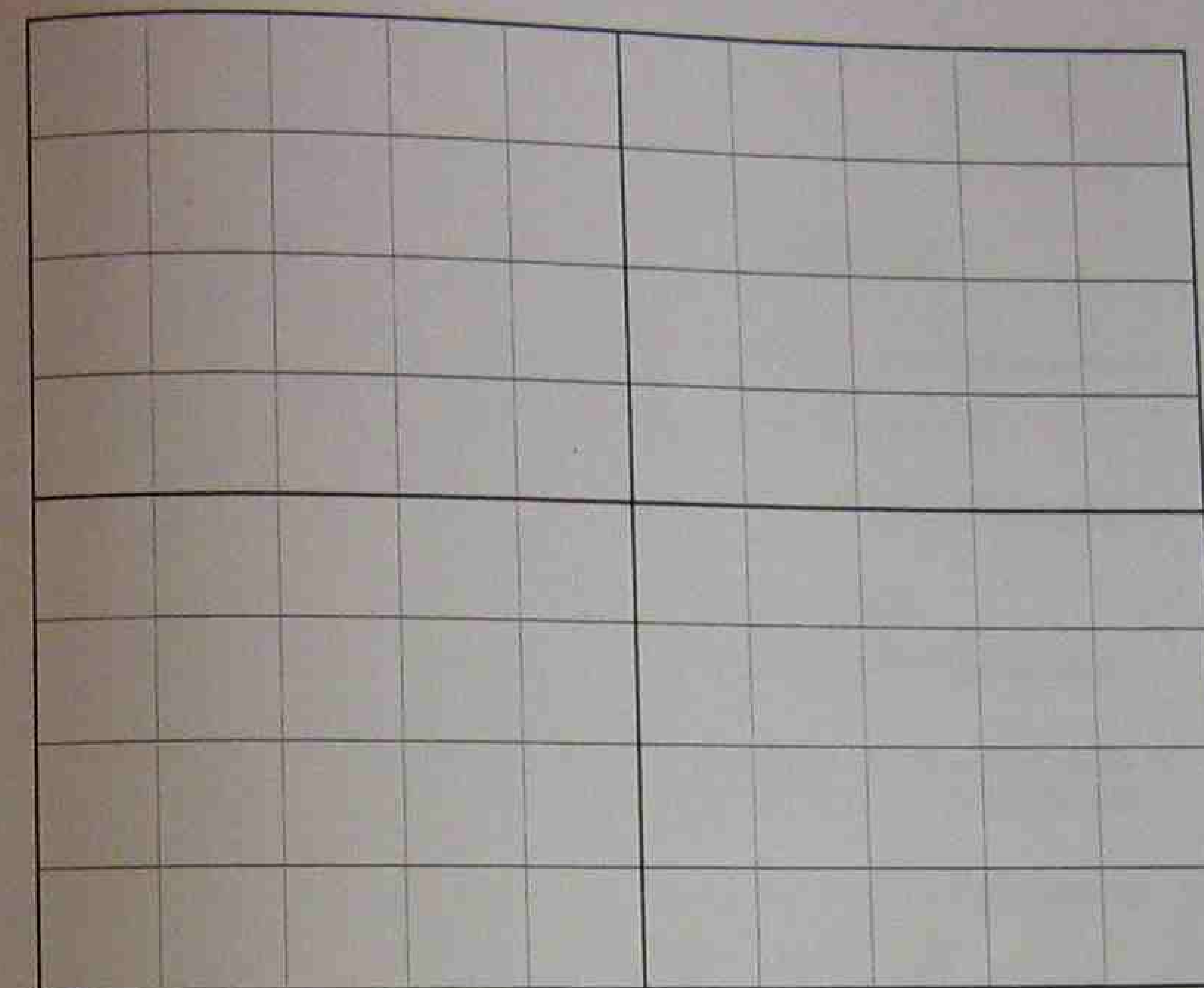
3. Replace voltage divider for V_{i-} with a function generator shown below then observe the output waveform.



(a) Draw theoretical output waveform.



(b) Sketch both V_{i-} and V_O waveforms.



(c) Briefly explain how PWM is working.

4

Closed-loop Amplifiers

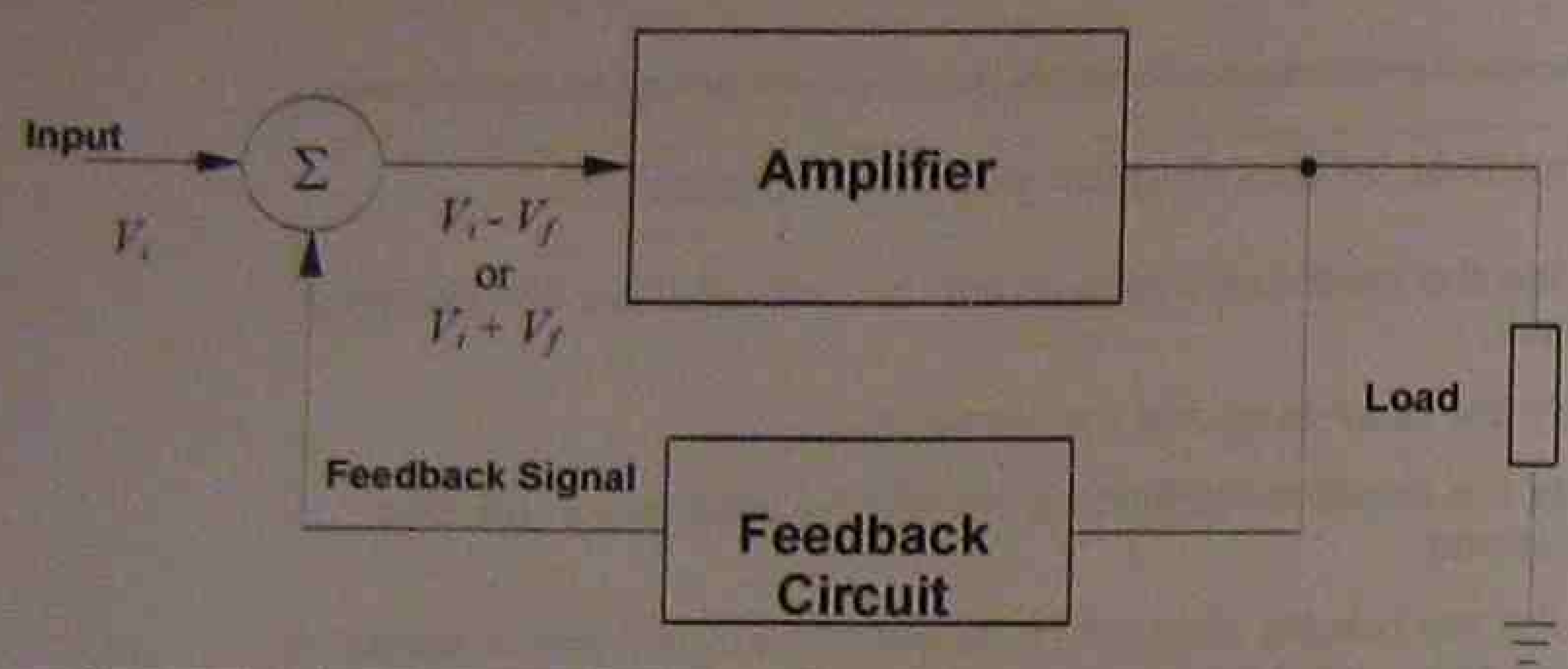
Upon completion of this chapter, you should be able to:

- Describe the feedback configuration for both positive and negative feedback.
- Identify the open-loop and the closed-loop circuit.
- Identify the positive feedback and negative feedback in op-amp applications.
- Describe the results of negative feedback on circuit performance in terms of gain, input resistance, output resistance, bandwidth and distortion.
- Describe three basic concepts for the analysis of an ideal op-amp circuit applications.
- Define voltage gain, input resistance and output resistance for inverting amplifier, non-inverting amplifier and voltage follower using three basic ideal op-amp concepts .
- Design an inverting amplifier for your application.
- Design a non-inverting amplifier for your application.
- Describe the purpose of votage follower.
- Design a voltage follower for your application.

4.1 Feedback

Many of the complex circuits used in modern electronics incorporate the concept of *feedback*, in which a portion of circuit's output is returned to its input either to augment or to reduce the original input signal. Figure 4.1 shows the general feedback configuration.

Figure 4.1 Basic feedback configuration

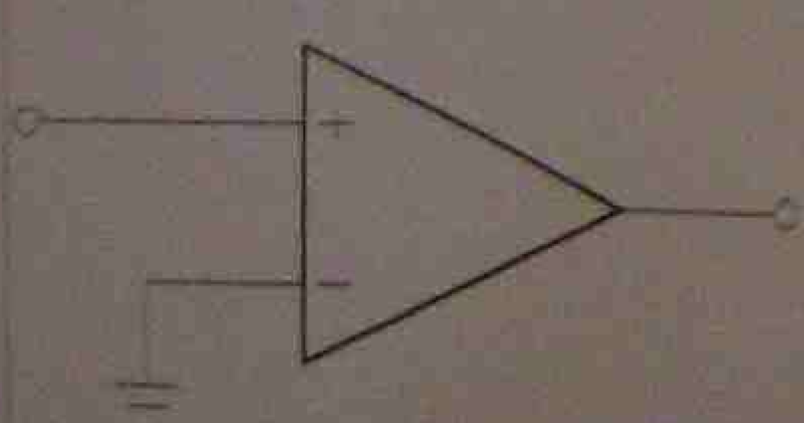


Open-loop and Closed-loop

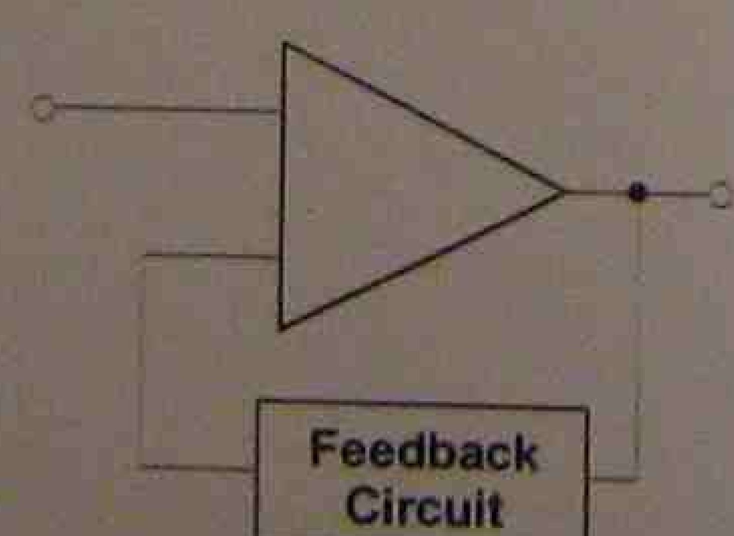
In an **open-loop** circuit, there is no feedback link between the output terminal and the input terminal. Figure 4.2 (a) shows an open loop amplifier using an op amp. You have already discovered that an open-loop op amp has a very large voltage gain (more than 100,000). This characteristic creates a problem in using an op amp as an amplifier since even tiny signals can drive an op amp into saturation.

If there is a feedback link between the output terminal and the input terminal, it is called a **closed-loop** circuit (Figure 4.2(b).

Figure 4.2 Open-loop and closed-loop circuit



(a) Open-loop circuit

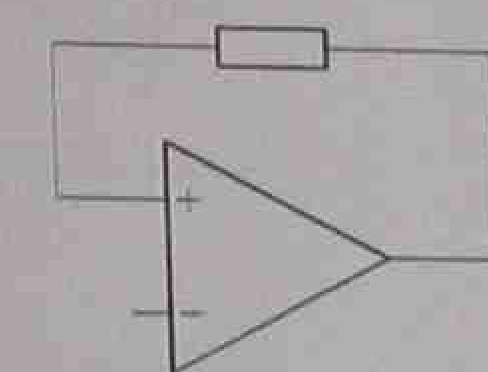


(b) Closed-loop circuit

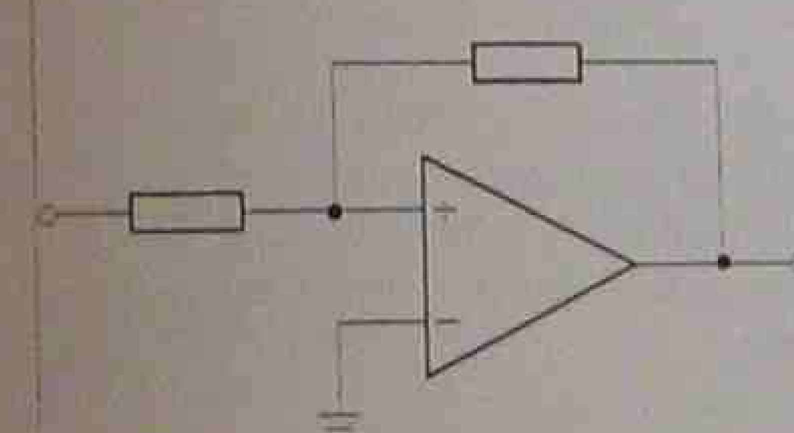
Positive Feedback

If the feedback tends to increase the input amplitude, it is called **positive feedback**. Figure 4.3 (a) shows a basic concept of positive feedback using an op amp and Figure 4.3 (b), (c) shows a real application of positive feedback (Schmitt trigger). Notice that the feedback link is connected to the non-inverting input terminal. Therefore, this positive feedback link tends to increase the input amplitude and eventually it drives an op amp into saturation. An op amp with positive feed back has only two output conditions - positive saturation and negative saturation. This is very good for an on-off control circuit including a digital logic circuit, however, it is not suitable for an amplifier design.

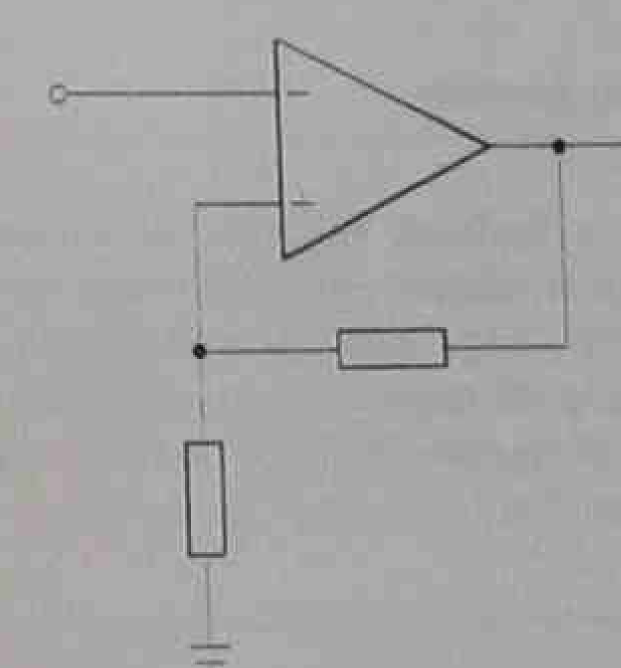
Figure 4.3 Positive feedback



(a) Positive feedback concept



(b) Non-inverting Schmitt rigger

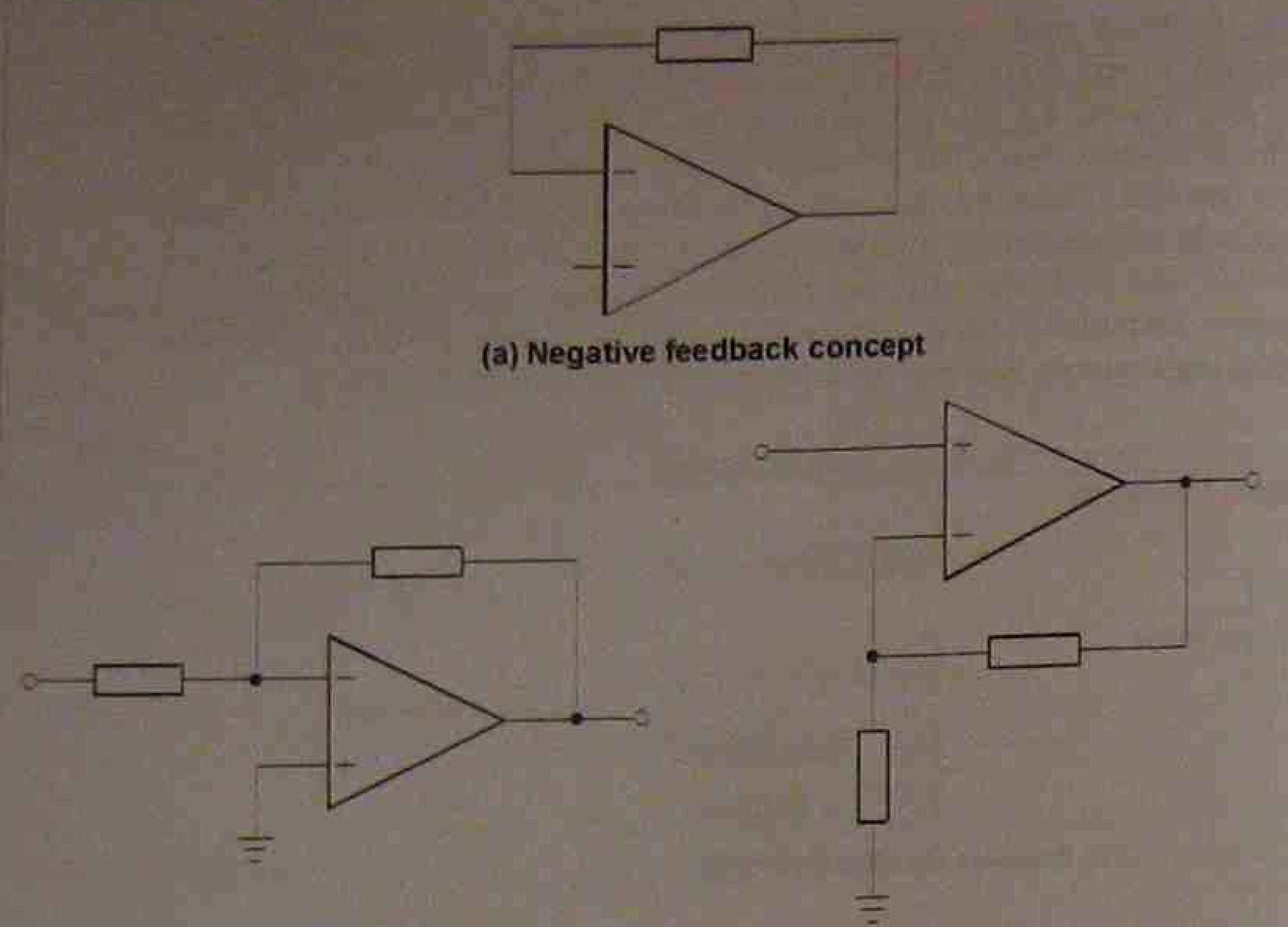


(c) Inverting Schmitt trigger

Negative Feedback

If the feedback tends to decrease the input amplitude, it is called **negative feedback**. Figure 4.4 (a) shows a negative feedback concept. Notice that the feedback link is connected to the inverting input to suppress the input voltage. Therefore, the amplifier output signal will not be saturated unless an excessive input voltage is applied. The negative feedback is essential to design an amplifier using an op-amp.

Figure 4.4 Negative feedback



(a) Negative feedback concept

(b) Inverting amplifier

(c) Non-inverting amplifier

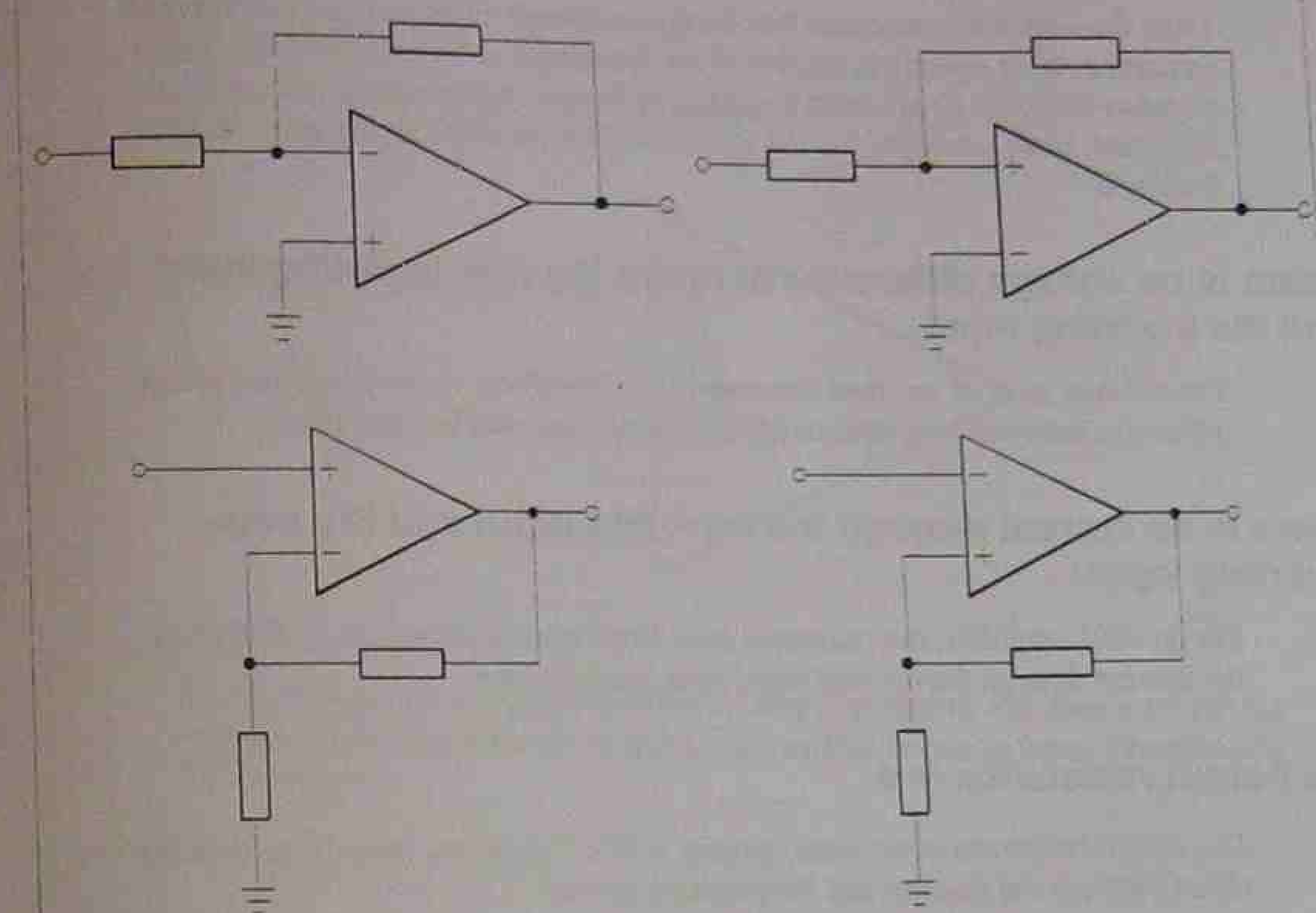
Negative feedback enhances all the desirable properties of a circuit at the expense of reduced gain. That is why the general purpose op amps are designed with a very large open-loop voltage gain. Negative feedback is extremely useful for nearly all control circuits and many signal circuits. Table 4.1 shows the results of negative feedback on circuit performance.

Table 4.1

	Type of amplifier			
	Voltage	Current	Transconductance	Transresistance
Reduces	voltage gain	current gain	transconductance	transresistance
Input resistance	increases	decreases	increases	decreases
Output resistance	decreases	increases	increases	decreases
Bandwidth	increases	increases	increases	increases
Distortion	decreases	decreases	decreases	decreases

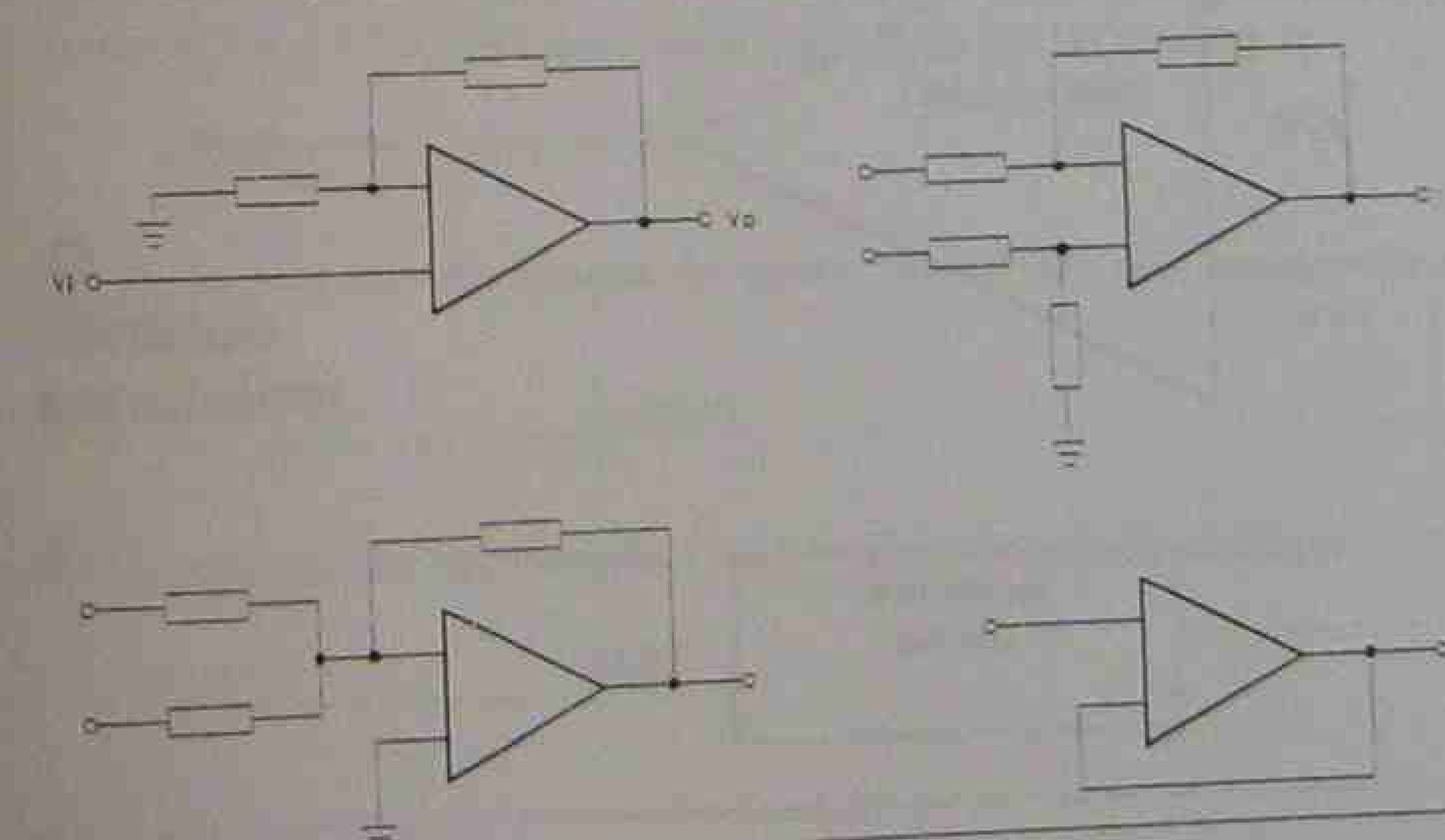
Drill Question 1

For the circuit diagrams shown below, identify the type of feedback.



Drill Question 2

The circuit diagrams shown below need negative feedback. Indicate the polarity of the input terminals.



4.2 Three Basic Concepts for the Operational Amplifier

From the section 4.1, we learnt that the op-amp needs negative feedback to avoid saturation. If an op-amp is employed for the design of an amplifier, the proper negative feedback circuit must be designed. In this chapter we are learning three important basic concepts for the analysis of an op-amp circuit with negative feedback.

There is no voltage difference between the non-inverting input and the inverting input.

The voltage gain of an ideal op-amp is ∞. Therefore, if there is any voltage difference between two input terminals the op-amp will be saturated.

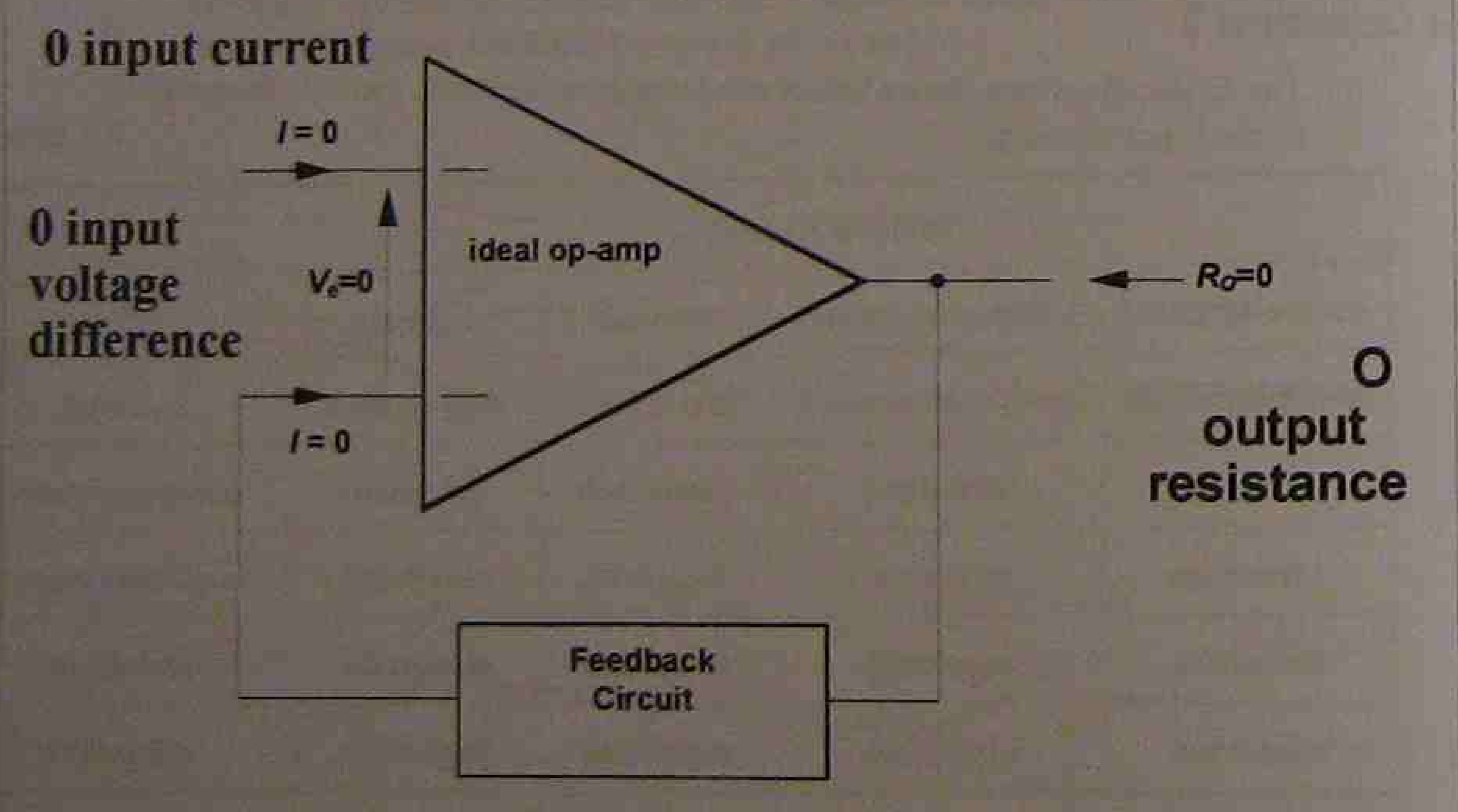
There is no current through the inverting input and the non-inverting input

For an ideal amplifier, the resistance seen from input terminals is ∞. Therefore, the current through the op-amp input lines must be zero.

The output resistance is 0.

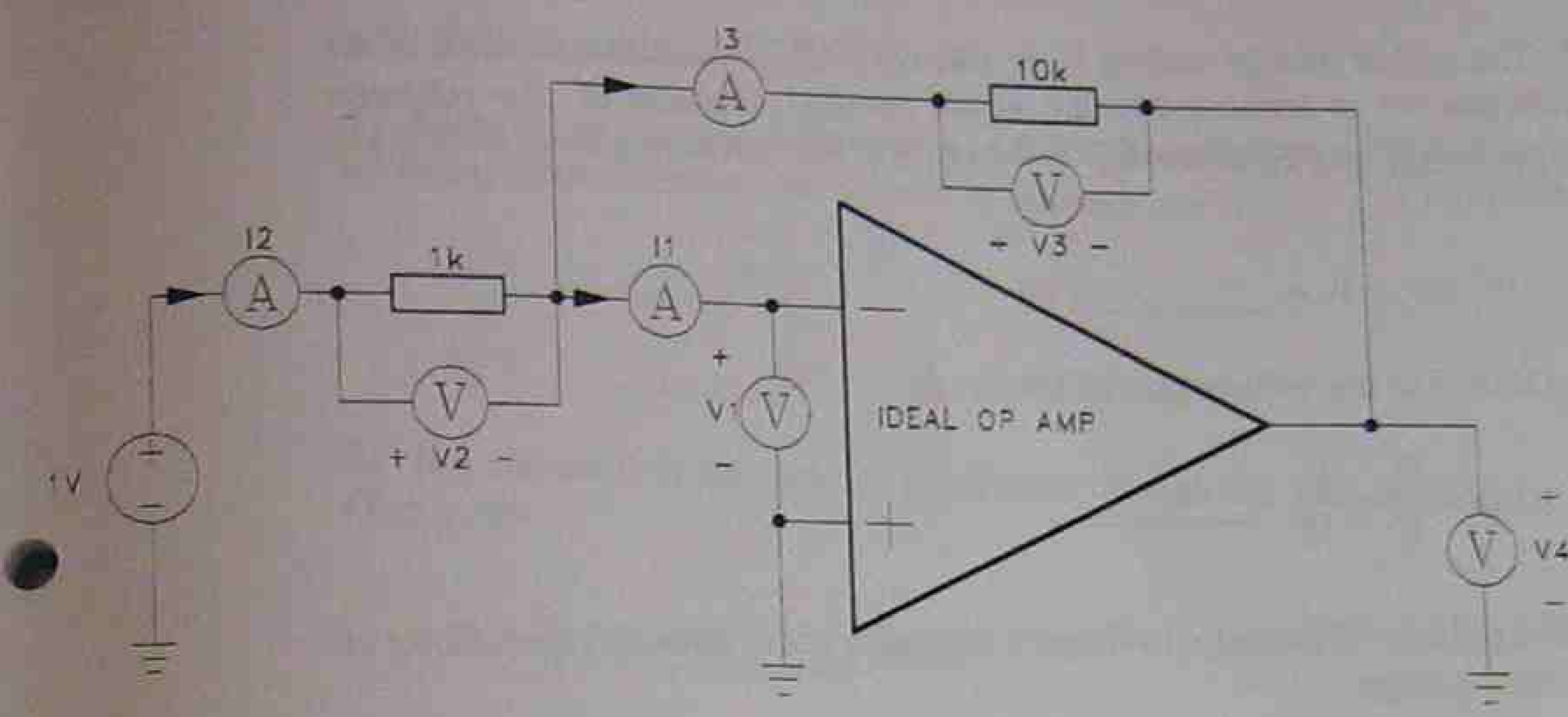
The output resistance of an ideal op-amp is 0Ω. Therefore, there is no loading effect between the op-amp and the feedback circuit.

Figure 4.5 Three basic concepts for the analysis of an op-amp circuit with negative feedback



Drill Question 3

For the voltage and current measurements shown below, determine the voltmeter and the current meter reading.



1. The non-inverting input (+) of the op-amp is grounded and there is no voltage difference between the non-inverting input andthe inverting input. Therefore the V_1 reading is:

V_1 = ________ (V)

2. There is no current through the inverting input and the non-inverting input. Therefore, I_1 reading is:

I_1 = ________

3. A 1V DC input signal is connected between the 1kΩ resistorand ground. Also, there is no voltage difference between the non-inverting input and the inverting input. Therefore, the voltage drop across the 1kΩ resistor is:

V_2 = ________ (V)

4. The current I_2 is the same as the current through the 1kΩ resistor. By the Ohm's law

$I_2 = \frac{V_2}{1k} =$ ________ (mA)

5. I_2 is divided into two currents, I_1 and I_3. Therefore, the I_3 reading is:

I_3 = ________ (mA)

Drill Question 3. cont.

6. By the Ohm's law, the voltage drop across the 10kΩ resistor is:

$V_3 = I_3 \times 10k =$ __________ (V)

7. The output voltage reading V_4 is: (**Be careful!** The positive terminal of the voltmeter V4 is connected to the output terminal. Because of the voltmeter connection V3 is opposite with V4, the value of V_3 must be negative (-) for the V_4 calculation.)

$V_4 = V_3 + V_1 =$ __________ (V)

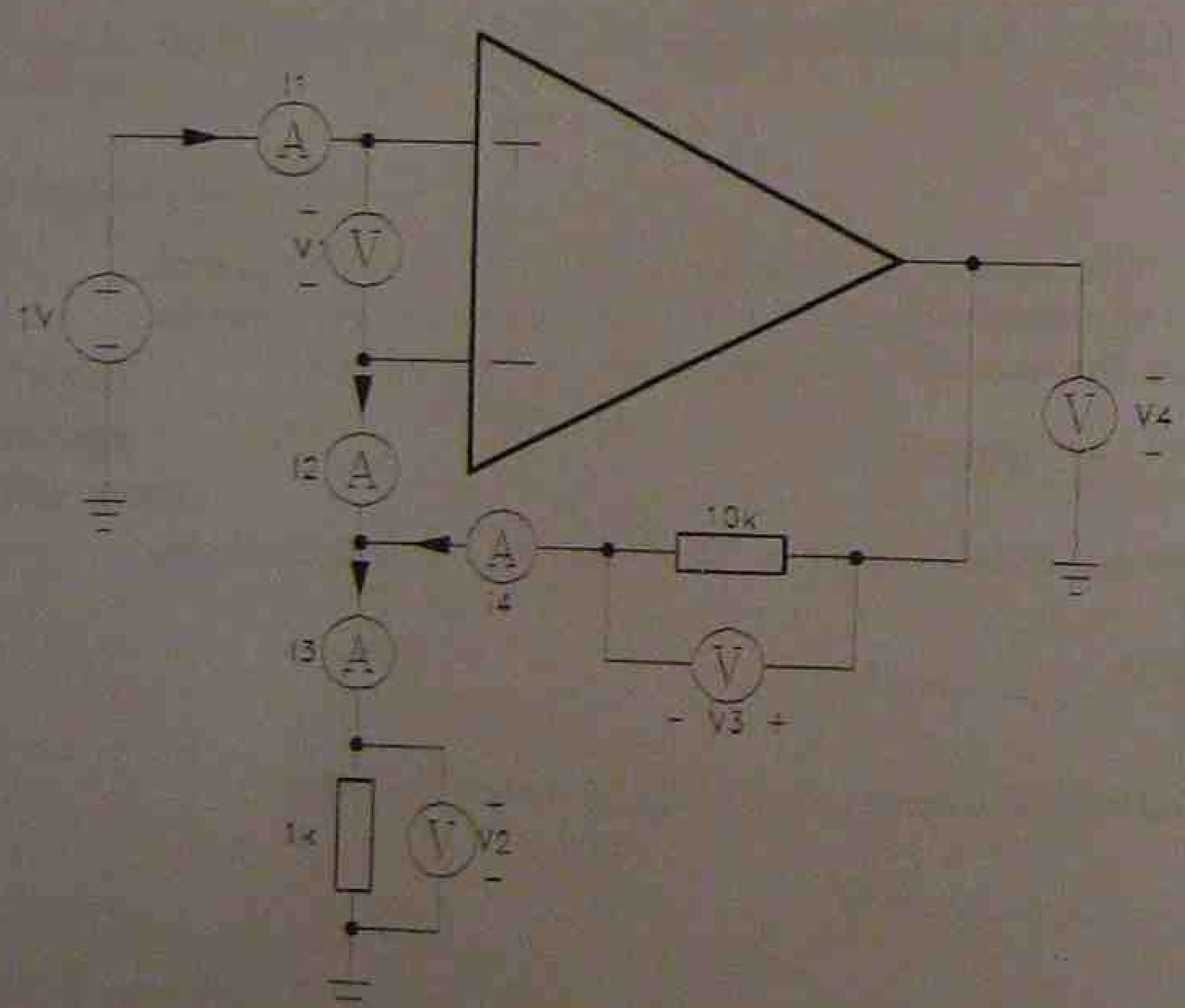
8. Determine the voltage gain of the circuit.

$$A_V = \frac{V_O}{V_i} = \frac{V_4}{1V} = \_\_\_\_\_\_\_\_\_\_$$

9. The polarity of the output voltage is (opposite with / same as) the polarity of the input voltage.

Drill Question 4

For the voltage and current measurements shown below, determine the voltmeter and the current meter reading.



1. There is no current through the non-inverting input and the inverting input. Therefore:

$I_1 =$ __________

$I_2 =$ __________

2. There is no voltage difference between the non-inverting input and the inverting input. Therefore the V_1 and V_2 readings are:

$V_1 =$ __________ (V)

$V_2 =$ __________ (V)

3. The current I_3 is the same as the current through the 1kΩ resistor. By the Ohm's law

$$I_3 = \frac{V_2}{1k} = \_\_\_\_\_\_\_\_\_\_ \text{ (mA)}$$

4. $I_3 = I_2 + I_4$ Therefore:

$I_4 =$ __________ (mA)

5. By the Ohm's law, $V_3 = I_4 \times 10k$. Therefore:

$V_3 =$ __________ (V)

6. The output voltage reading $V_4 = V_3 + V_2$. Therefore:

$V_4 =$ __________ (V)

7. Determine the voltage gain of the circuit.

$$A_V = \frac{V_O}{V_i} = \frac{V_4}{1V} = \_\_\_\_\_\_\_\_\_\_$$

8. The polarity of the output voltage is (opposite with / same as) the polarity of the input voltage.

4.3 Inverting Amplifier

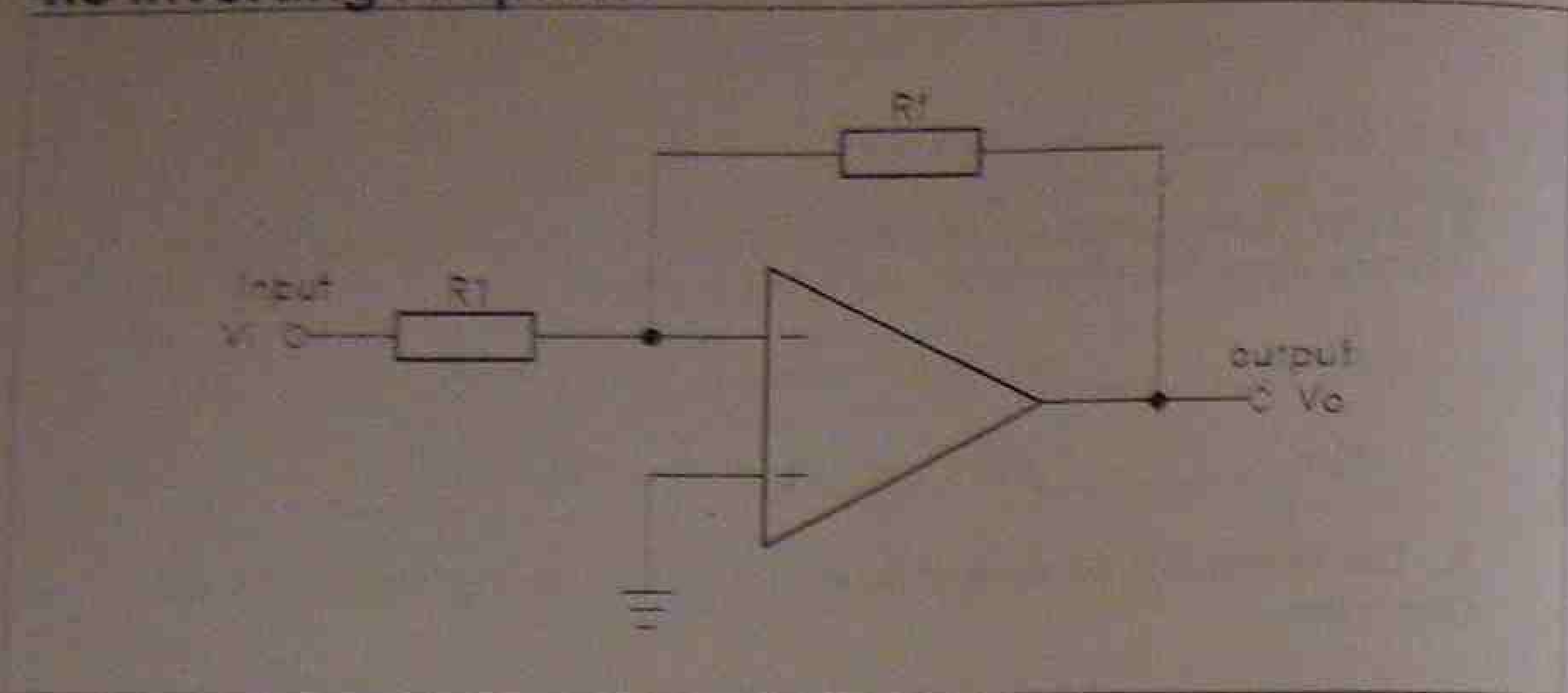


Figure 4.6 Inverting amplifier

The amplifier in Figure 4.6 is called an inverting amplifier because the signal to be amplified is applied to the inverting input. We have already tried the same circuit in Drill Question 3 and we noticed that the polarity of the output voltage is the opposite with the polarity of the input volute. Therefore, if we apply an ac signal to the input terminal the phase of the output voltage will be reversed (180° difference).

Voltage Gain

Let's work out the voltage gain of the inverting amplifier using basic concepts.

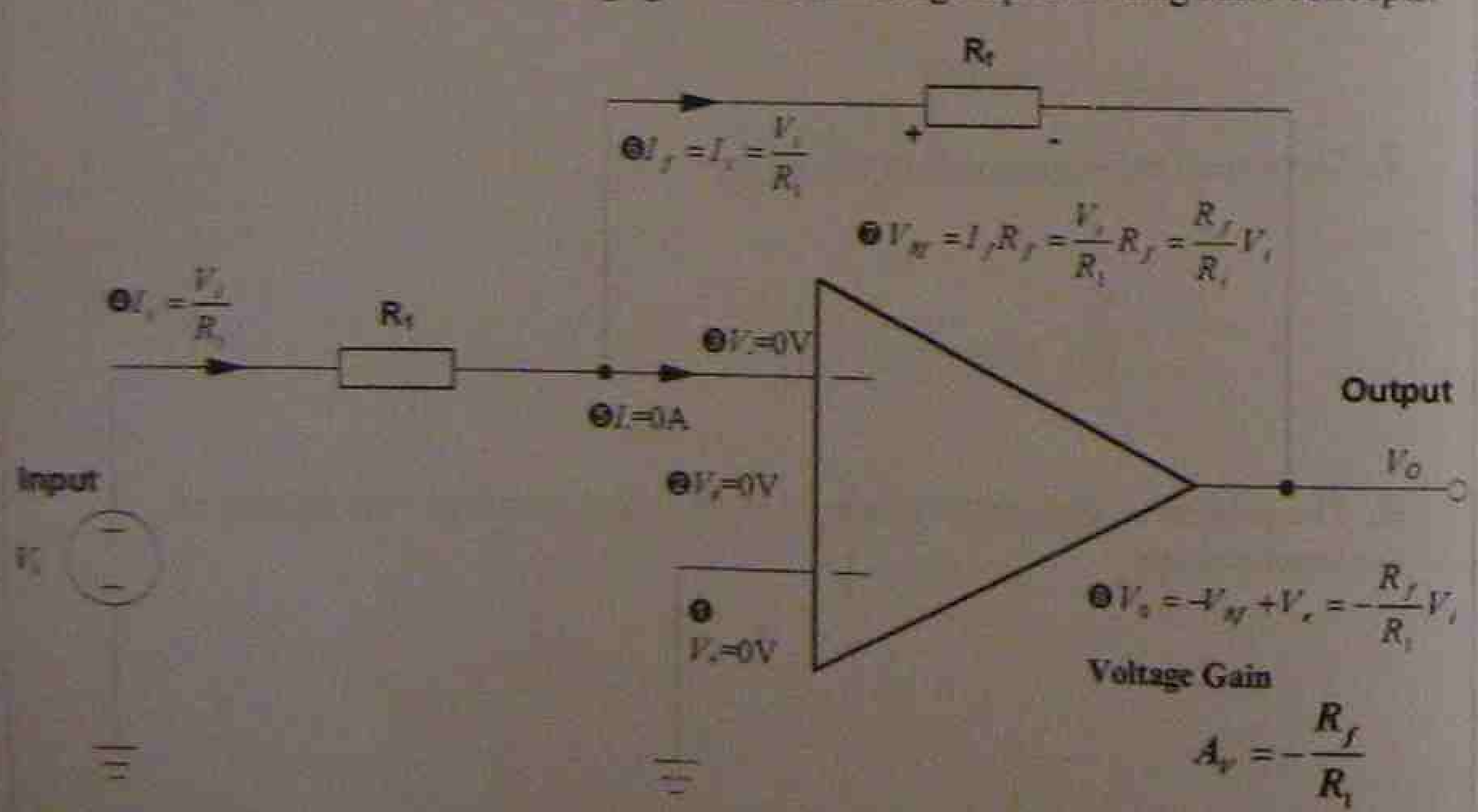


Figure 4.7 An analysis of an inverting amplifier

For the circuit analysis in Figure 4.7,

❶ The voltage at the non-inverting input is 0V since the non-inverting input is grounded.

❷ For an ideal op amp with negative feedback, there is no voltage difference between the non-inverting input and the inverting input. The voltage difference between the non-inverting input and the inverting input is used to called the **error voltage** (V_e) and it is extremely tiny in the practical op amp circuit. For example, If an op amp has an 100,000 open-loop voltage gain and the output voltage is 10V,

$$V_e = \frac{10V}{100,000} = 0.1mV$$

❸ Therefore, the voltage at the inverting input is 0 even though it is not grounded. Because the inverting input is virtually at ground potential, this is called the **virtual ground**.

❹ The amplifier input current can be easily determined using Ohm's law.

$$I_i = \frac{V_i}{R_1}$$

❺ Because of the input resistance of an ideal op amp is ∞Ω, there is no current through inverting input line.

❻ Therefore, the input current through the feedback resistor (I_f) should be same as the input current (I_i).

❼ Again by using Ohm's law, the voltage drop across R_f is:

$$V_{Rf} = I_f R_f = \frac{V_i}{R_1} R_f = \frac{R_f}{R_1} V_i$$

Because of the current direction of I_f is from left to right, the polarity of V_{Rf} must be the same as indicated on the circuit diagram.

❽ The output voltage is clearly the voltage difference between the output terminal and ground. There are two voltages between the output terminal and ground, V_{Rf} and V_e. The polarity of V_{Rf} at the output terminal is "-", therefore

$$V_o = -V_{Rf} + V_e$$

$$= -\frac{R_f}{R_1} V_i$$

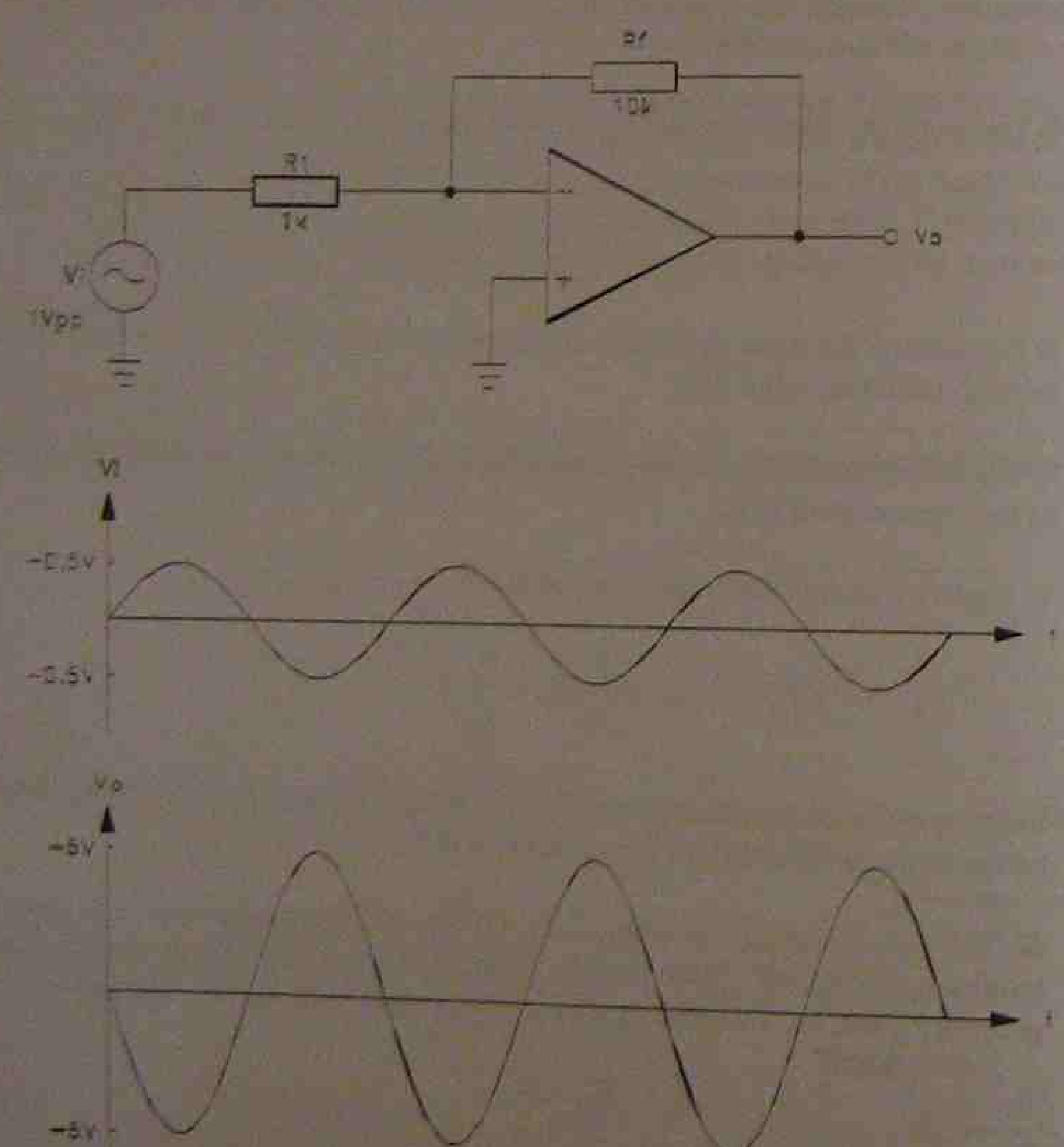
Now we can determine the voltage gain of the amplifier.

$$A_v = \frac{V_O}{V_i} = -\frac{R_f}{R_1}$$

The negative sign clearly tells us that it is an inverting amplifier and the output will be phase reversed with respect to the input. For the circuit diagram shown in Figure 4.8, the voltage gain of the amplifier is:

$$A_v = -\frac{R_f}{R_1} = -\frac{10k}{1k} = -10$$

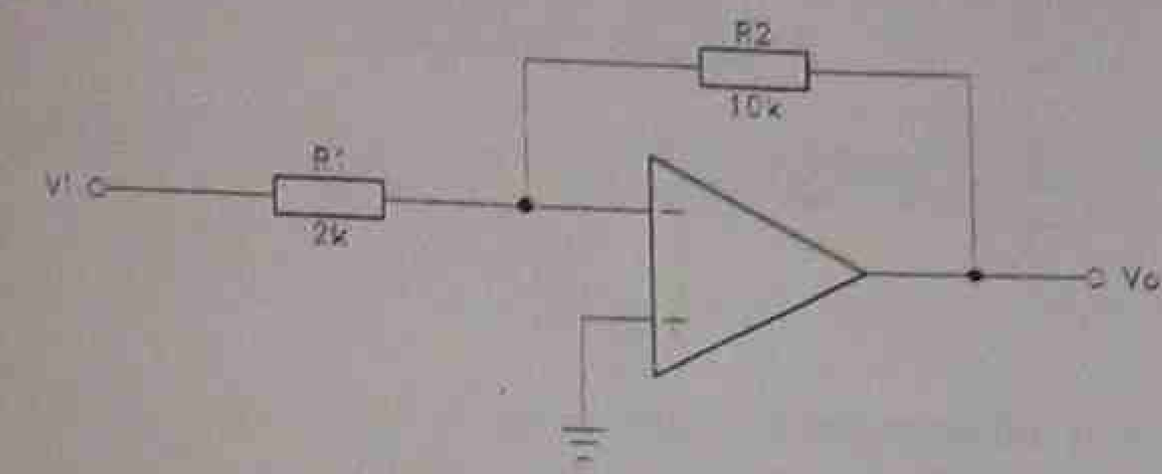
Figure 4.8 Reversed phase in an inverting amplifier



If a 1Vpp ac signal is applied to the input, the output signal will be 10Vpp with reversed phase with respect to the input as shown in Figure 4.8.

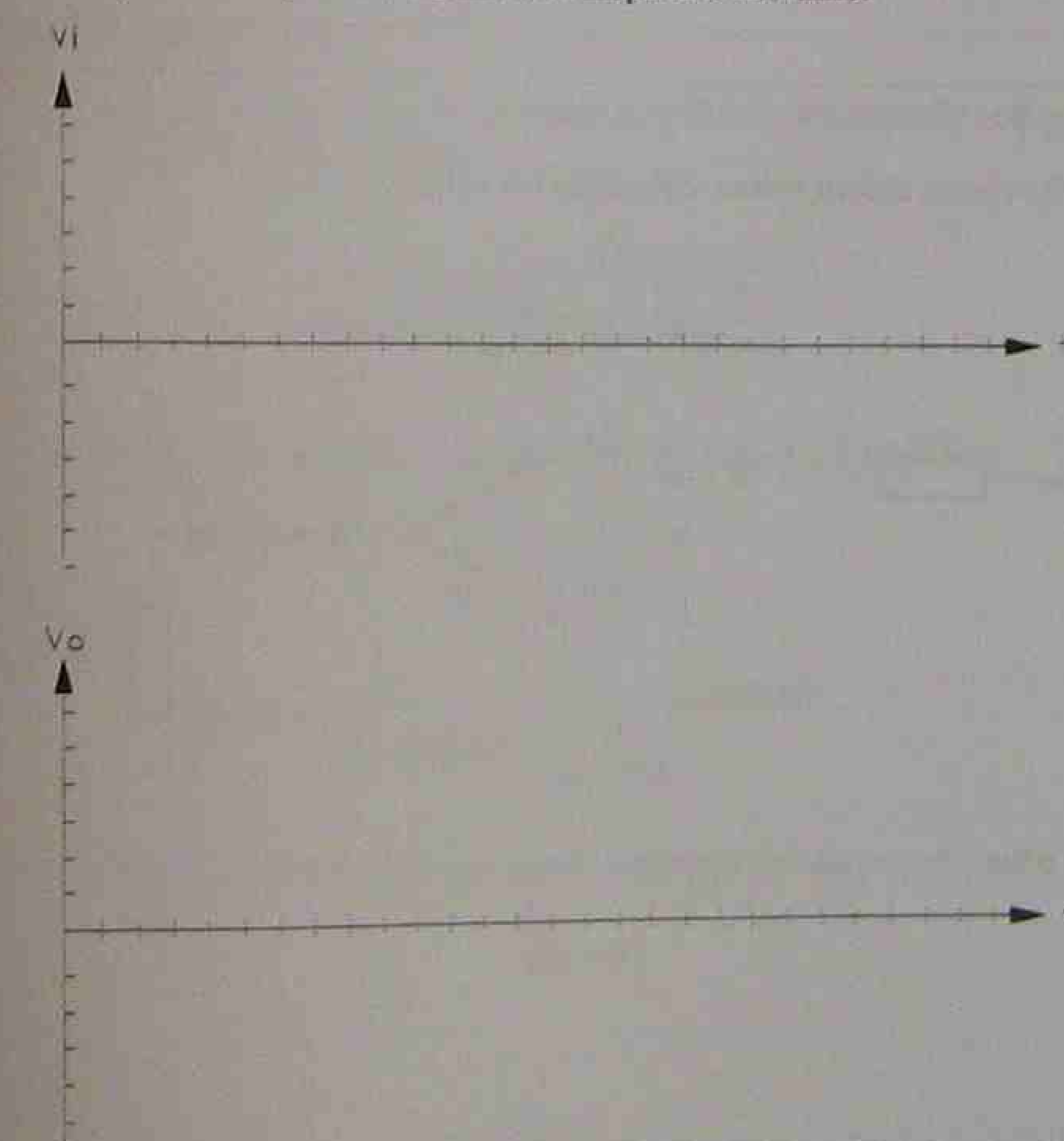
Drill Question 5

For an amplifier shown below:



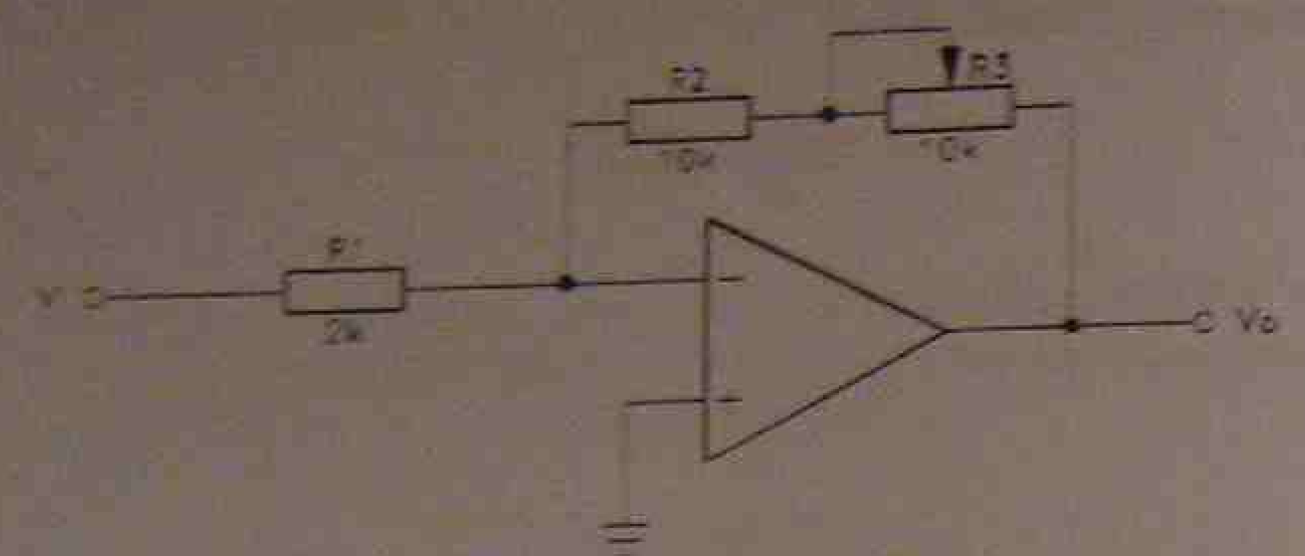
(a) Determine the voltage gain of the amplifier.

(b) A 0.5Vpp 1kHz sine wave signal is applied to the input terminal. Draw both input and output waveforms with amplitude and time.



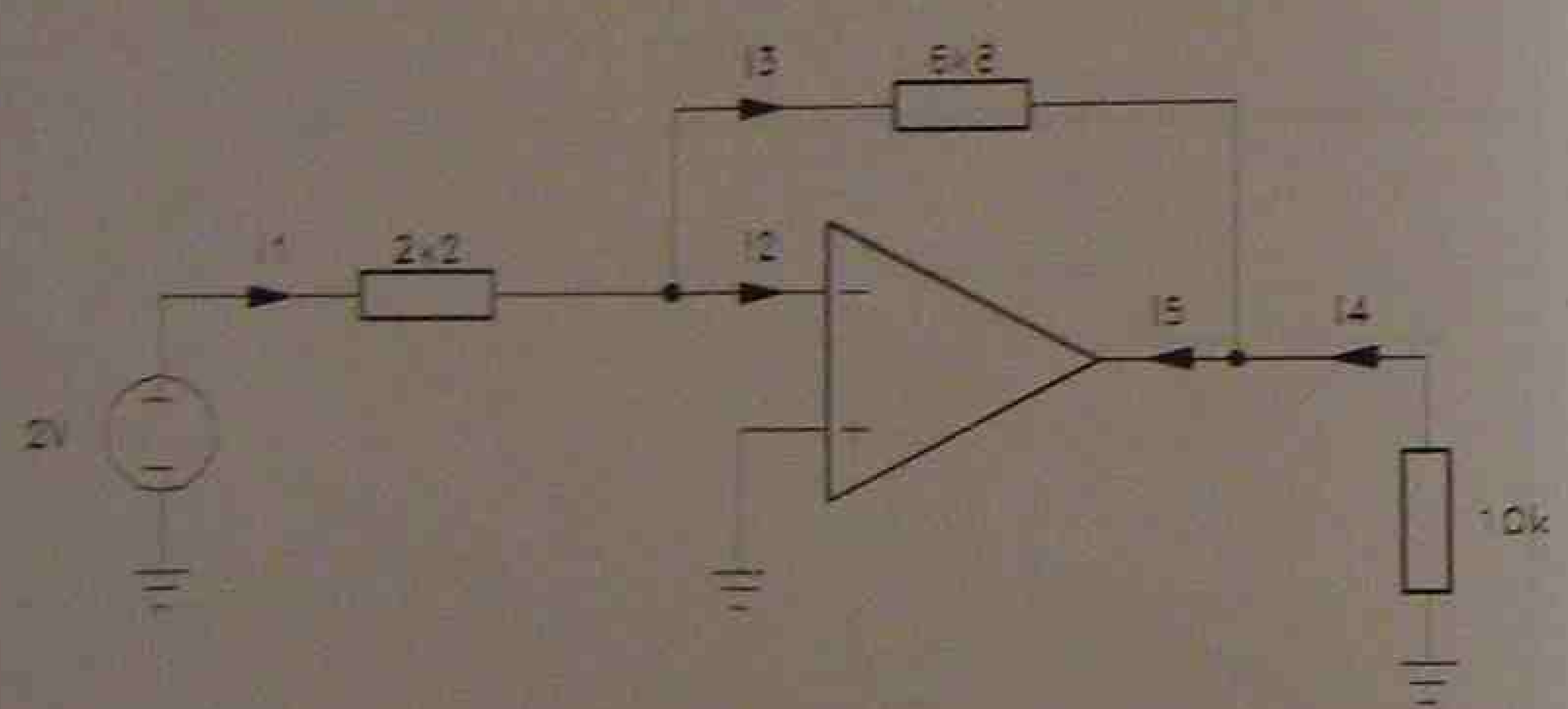
Drill Question 6

For the amplifier circuit shown below, determine the minimum voltage gain and the maximum voltage gain.



Drill Question 7

For the circuit shown below, determine the current $I_1 \sim I_5$.



Input Resistance

The input resistance of an op amp inverting amplifier can be simply determined by Ohm's law.

$$R_i = \frac{V_s}{I_s}$$

From the voltage gain analysis $I_s = \frac{V_s}{R_1}$, therefore:

$$R_i = R_1$$

The input resistance of an op amp inverting amplifier is R1. Theoretically a large value of R1 is required to increase the input resistance. However, there are many difficulties to raise the value R1 due to the offset voltage and amplifier stability. In a practical design, the value of feedback resistance is limited to 100kΩ for the general purpose op amp such as 741. Therefore, low input resistance is inevitable for a high gain inverting amplifier. This is the major disadvantage of the op amp inverting amplifier.

Output Resistance

The output resistance of inverting amplifier is dramatically reduced due to the negative feedback effect.

$$R_{O(closed)} = \frac{R_{O(open)}}{1 + A_{V(open)} \times \frac{R_1}{R_1 + R_f}}$$

If a LM741 op amp is employed for the Figure 4.8 amplifier,

LM741: $A_{V(open)} = 200{,}000$
$R_{O(open)} = 75\Omega$

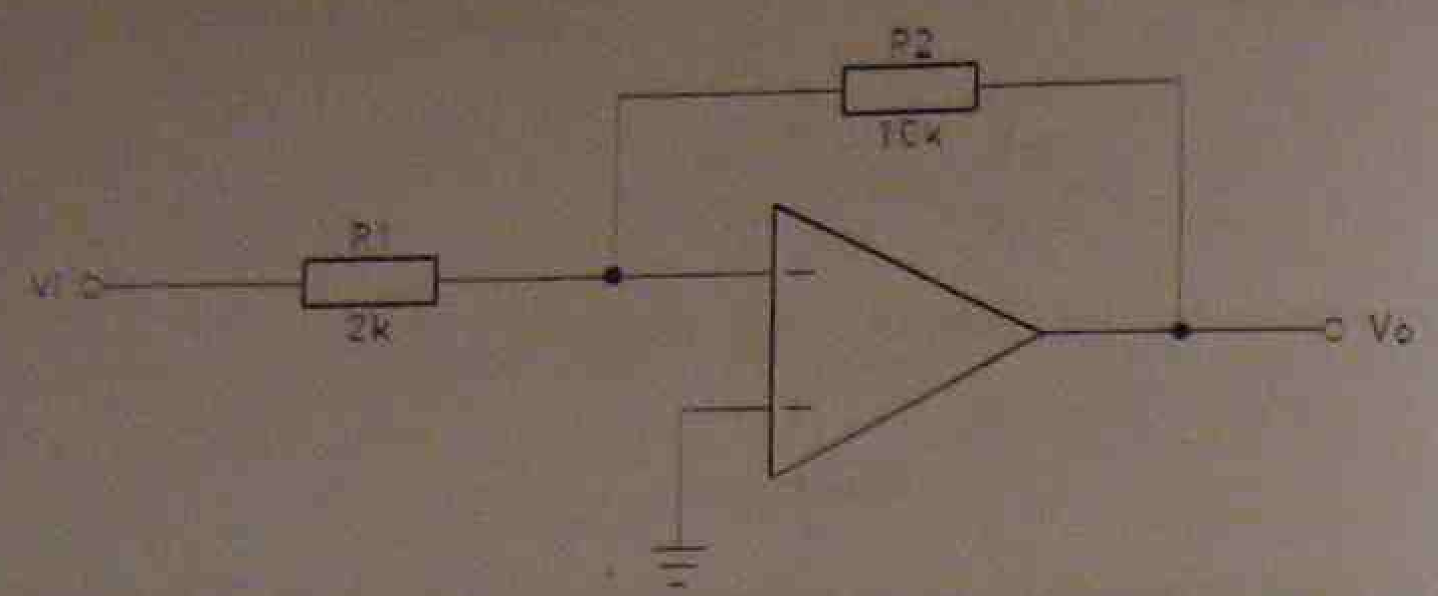
$$R_{O(closed)} = \frac{75}{1 + 200{,}000 \times \frac{1k}{1k + 10k}} \approx 0.004\Omega$$

It is a very low output resistance! Generally the output resistance is:

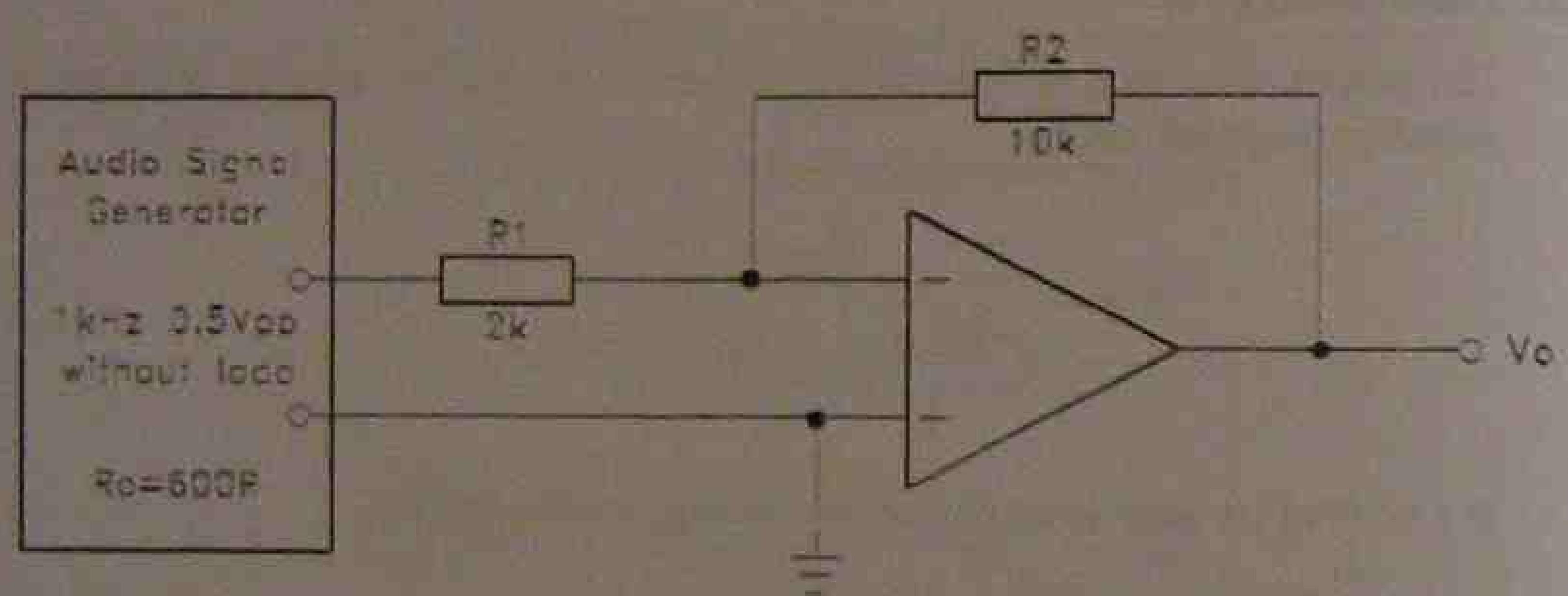
$$R_O = 0$$

Drill Question 8

(a) Determine the input resistance of the amplifier shown below.



(b) An audio signal generator is connected to the input terminal as shown below. The audio signal generator is adjusted for 1 kHz, 0.5Vpp without amplifier connection. Determine the output voltage (Vpp). The output resistance of the audio signal generator is 600Ω.

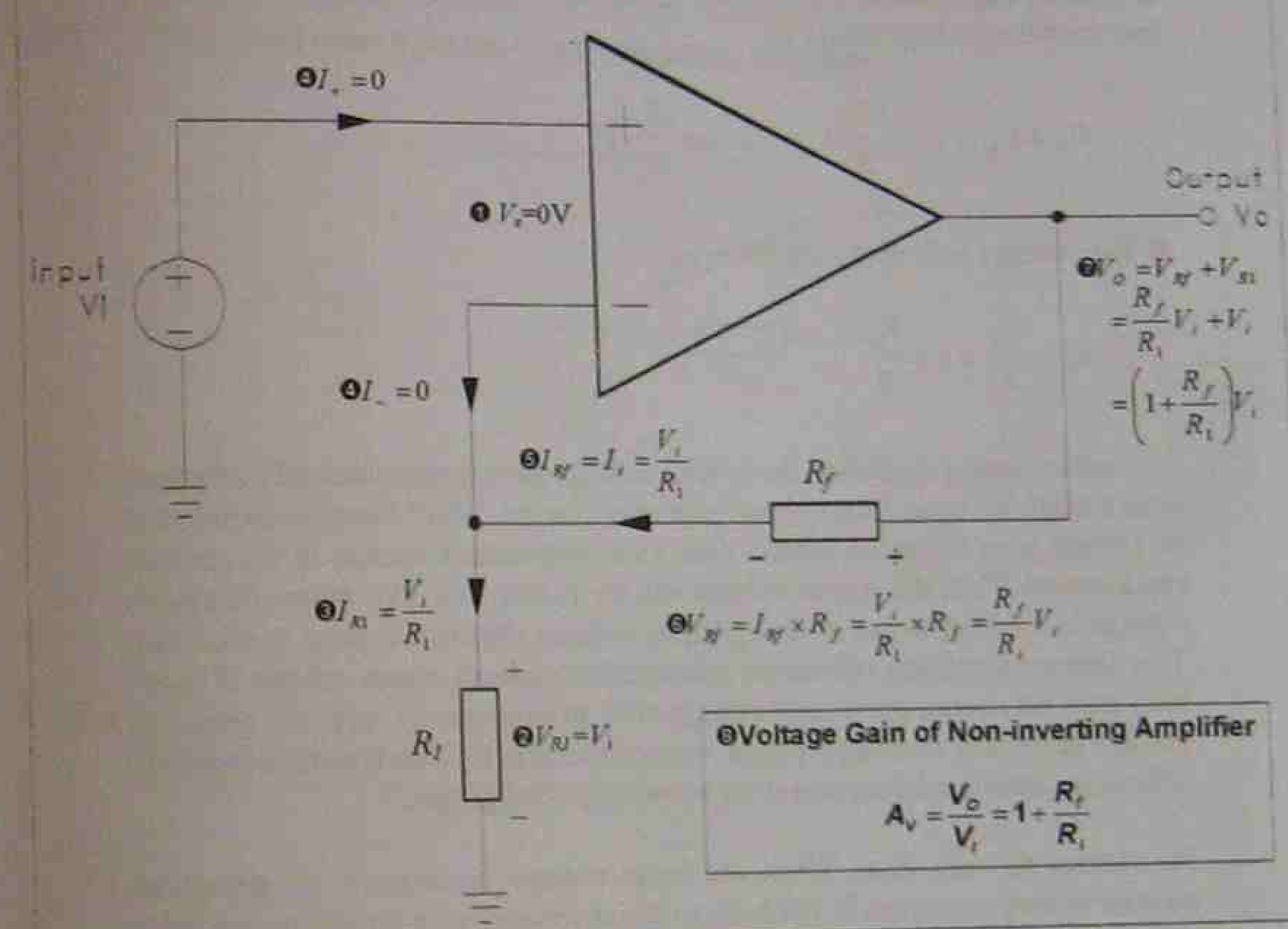


4.4 Non-inverting Amplifier

The most basic type of negative feedback is **non-inverting voltage feedback**. The word "noninverting" refers to the input voltage driving the non-inverting input of an op amp amp. An amplifier with non-inverting voltage feedback tends to act like a perfect voltage amplifier, one with infinite input resistance, zero output resistance, and constant voltage gain.

Figure 4.9 shows an analysis procedure for the non-invering amplifier using an op amp.

Figure 4.9 An analysis of a non-inverting amplifier



❶ There is no voltage difference between the non-inverting input and the inverting input.

❷ Therefore, the voltage drop across R_1 is the same as the input voltage V_i.

❸By Ohm's law, the current through R_1 is:

$$I_{R1} = \frac{V_{R1}}{R_1} = \frac{V_i}{R_1}$$

❹ There is no current through the non-inverting input and the inverting input since the input resistance of an ideal op amp is ∞.

❺ Therefore, the current I_{R1} should be supplied from the output of op amp through $r_{reference}$. In other words $I_{R1} = I_{Rf}$

❻ Now we can calculate the voltage drop across R_f using Ohm's law.

$$V_{Rf} = I_{Rf} \times R_f = \frac{V_i}{R_1} \times R_f = \frac{R_f}{R_1} V_i$$

❼ Now we can determine the output voltage by adding the voltage drop across R_f and the voltage drop across R_1.

$$V_O = V_{Rf} + V_{R1} = \frac{R_f}{R_1} V_i + V_i = \left(1 + \frac{R_f}{R_1}\right) V_i$$

❽ The voltage gain of the amplifier is:

$$A_V = \frac{V_O}{V_i} = 1 + \frac{R_f}{R_1}$$

In a non-inverting amplifier, the overall voltage gain is approximately constant, even though the open-loop voltage gain may change. Why? Suppose an increase in voltage gain for some reason such as a temperature change or an op amp replacement. Then the output voltage will try to increase. This means that more voltage is fed back to the inverting input, causing the error voltage to decrease. This almost completely offsets the attempted increase in output voltage. If open-loop gain decreases, the output voltage tries to decrease. In turn, the feedback voltage decreases, causing the error voltage to increase. This almost completely offsets the attempted decrease in an open-loop voltage gain.

Remember the key idea. When the input voltage is constant, an attempted change in output voltage is fed back to the inverting input, producing an error voltage that automatically compensates for the attempted change. This idea is also widely applied for the voltage regulator design.

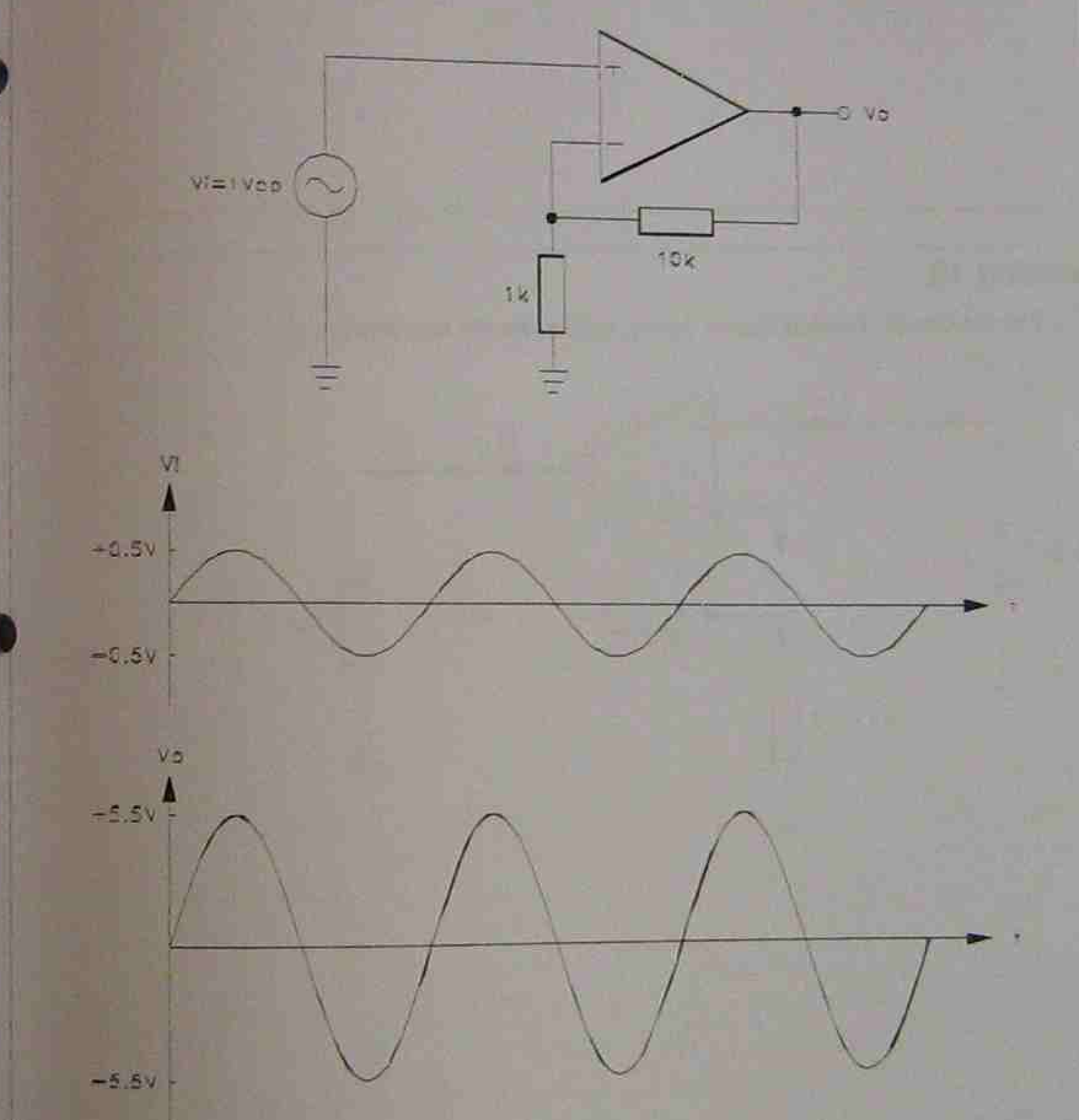
Input and Output Phase

The positive voltage gain clearly tells us that it is a non-inverting amplifier and the output will be in phase with respect to the input. For the circuit diagram shown in Figure 4.10, the voltage gain of the amplifier is:

$$A_V = 1 + \frac{R_f}{R_1} = 1 + \frac{10k}{1k} = 11$$

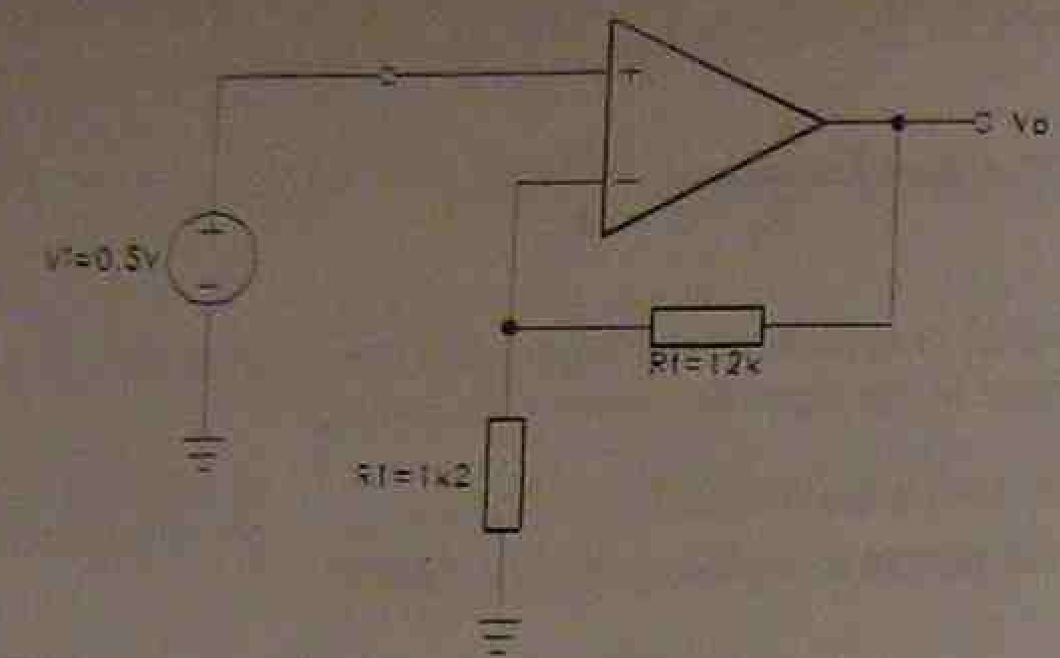
If a 1Vpp ac signal is applied to the input, the output signal will be 11Vpp in phase with respect to the input as shown in Figure 4.8.

Figure 4.10 Input and output phase in non-inverting amplifier



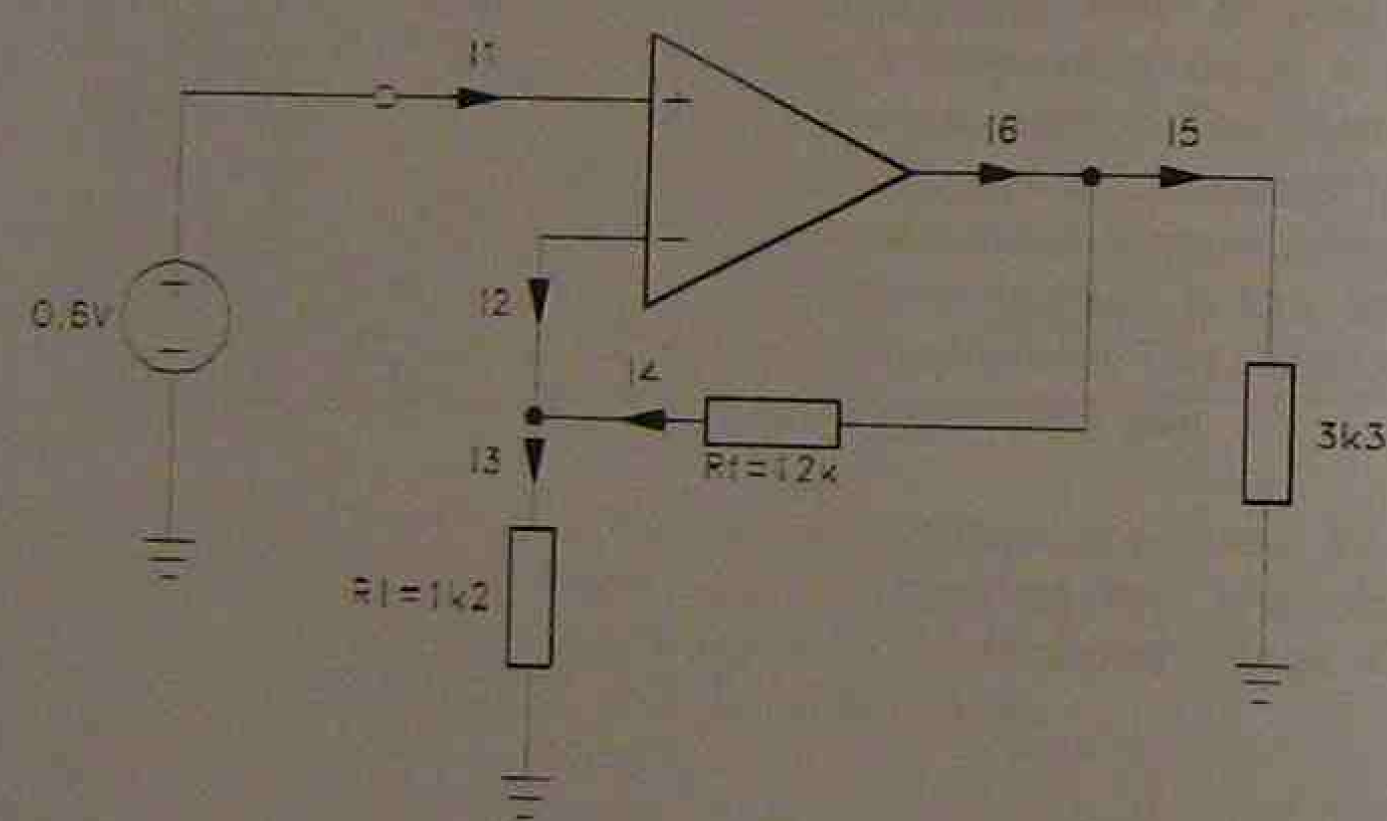
Drill Question 9

For the circuit diagram shown below, determine the expected output voltage.



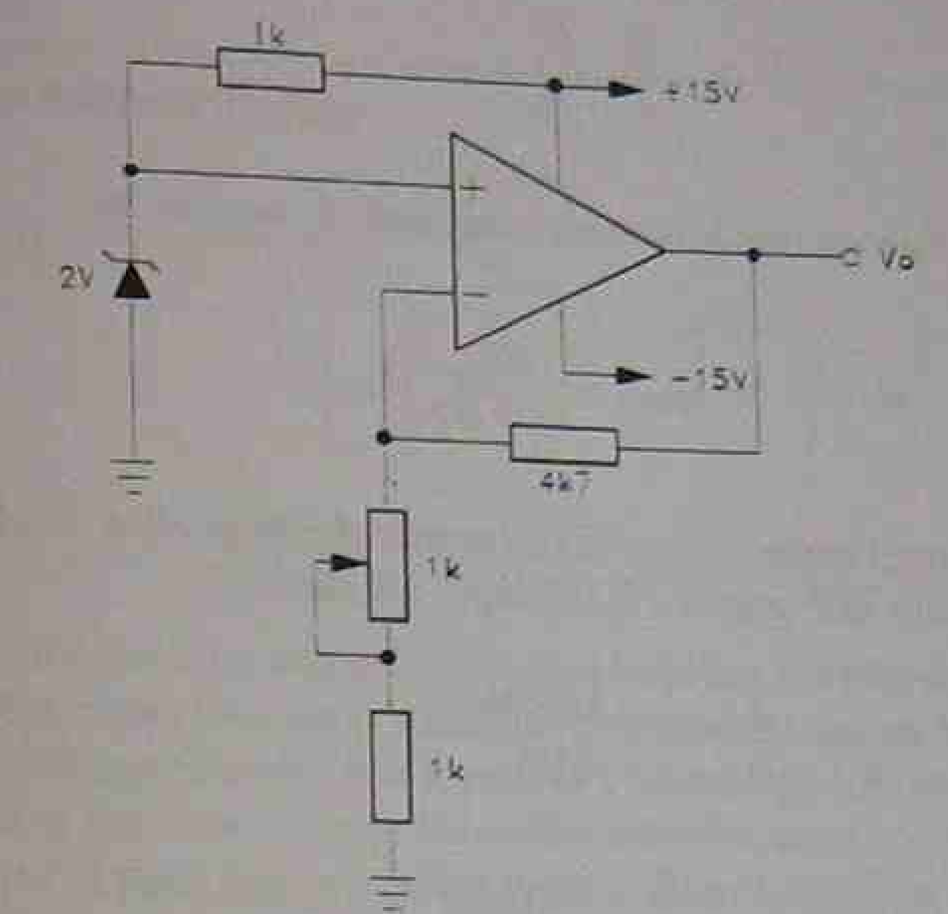
Drill Question 10

For the circuit diagram shown below, determine the currents $I_1 \sim I_6$.



Drill Question 11

For the circuit diagram shown below, determine the minimum output voltage and the maximum output voltage.



Input Resistance

Because the non-inverting input current of an ideal op amp is 0, the input resistance of the non-inverting amplifier in Figure 4.9 is:

$$R_i = \frac{V_i}{I_i} = \frac{V_i}{0} = \infty$$

For a more accurate input resistance calculation, we can use:

$$R_i = \left(1 + A_{V(open)} \times \frac{R_1}{R_1 + R_f}\right) R_{i(open)}$$

Under the normal applications, it is an extremely large value. If a LM324 is used for the amplifier in Figure 4.10, the input resistance is:

LM324: $A_{V(open)} = 100{,}000$
$R_{i(open)} = 1\text{M}\Omega$

$$R_i = \left(1 + 100{,}000 \times \frac{1k}{1k + 10k}\right) \times 1M\Omega \approx 9{,}092M\Omega \approx 9T\Omega$$

Output Resistance

Like the inverting amplifier in section 4.3, the output resistance is:

$$R_{O(closed)} = \frac{R_{O(open)}}{1 + A_{V(open)} \times \frac{R_1}{R_1 + R_f}}$$

As we discussed in Section 4.3, it is an extremely low resistance.

Summary

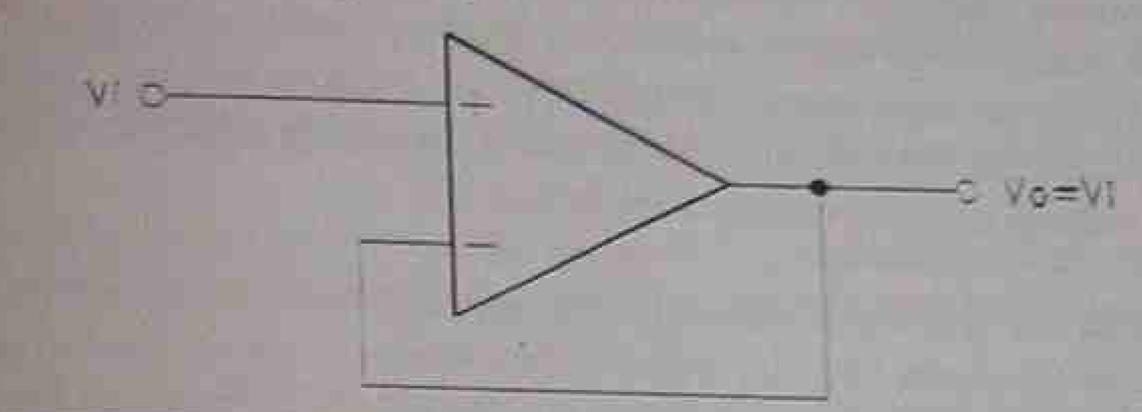
Table 4.2 Characteristic comparison of inverting amplifier and non-inverting amplifier

	Voltage Gain	Input Resistance	Output Resistance
Inverting Amplifier	$-\frac{R_f}{R_1}$	R_1	0
Non-inverting Amplifier	$1+\frac{R_f}{R_1}$	∞	0

4.5 Voltage Follower (Buffer)

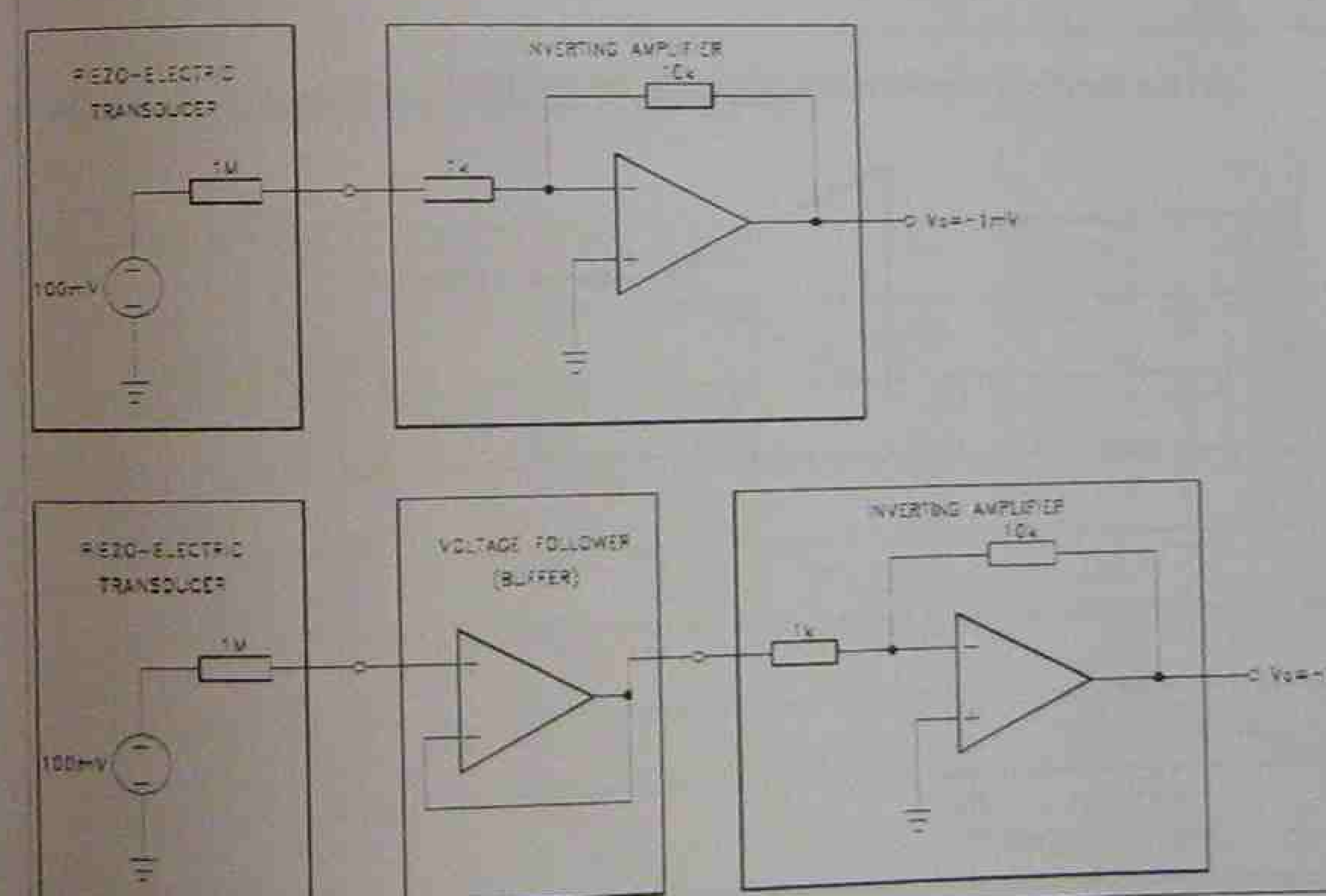
The voltage follower is a non-inverting amplifier whose voltage gain is 1. Figure 4.11 shows a voltage follower using an op amp.

Figure 4.11 Voltage follower (buffer)



Because there is no voltage difference between the non-inverting input and the inverting input, the output voltage is exactly the same as the input voltage. That is, the **voltage follower has a gain of 1**. A voltage follower's output is an exact replica of the input voltage, hence the name follower because the output follows the input voltage. It seems to be a useless type of amplifier. However, the voltage follower is widely used in interfacing circuits because **it has an extremely high (∞) resistance and has almost 0 output resistance like** other non-inverting amplifiers. Figure 4.12 shows an example of how a voltage follower could be used between a transducer and an inverting amplifier.

Figure 4.12 An example of a voltage follower application



In Figure 4.12 (a), a high internal resistance transducer is connected to an inverting amplifier. Because the output voltage of the transducer is 100mV and the voltage gain of the inverting amplifier is -10, we can expect -1V output voltage. However, if we include a 1MΩ internal resistance of the transducer, the output voltage will be drop to approximately -1mV since the 1MΩ and 1kΩ are in series. In Figure 4.12 (b), a voltage follower is placed between the transducer and the inverting amplifier. Because the input resistance of the voltage follower is ∞, there is no voltage loss between the transducer and the voltage follower. Similarly due to the zero output resistance of the voltage follower, there is no voltage loss between the voltage follower and the inverting amplifier. Therefore, the output voltage of the inverting amplifier is -1V.

It is true that we can use a non-inverting amplifier instead of an inverting amplifier to avoid signal loss due to the loading effect. However, the majority of op amp applications, such as a summing amplifier, differential amplifier, integrator ..., need an inverting configuration.

Drill Question 12

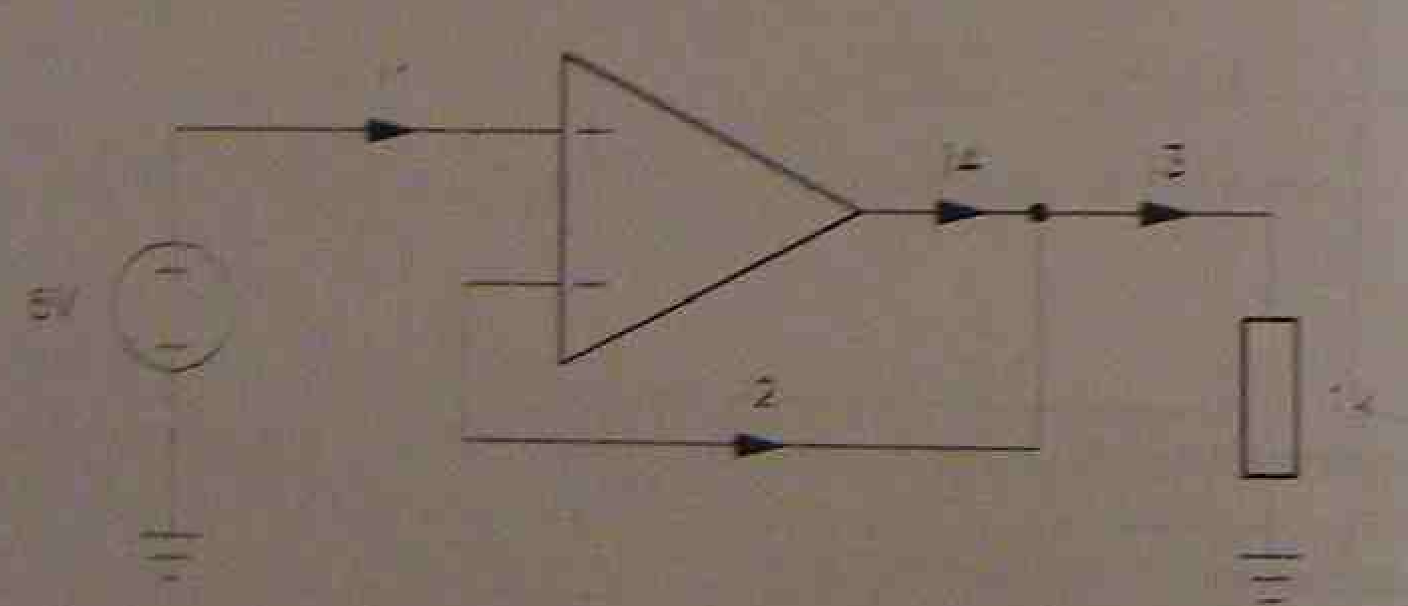
(a) Describe the characteristics of a voltage follower in terms of:

(i) Voltage gain:

(ii) Input resistance:

(iii) Output resistance

(b) For the circuit shown below, determine the currents $I_1 \sim I_4$.



4.6 Gain-Bandwidth Product

The gain-bandwidth product of an amplifier is a commonly used figure of merit. It is defined:

$$f_{unity} = A_v \times BW$$

where: f_{unity}; gain-bandwidth product or the bandwidth with unity gain (A_v=1)
A_V; closed-loop voltage gain
BW; bandwidth of the closed loop amplifier whose voltage gain is A_v

Data sheets usually list the value of f_{unity} because it equals the gain-bandwidth product. The higher the f_{unity} the larger the gain-bandwidth product of op amp.

The gain-bandwidth product gives us a fast way to compare op amps. The greater the gain-bandwidth product, the higher we can go in frequency and still have usable gain. The bandwidth (BW) for the closed-loop amplifier is:

$$BW = \frac{f_{unity}}{A_v} = f_{c(-3dB)}$$

With an op amp, there is no lower -3dB frequency because the stages are direct-coupled. Therefore, the bandwidth equals upper -3dB frequency. Figure 4.13 shows a frequency response of a typical open-loop op amp. For this op-amp, the unity gain frequency is 1MHz. Therefore, the bandwidth of closed-loop amplifiers for the different gains can be given like Figure 4.14.

Figure 4.13 A frequency response of an op amp

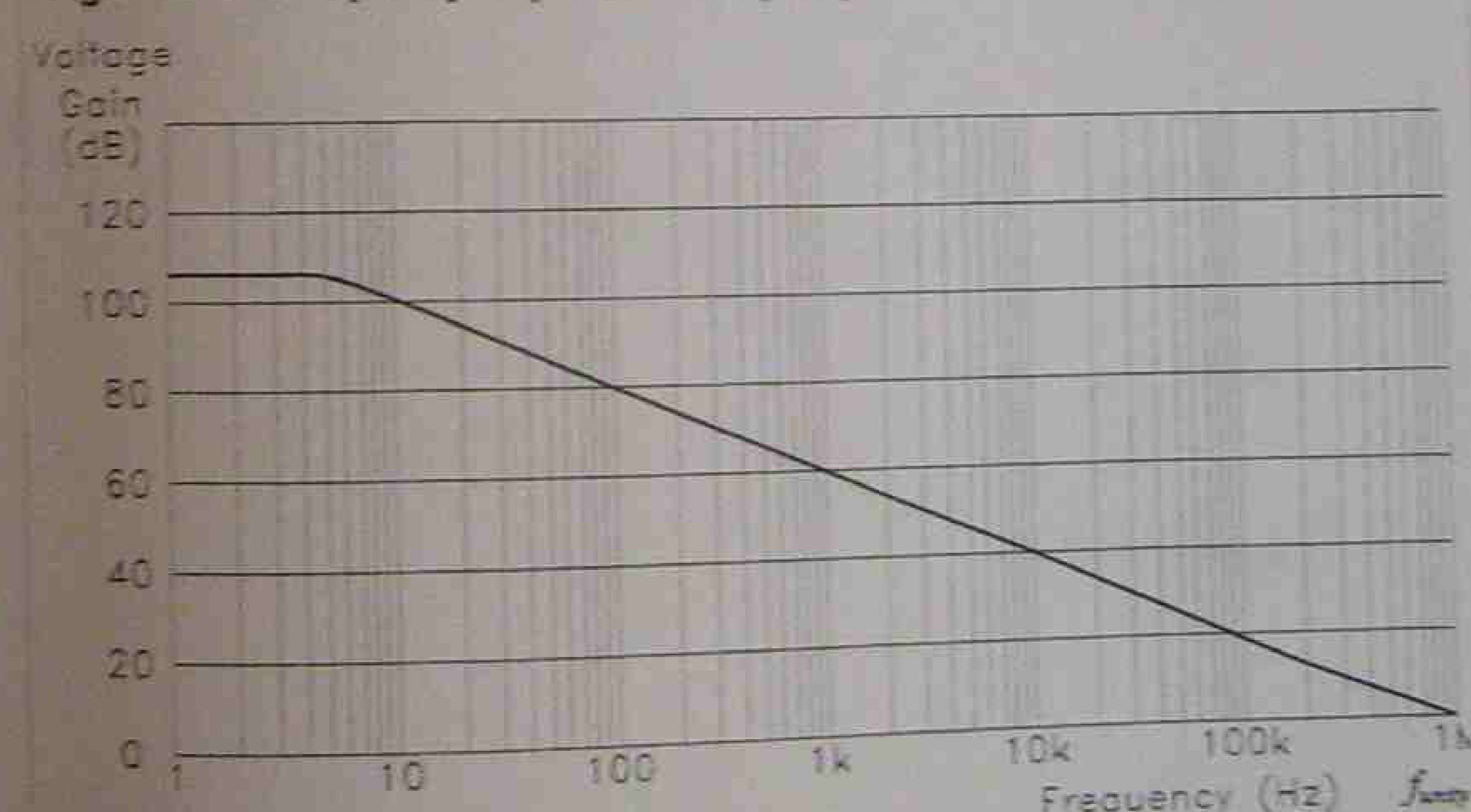
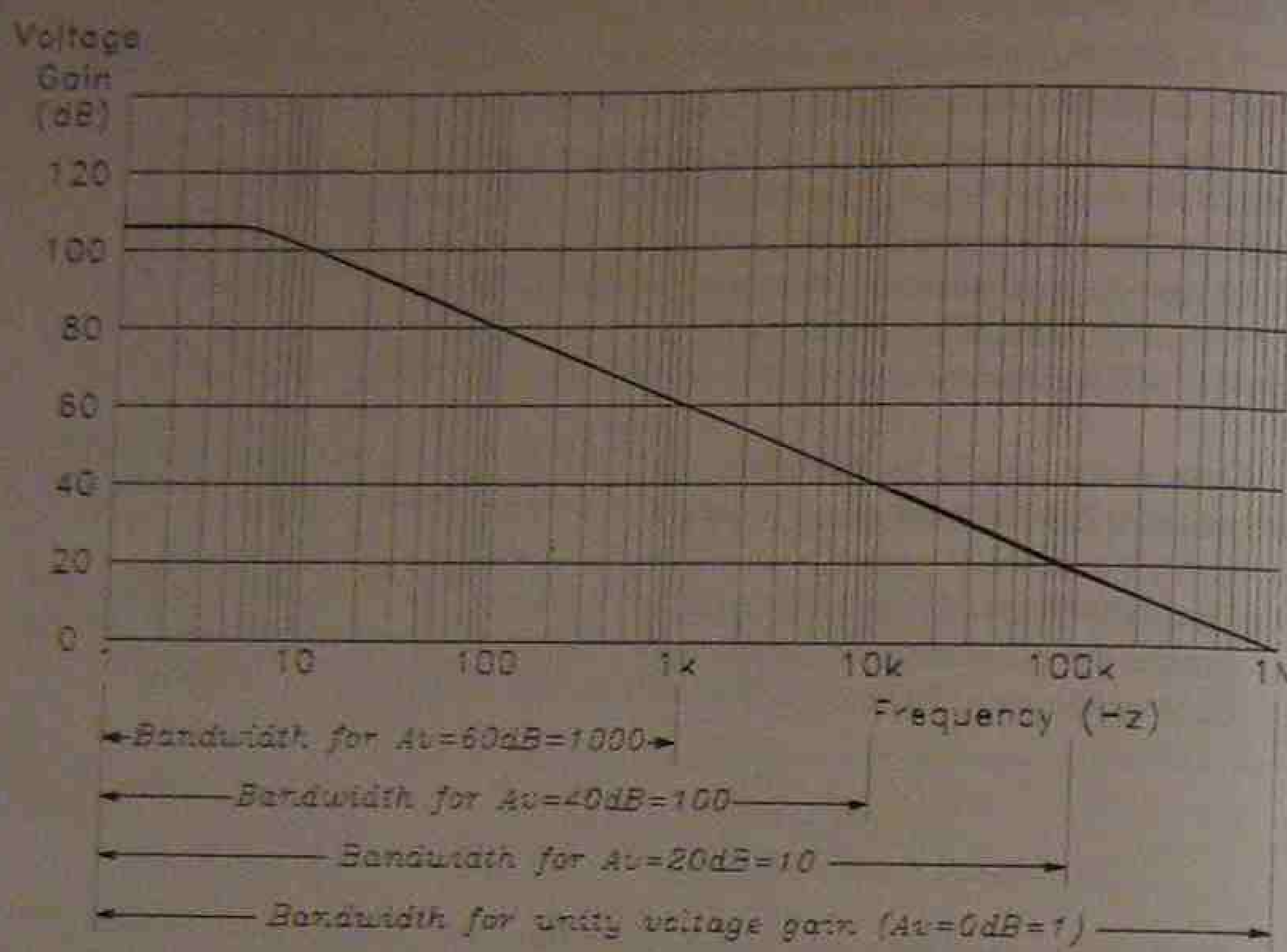


Figure 4.14 Bandwidth increases when closed-loop voltage gain decreases.



Drill Question 13.

The unity gain frequency of LF351 op amp is 4MHz. Calculate bandwidth for the each of the following values of closed-loop voltage gain using LF351.

(i) $A_V = 1000$

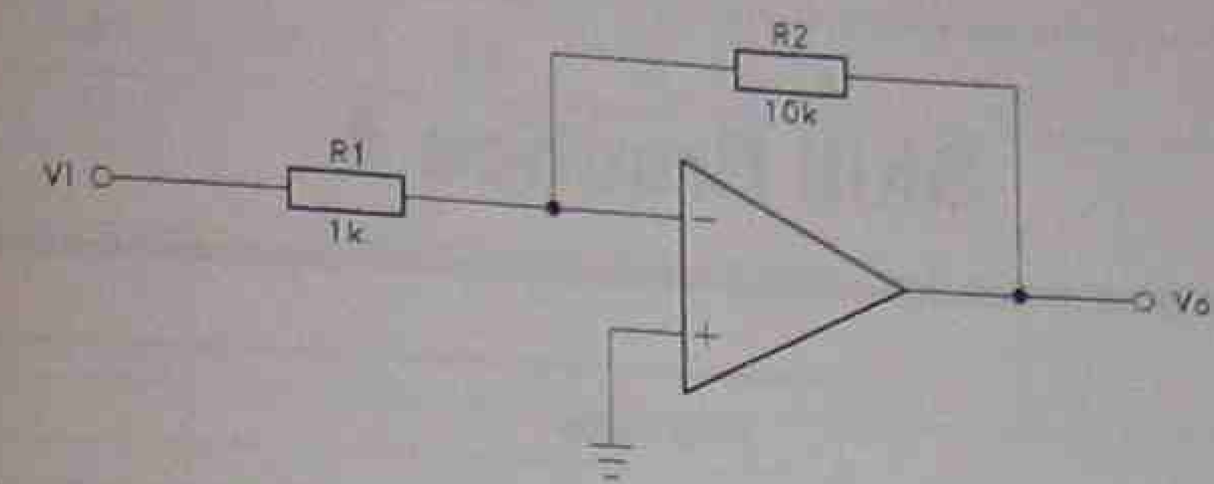
(ii) $A_V = 50$

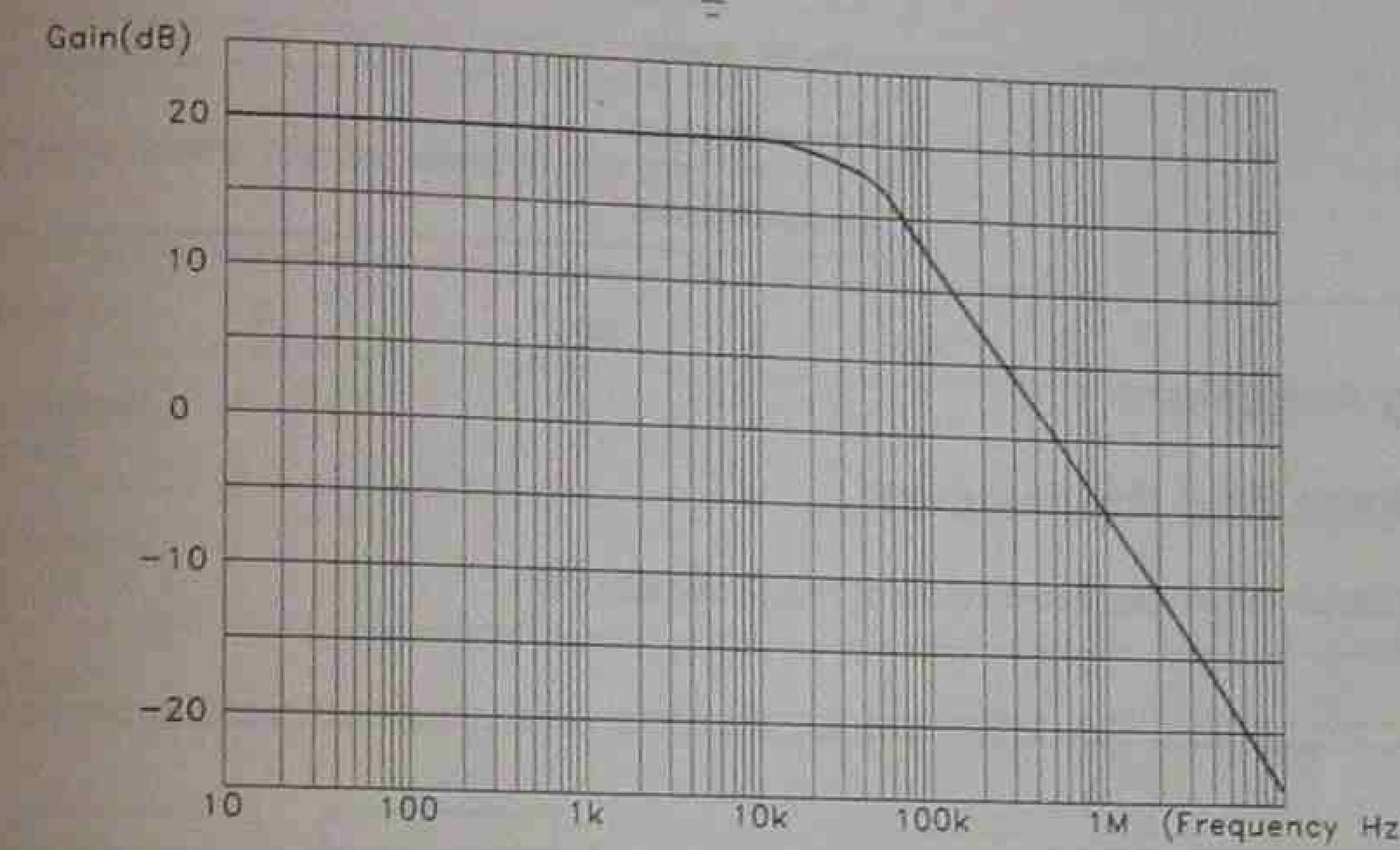
(iii) $A_V = 1$

(iv) $A_V = 20\text{dB}$

Drill Question 14

For an inverting amplifier shown below, frequency response is measured as shown below. Determine:





(i) The unity gain frequency.

(ii) Bandwidth for R_2=5k.

(iii) Bandwidth for R_2=100k.

Amplifiers 1

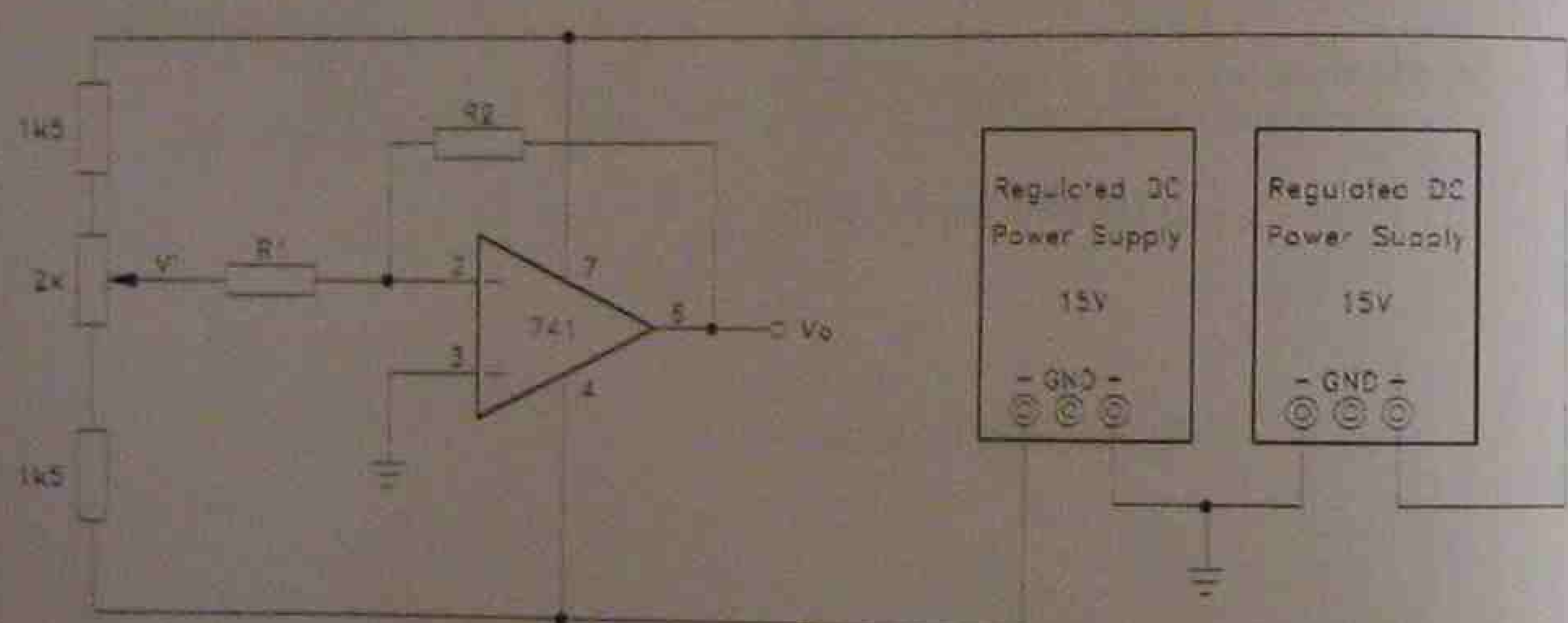
Skill Practice 4

First Name	
Family Name	
Course Number	
Student Number	

Inverting Amplifier

DC Voltage Gain Measurement

(a) Construct the circuit shown below. R1=2.2k and R2=12k



(b) Determine the theoretical voltage gain of the amplifier.

(c) Measure the output voltage for the input voltages given below.

Input Voltage (V)	-3	-1	-0.5	0	0.5	1	3	4
Theoretical Output Voltage (V)								
Measured Output Voltage (V)								
Voltage Gain								

(d) Replace R_f with 22k and repeat (b) and (c)

Input Voltage (V)	-3	-1	-0.5	0	0.5	1	3	4
Theoretical Output Voltage (V)								
Measured Output Voltage (V)								
Voltage Gain								

(e) From the previous measurements, determine:

(i) Voltage gain of the amplifiers without saturation.

(ii) Positive saturation output voltage.

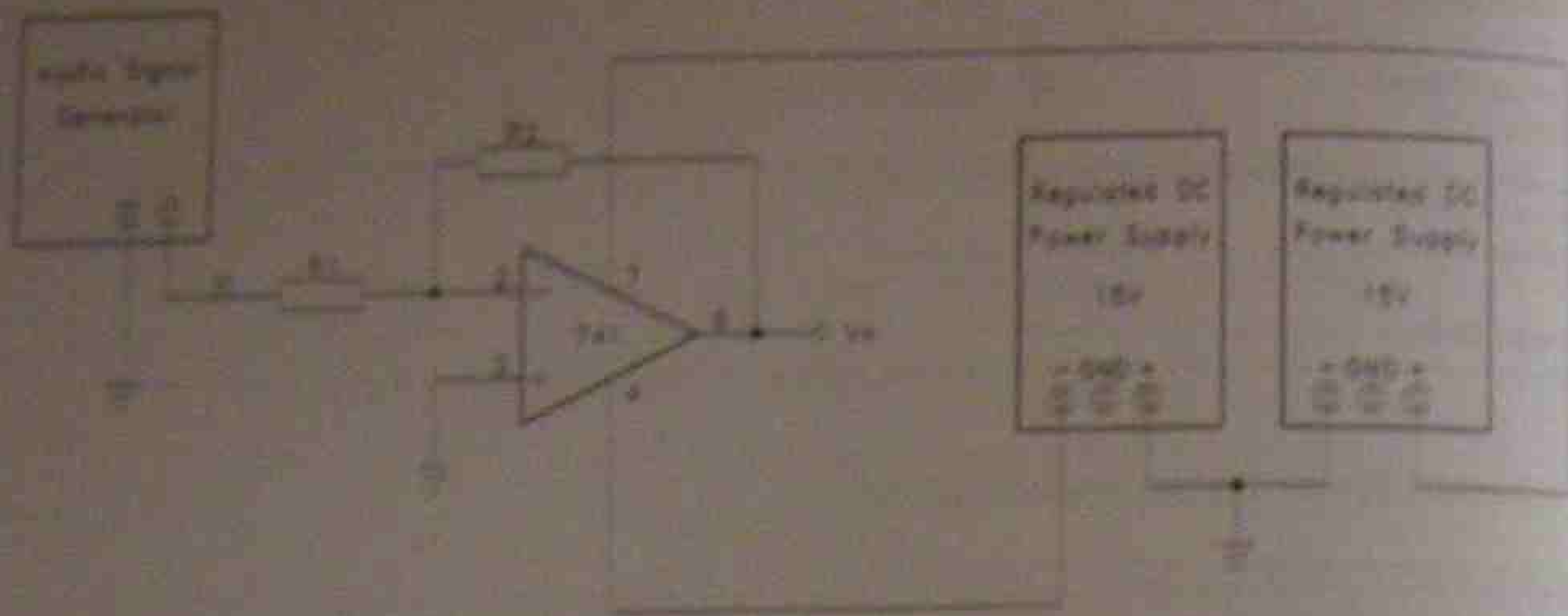
(iii) Negative saturation output voltage.

(iv) DC output offset voltages.

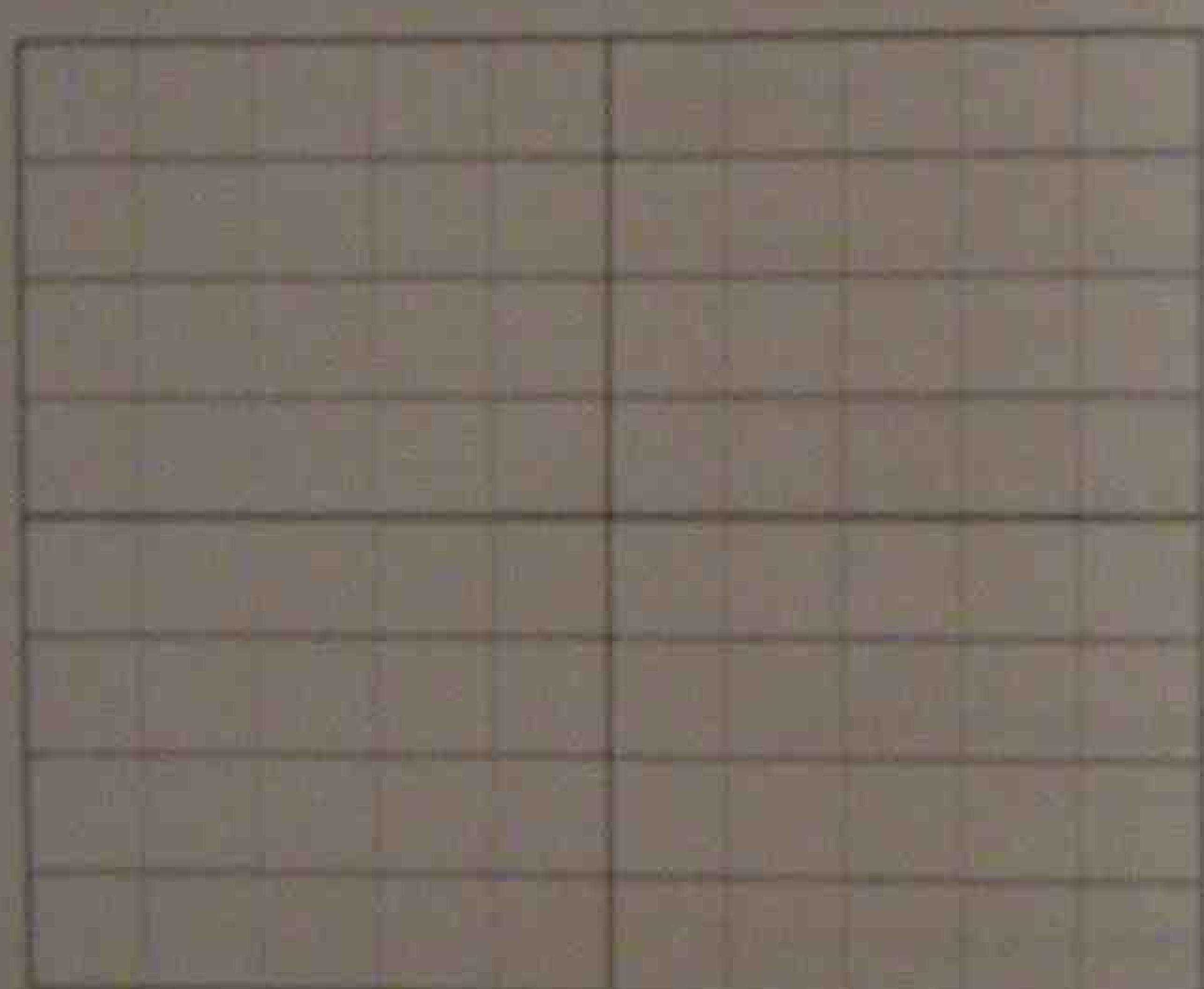
(f) Explain why it is an inverting amplifier.

AC Voltage Gain and Waveform Observations

(a) Connect an audio signal generator as shown below. R1=2.2k, R2=22k

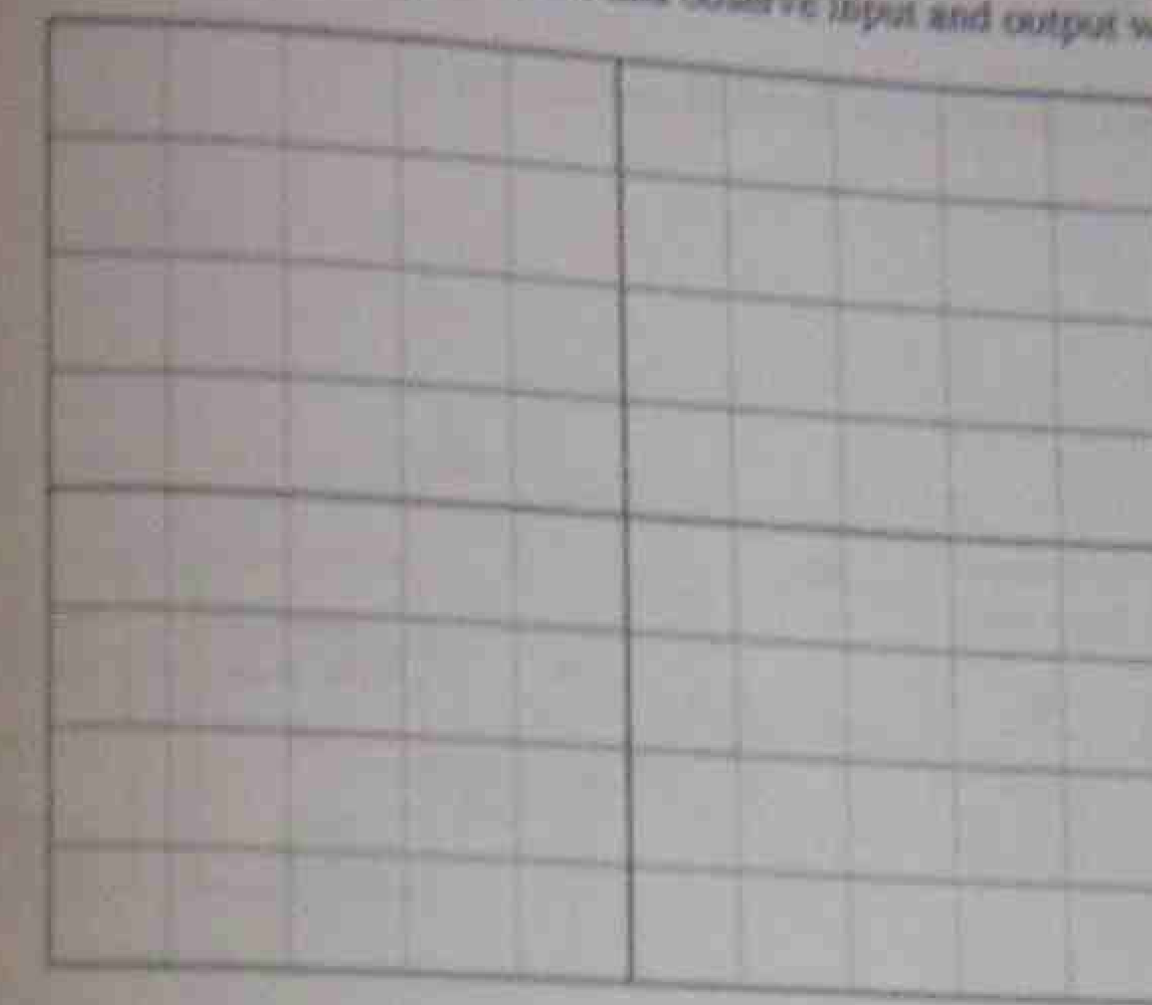


(b) Set Vi for 1kHz 1Vpp sine wave and observe both input and output voltage waveforms.
Determine the voltage gain of the amplifier form the observation.



Confirm the inverting amplification from the observation.

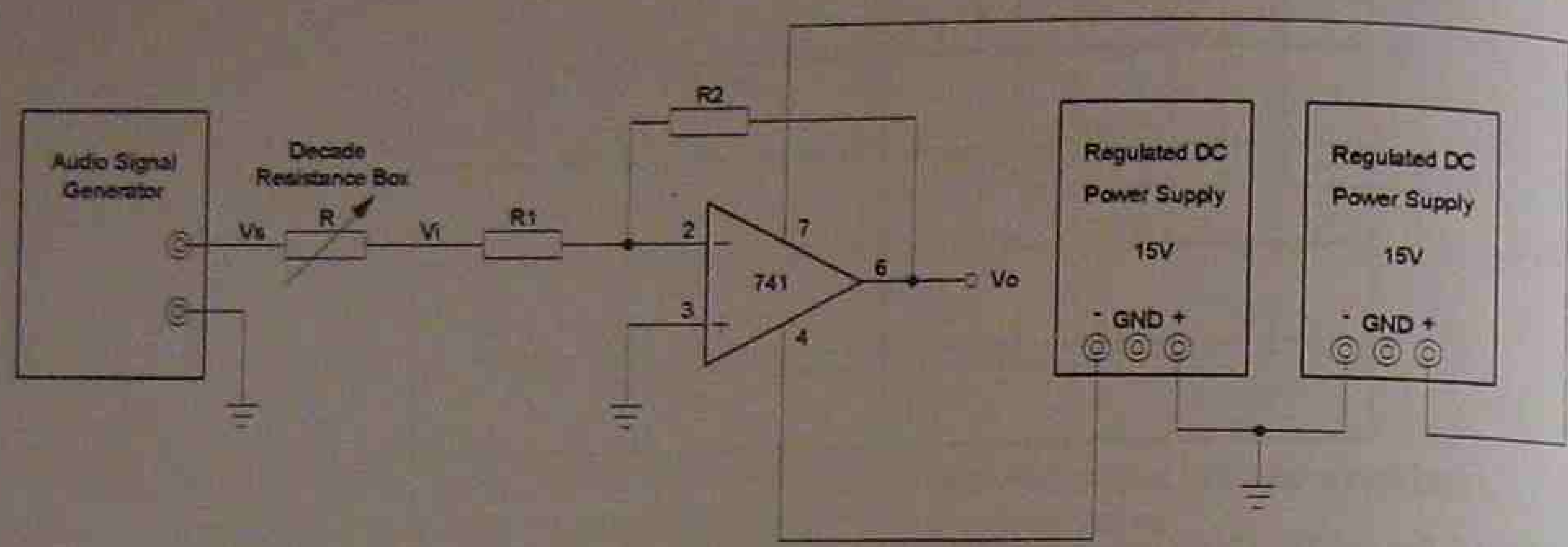
(c) Set input Vi for 1kHz 1Vpp square wave and observe input and output waveforms.



(d) Set Vi for 10kHz 1Vpp square wave and observe input and output waveforms.

Input Resistance Measurement

(a) Construct the circuit shown below. R1=2.2k and R2=12k



(b) Adjust the audio generator so that V_S is 100mVpp, 1kHz.

(c) Adjust $V_i = \frac{V_S}{2}$ by using a decade resistance box while observing Vo waveform. Vo should not be clipped.

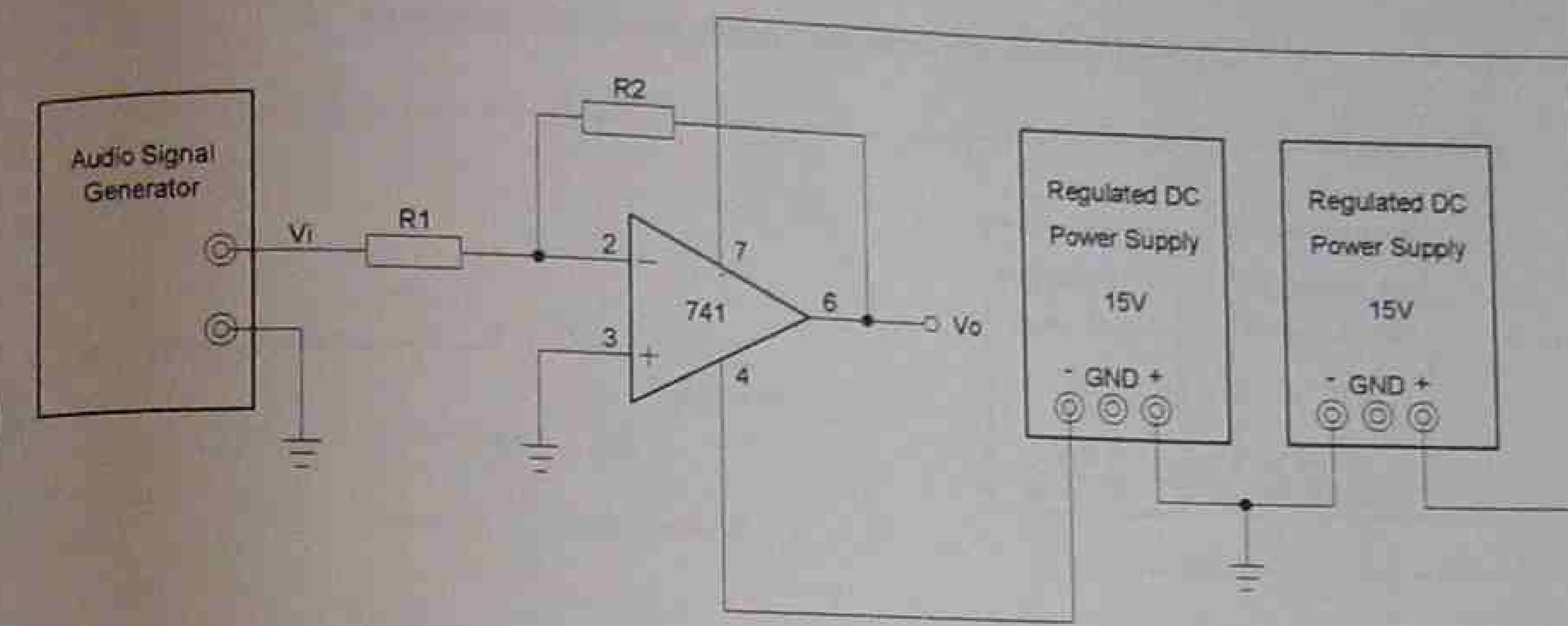
(d) Read the value of decade resistance box.

Ri = R =

(e) Repeat (a) ~ (d) for R1=4.7k and R2=47k.

Output Resistance Measurement

(a) Construct the circuit shown below.



(b) Adjust audio signal generator for 1kHz sine wave 10Vpp output. (V_o=10Vpp)

(c) Connect 2.2kΩ load to the output terminal of the amplifier and measure the load voltage V_L.

(d) Calculate the output resistance of the amplifier.

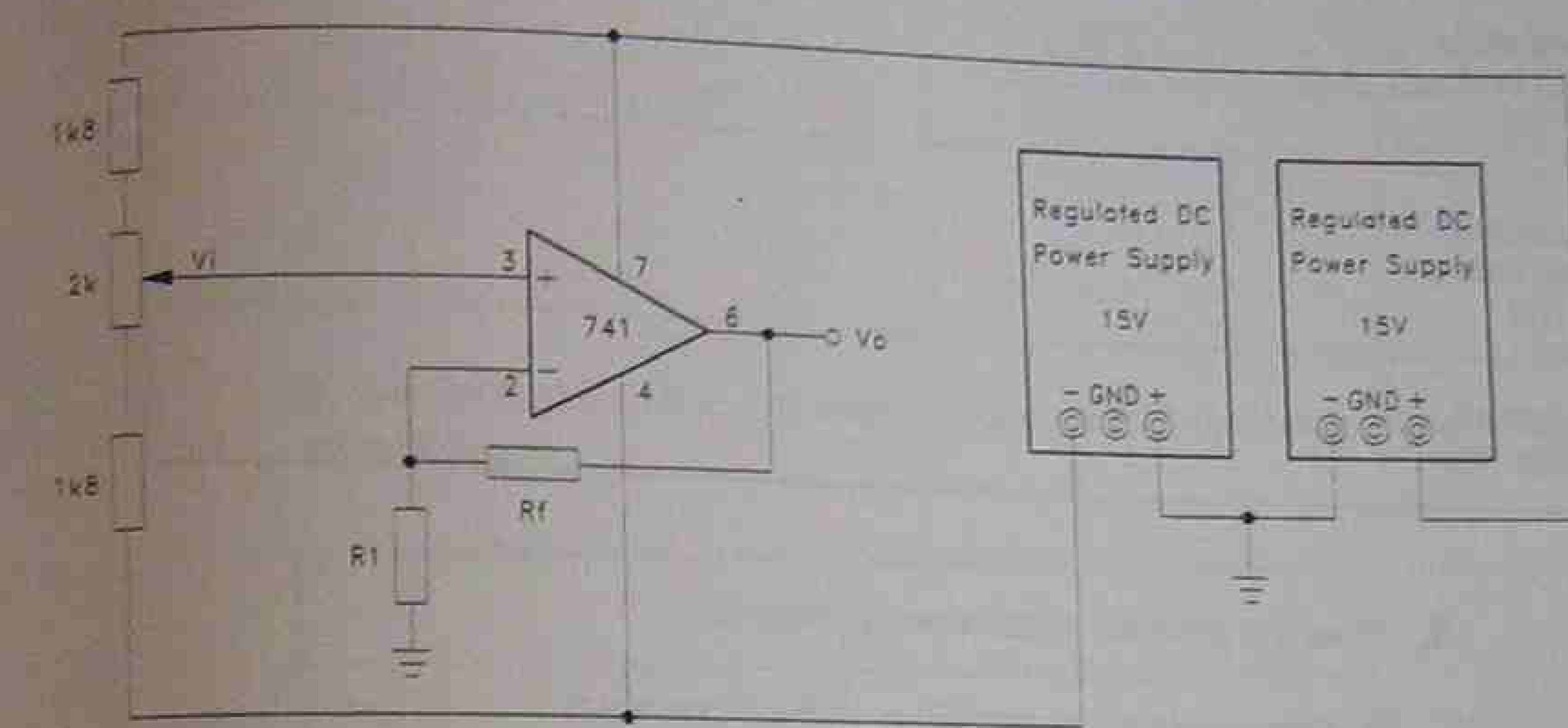
$$R_o = \frac{V_o - V_L}{V_L} \times R_L$$

(e) Repeat (a) ~ (d) for R1=4.7k and R2=47k.

Skill Practice 5

Non-inverting Amplifier

1. Construct the circuit shown below. R1=2.2k and Rf=12k



2. Determine the theoretical voltage gain of the amplifier.

$$A_v = 1 + \frac{R_f}{R_1} =$$

3. Calculate theoretical output voltage for the input voltages given below and measure the real output voltages .

Input Voltage (V)	-3	-1	-0.5	0	0.5	1	3	4
Theoretical Output Voltage (V)								
Measured Output Voltage (V)								
Measured Voltage Gain								

4. Replace R_f with 22k and repeat (2) and (3)

Theoretical Voltage Gain =

Input Voltage (V)	-3	-1	-0.5	0	0.5	1	3	4
Theoretical Output Voltage (V)								
Measured Output Voltage (V)								
Measured Voltage Gain								

5. From the previous measurements, determine:

(i) Voltage gain of the amplifiers without saturation.

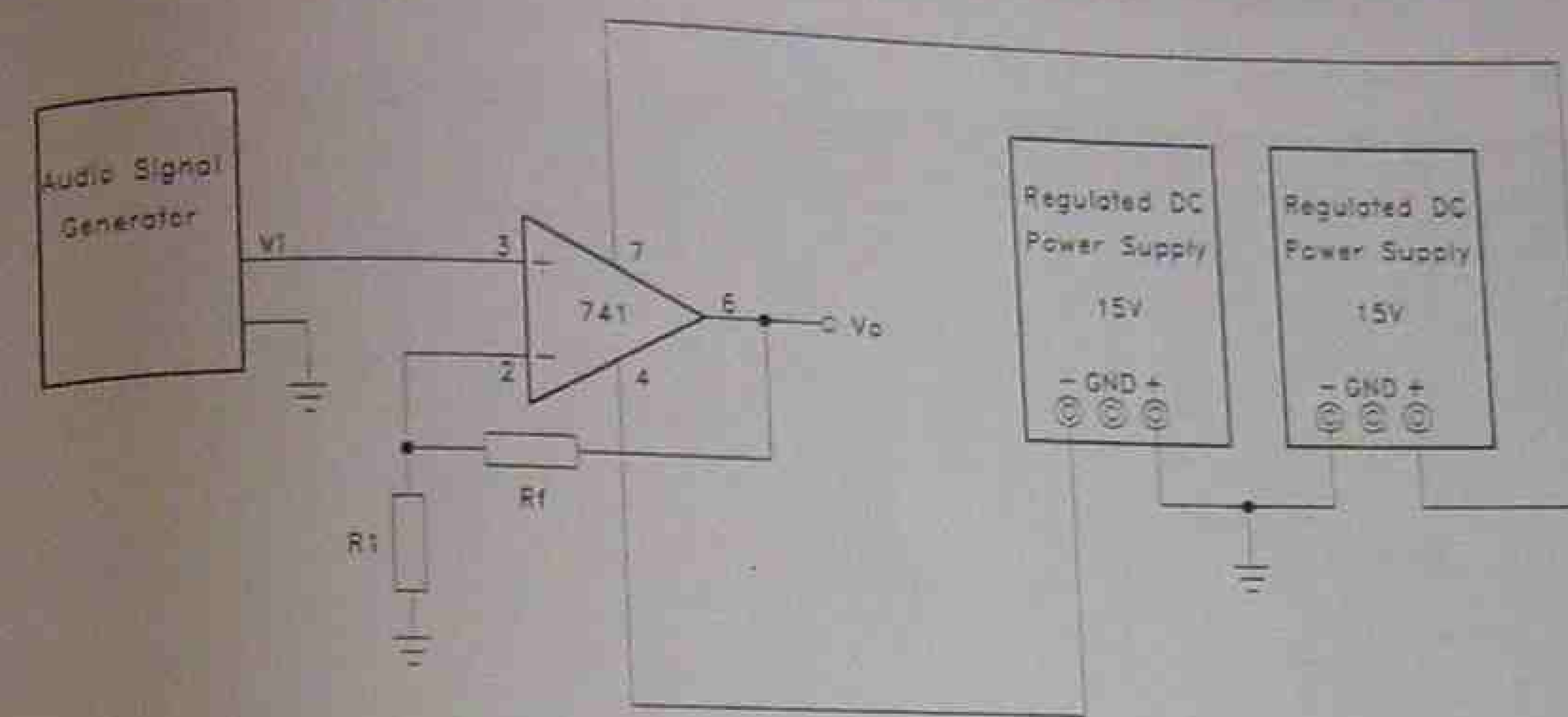
(ii) Positive saturation output voltage.

(iii) Negative saturation output voltage.

(iv) DC output offset voltage.

6. Explain why it is a non-inverting amplifier.

7. Connect an audio signal generator as shown below. R_i=2.2k, R_f=22k



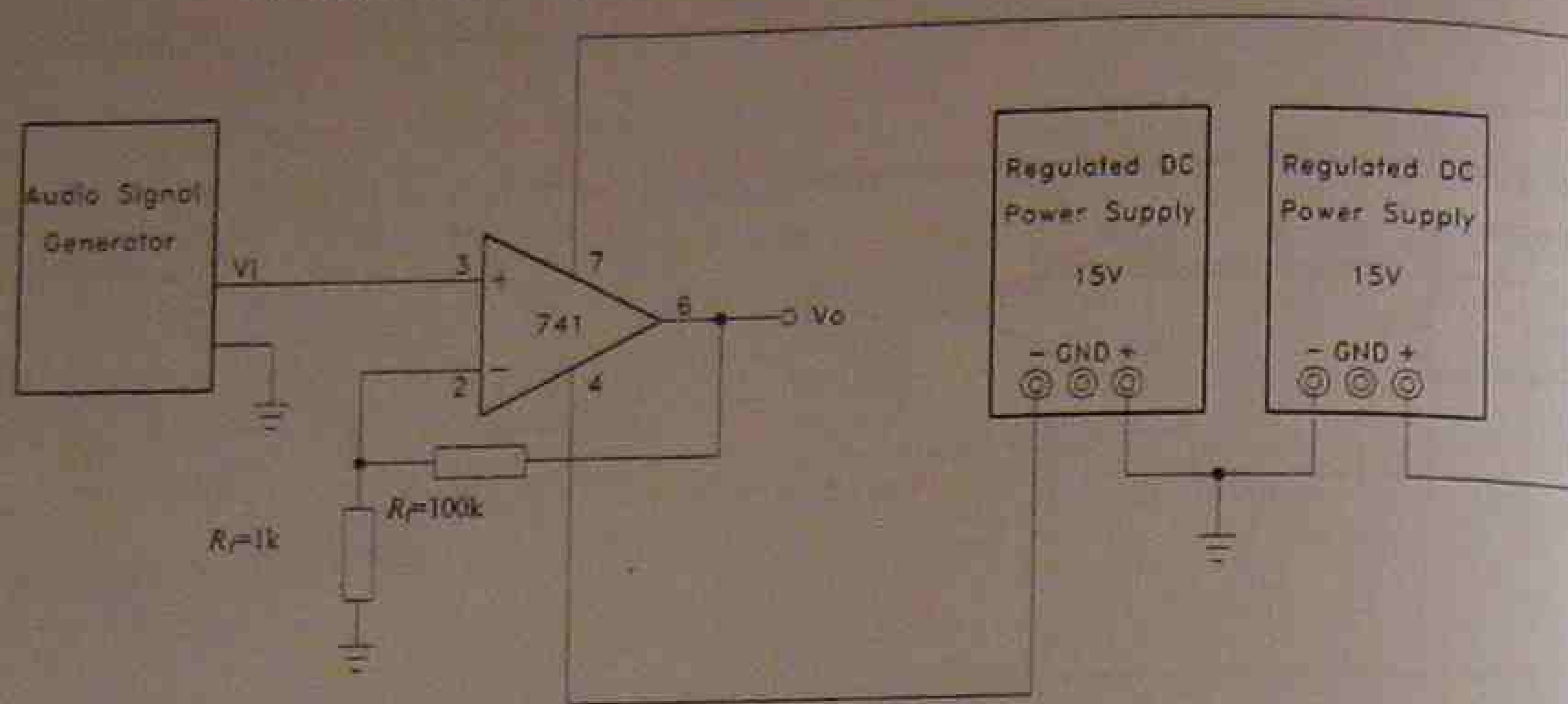
8. Set V_i for 1kHz 1Vpp sine wave and observe both input and output voltage waveforms.

Determine the voltage gain (both in ratio and dB) of the amplifier from the observation.

Confirm the non-inverting amplification from the observation.

Frequency Response and Bandwidth Measurement

1. Construct the circuit shown below.

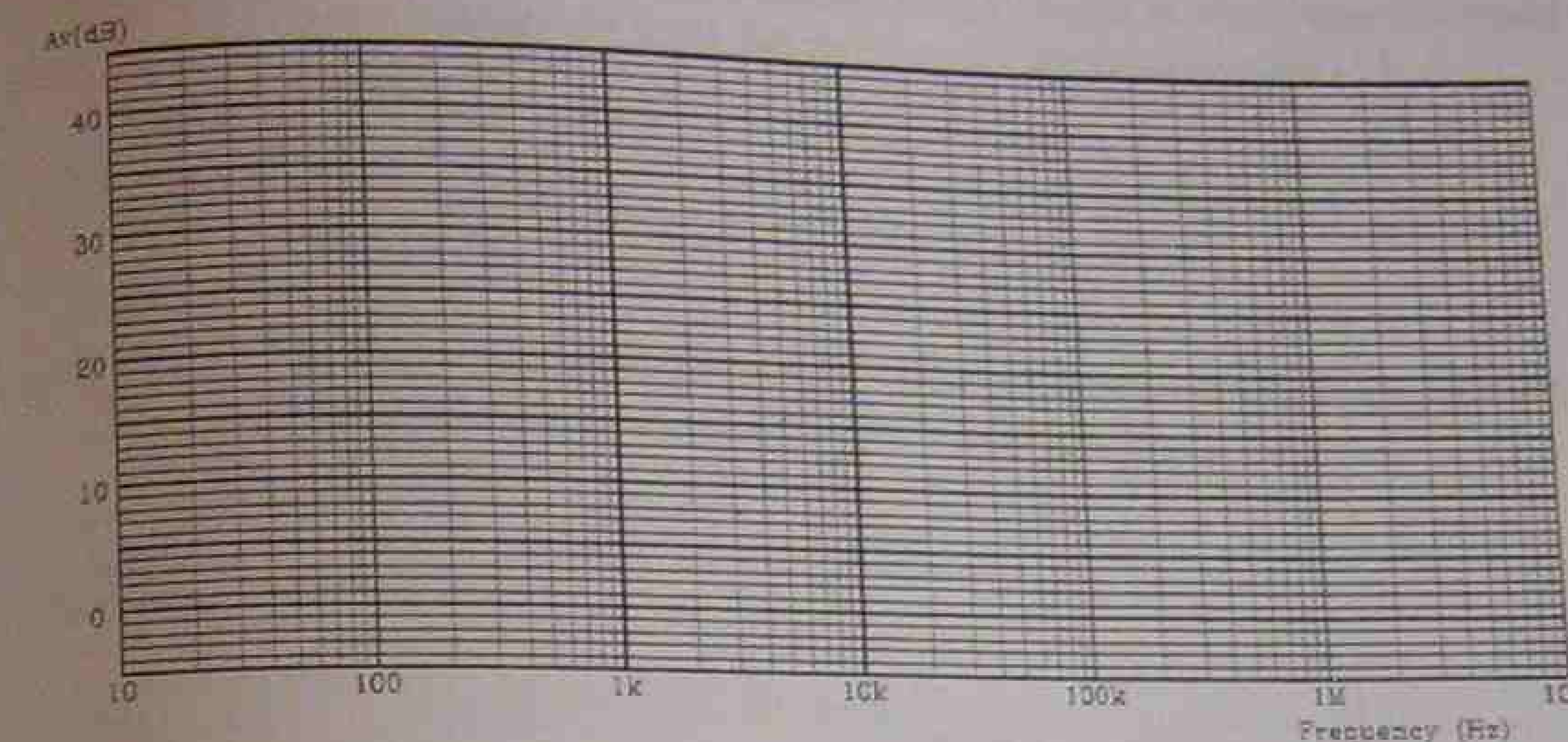


2. Determine the theoretical voltage gain of the amplifier.

3. Measure the both input and output voltages for the frequencies shown below. The output waveforms must not be distorted.
Recommended input voltage is 20 mVpp.

Frequency (Hz)	Input Voltage (mV)	Output Voltage (mV)	Voltage Gain	
			Direct Ratio	dB
10				
100				
1k				
2k				
4k				
8k				
10k				
20k				
50k				
100k				
200k				
500k				
1M				

4. Plot frequency response curve.



5. Using the measurement results and frequency response plot, determine:

Mid-band voltage gain; $A_V =$

Bandwidth (BW) or -3dB frequency; BW = $f_{-3dB} =$

Unity gain frequency; $f_{unity} =$

Verify the unity gain frequency using gain-bandwith product.

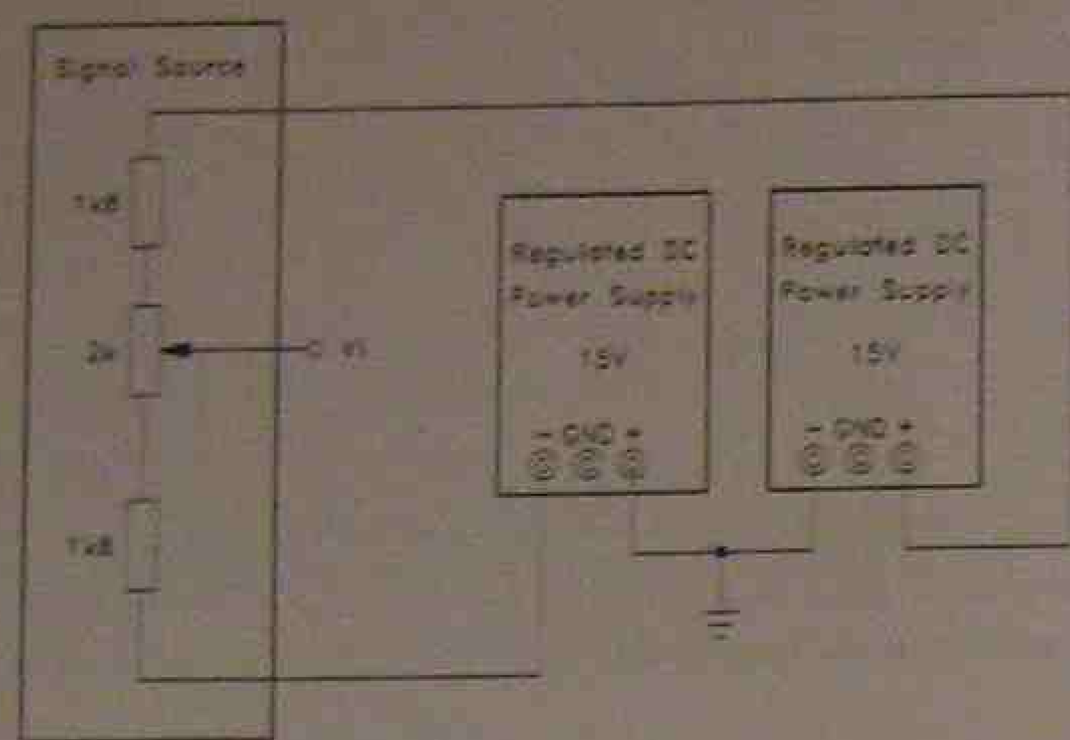
Gain-bandwidth product = $A_V \times f_{-3dB} =$

6. Replace R_f with 10k resistor and measure the -3dB frequency. Verify the unity gain frequency using gain-bandwith product.

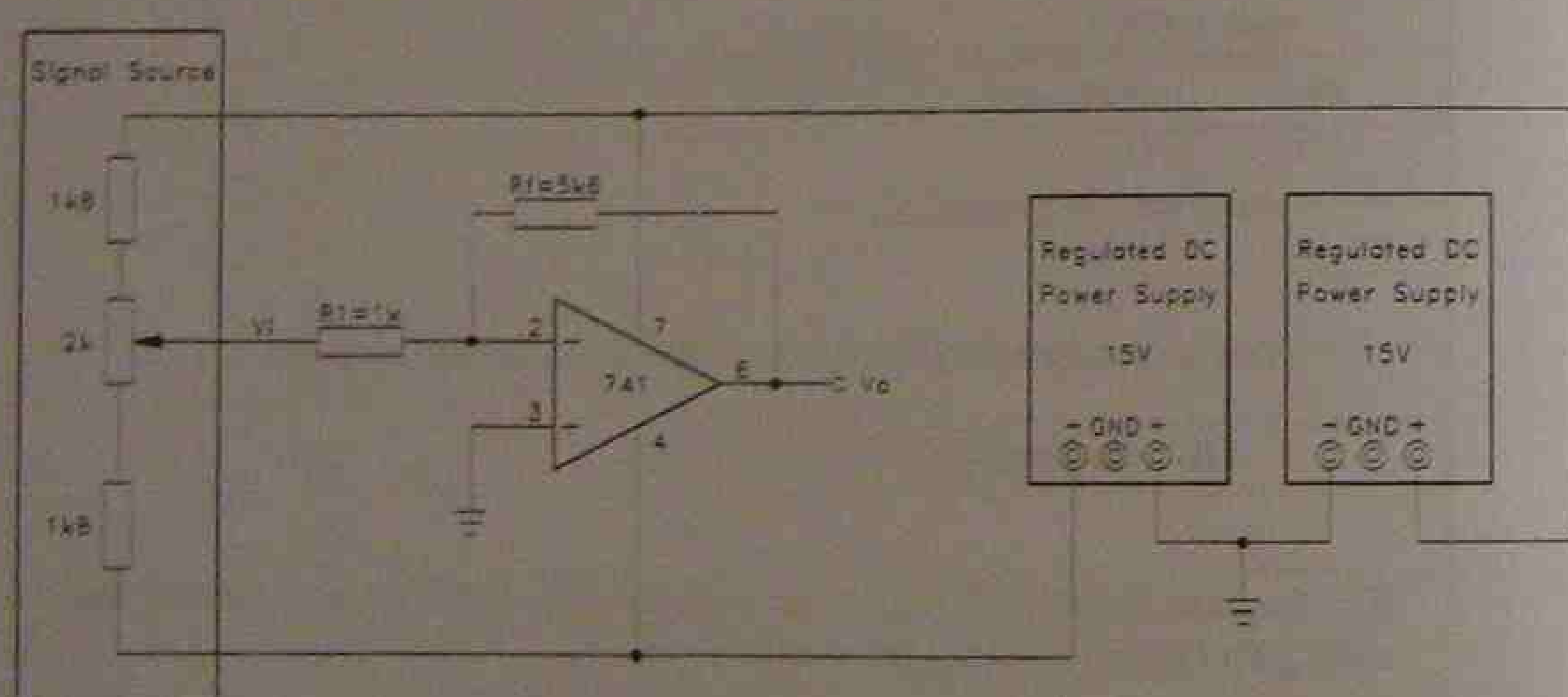
> The output voltage at -3dB frequency is :
>
> $$V_{O(-3dB)} = \frac{V_O}{\sqrt{2}} \approx 0.7V_O$$
>
> where the V_o is the output voltage for mid-band frequencies which is measured in step **3**. For example, if you measured V_o = 1.1Vpp, you just need to adjust output frequency of the audio signal generator to get $V_O = 1.1 \times 0.7 = 0.77$Vpp for V_i = 100mVpp.

Voltage Follower

1. Construct the circuit shown below and adjust Vi=1V.



2. Connect an inverting amplifier as shown below. Do not change the pot setting.



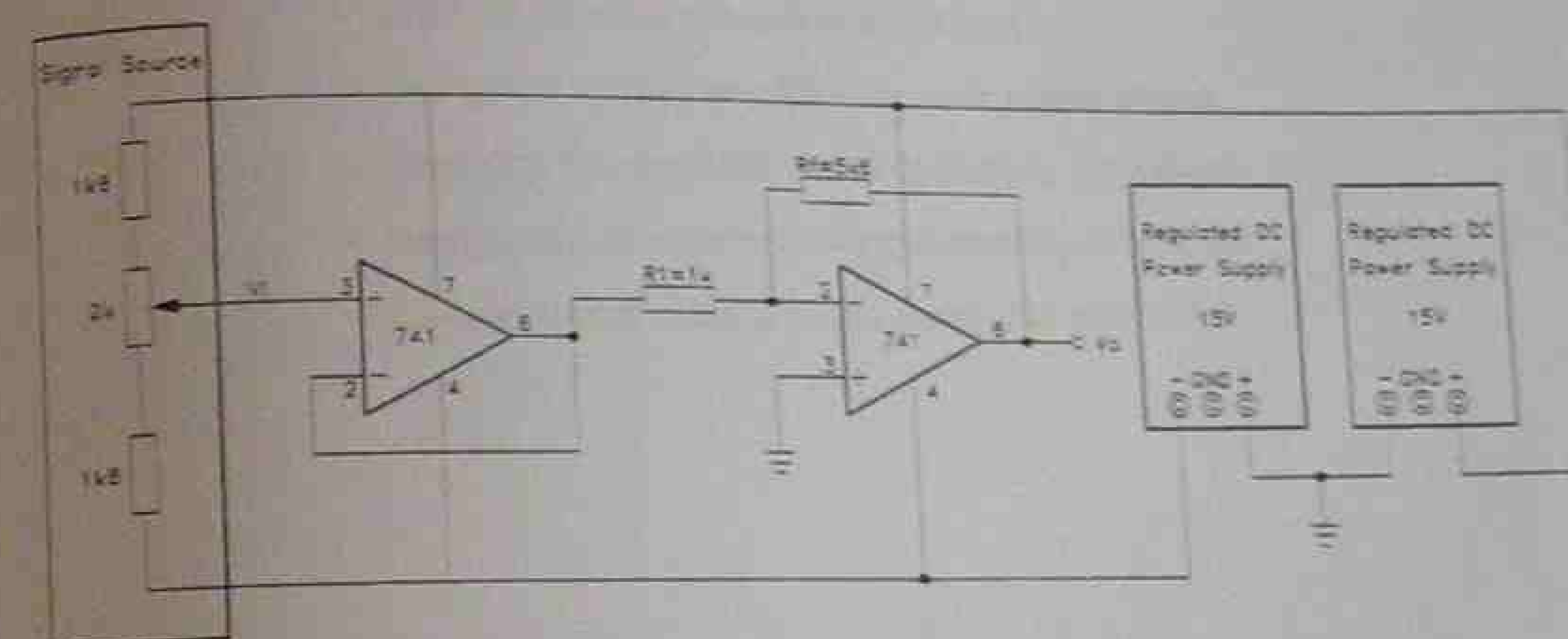
(i) Calculate theoretical voltage gain of the inverting amplifier.

(ii) Calculate the theoretical output voltage of the amplifier. (Ignore the output resistance of the signal source.)

(iii) Measure the real output voltage Vo.

(iv) Explain briefly why measured output voltage is lower than theoretical output voltage.

3. Insert voltage follower between the signal source and inverting amplifier as shown below and measure the output voltage Vo.



(i) Measure Output Voltage Vo =

(ii) Compare the theoretical output voltage and measured output voltage.

(iii) Explain the role of buffer in this application.

5

Summing Amplifier

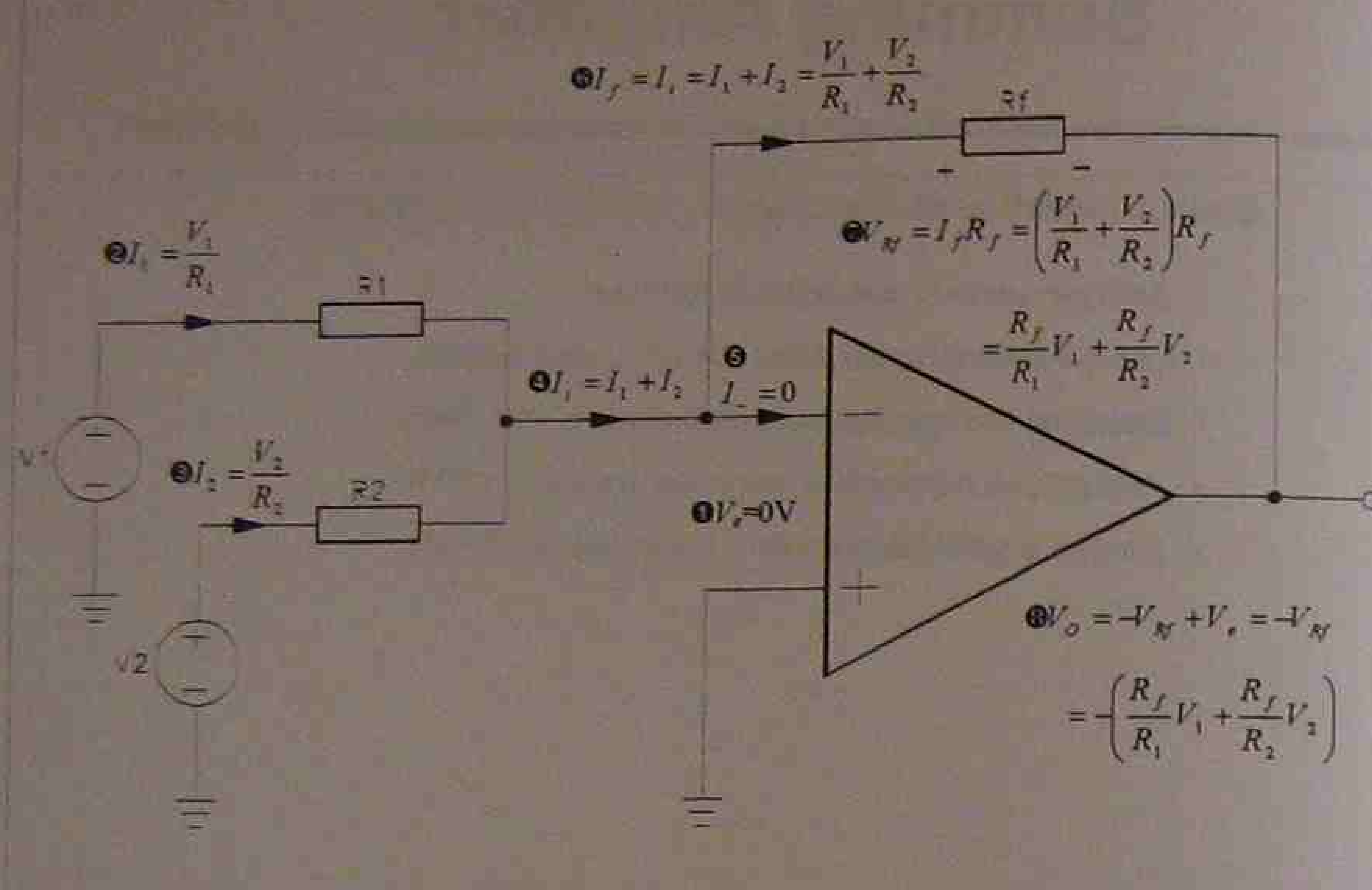
Upon completion of this chapter, you should be able to:

- Analyse various summing amplifiers.
- Design summing amplifier for your applications.
- Describe the applications of aumming amplifiers.
- Design D/A converters using summing amplifiers.
- Design signal mixers using summing amplifiers.

5.1 Summing Amplifier

Another advantage of an inverting amplifier is its ability to handle more than one input at a time. Figure 5.1 shows a summing amplifier. The analysis of a summing amplifier is exactly the same as the analysis of an inverting amplifier.

Figure 5.1 Analysis of a summing amplifier



For the summing amplifier in Figure 4.3, if R_1=R_2=R the output voltage is:

$$V_O = -\frac{R_f}{R}(V_1 + V_2)$$

Or if R_1=R_2=R_f then:

$$V_O = -(V_1 + V_2)$$

The output voltage is the sum of all the input voltages. Although only two inputs are shown, more than two voltages can be summed. For a 3 input summing amplifier in Figure 5.2, the output voltage is:

$$V_O = -\left(\frac{R_f}{R_1}V_1 + \frac{R_f}{R_2}V_2 + \frac{R_f}{R_3}V_3\right)$$

Figure 5.2 A three input summing amplifier

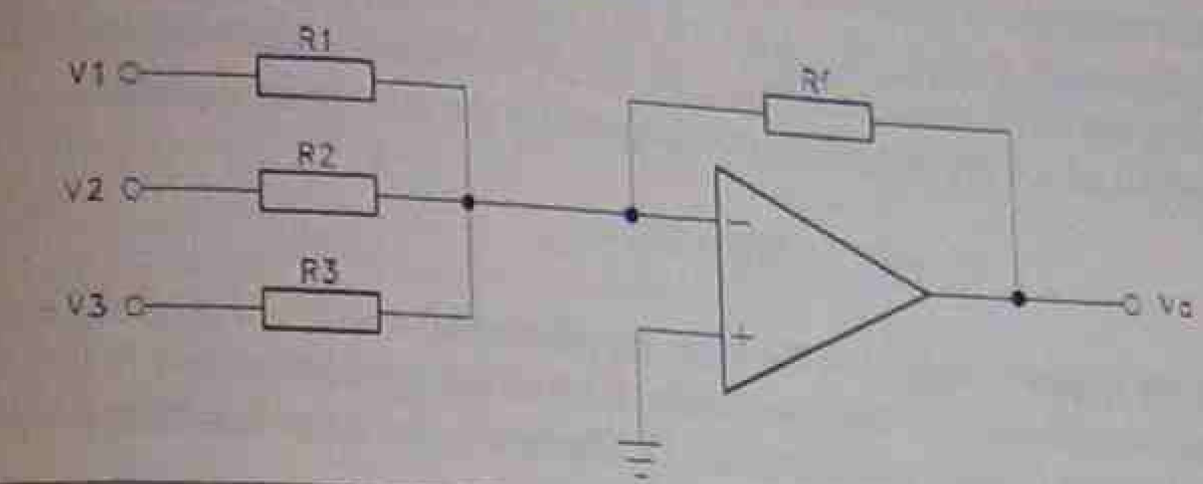
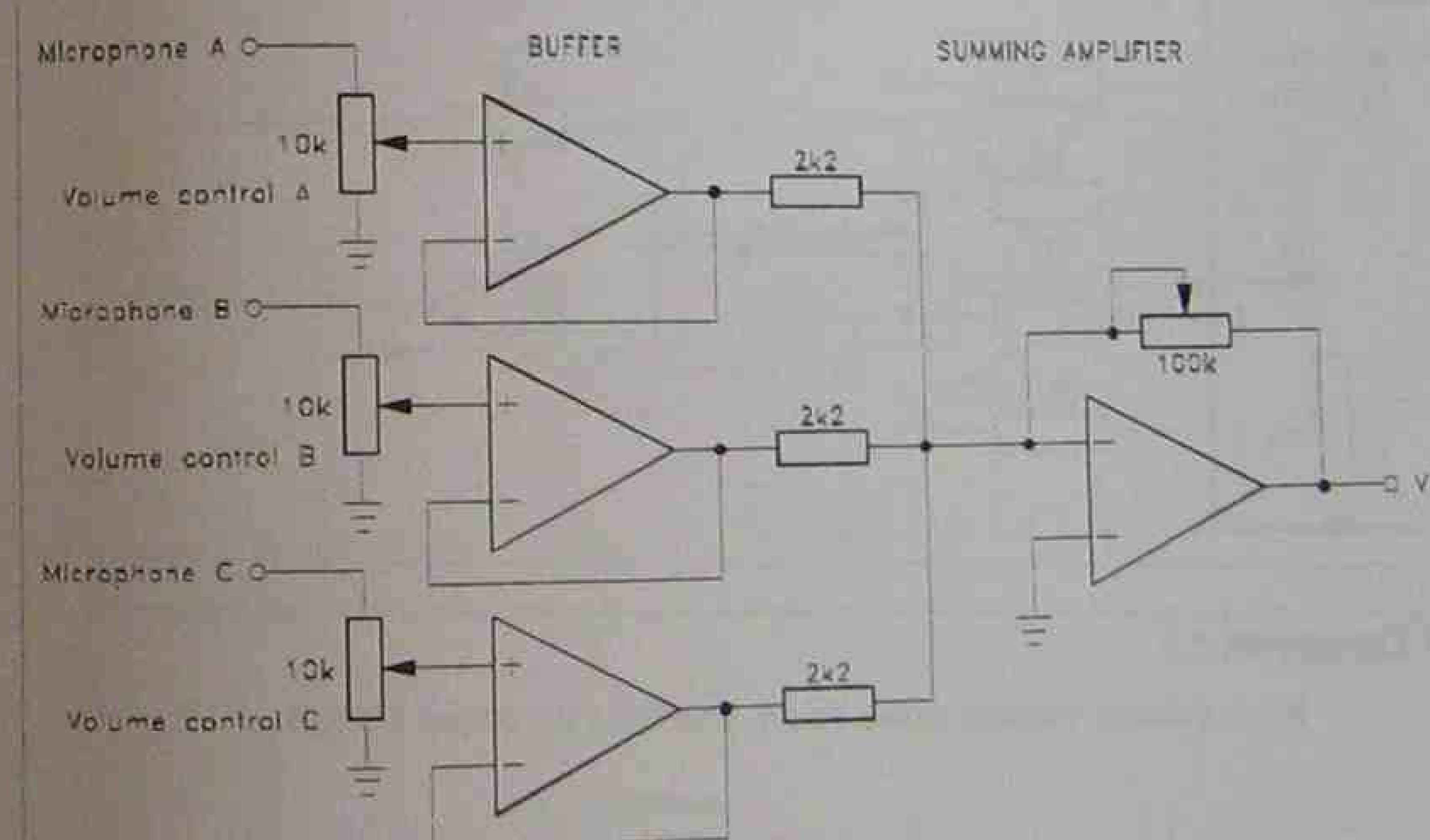


Figure 5.3 shows a convenient way to additively mix three microphone signals with a gain control. Three 10k potentiometers allow us to set the level of each input and voltage followers (buffers) are employed to avoid signal attenuation between the potentiometers and summing amplifier. The summing amplifier has a voltage gain controller using a 100k variable resistor. With this variable resistor, we can adjust the voltage gain from 0 to 45.6. Therefore, it is working as a total volume control.

Figure 5.3 A microphone mixer using a summing amplifier



Another example of summing amplifier application is a 4 bit digital to analogue (D/A) converter shown in Figure 4.16. The digital word is presented in a variety of codes, the most common being pure binary or binary-coded-decimal (BCD). The output of a 4 bit D/A converter is given by the following equation:

$$V_O = 8V_A + 4V_B + 2V_C + V_D$$

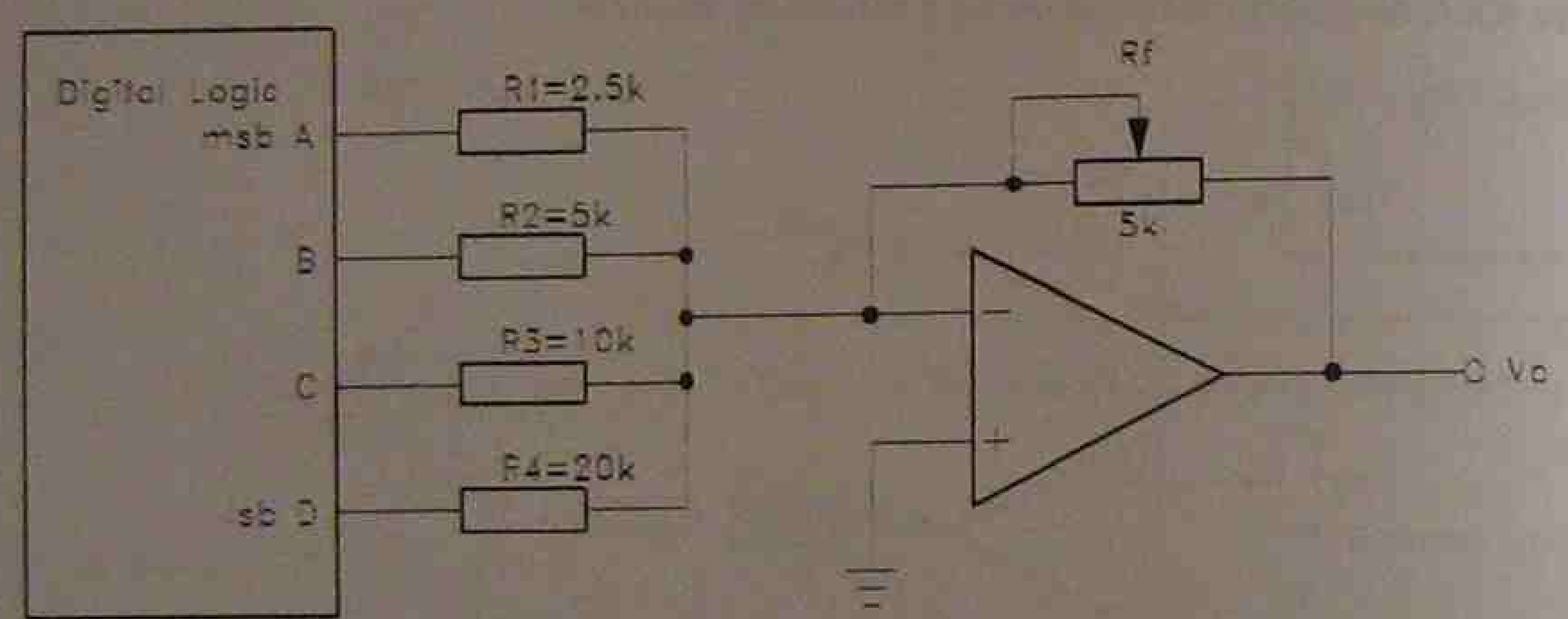
where the digital output A is the most significant bit and the digital output D is the least significant bit. In other words, the voltage gain for signal A is 8 times bigger than the voltage gain for signal D. Therefore, the input resistance ratio is

R1 : R2 : R3 : R4 : = 1 : 2 : 4 : 8

since the voltage gain is determined by $\frac{R_f}{R_{in}}$.

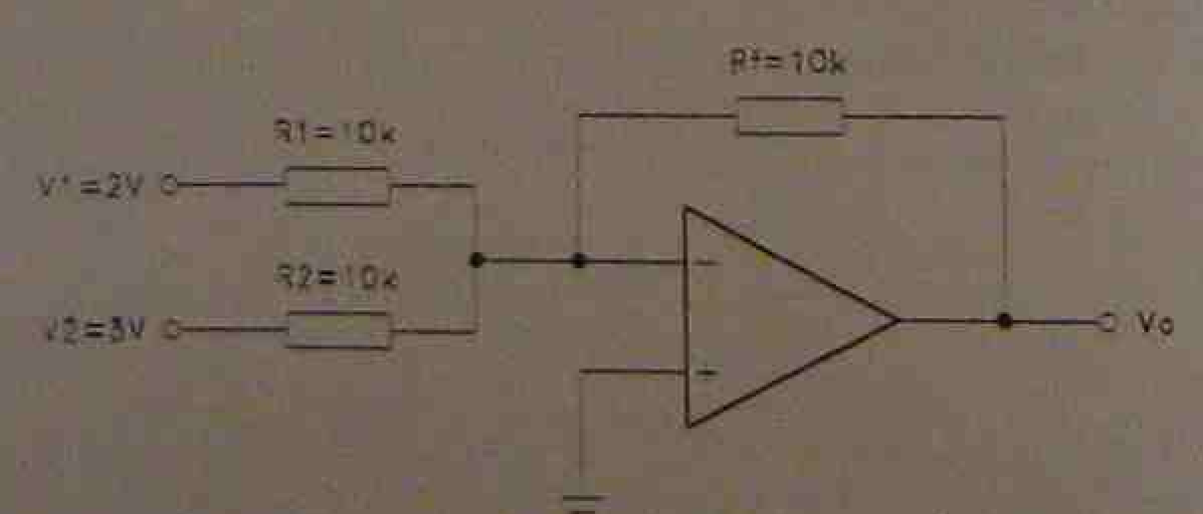
In Figure 5.4, a 5k variable resistor is employed to adjust the output voltage level.

Figure 5.4 A 4bit D/A converter



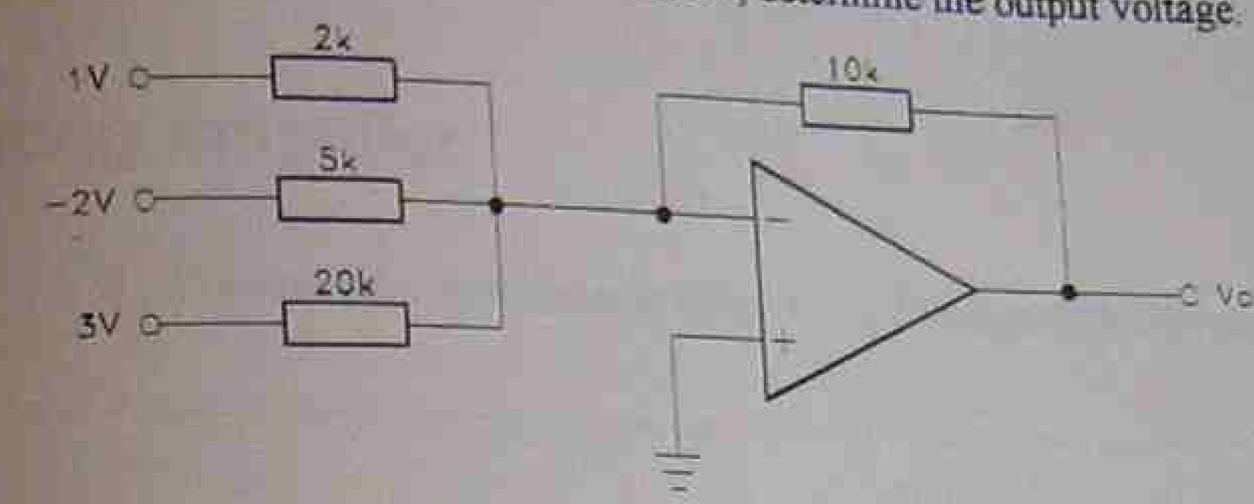
Drill Question 13

For the circuit diagram shown below, determine the output voltage.



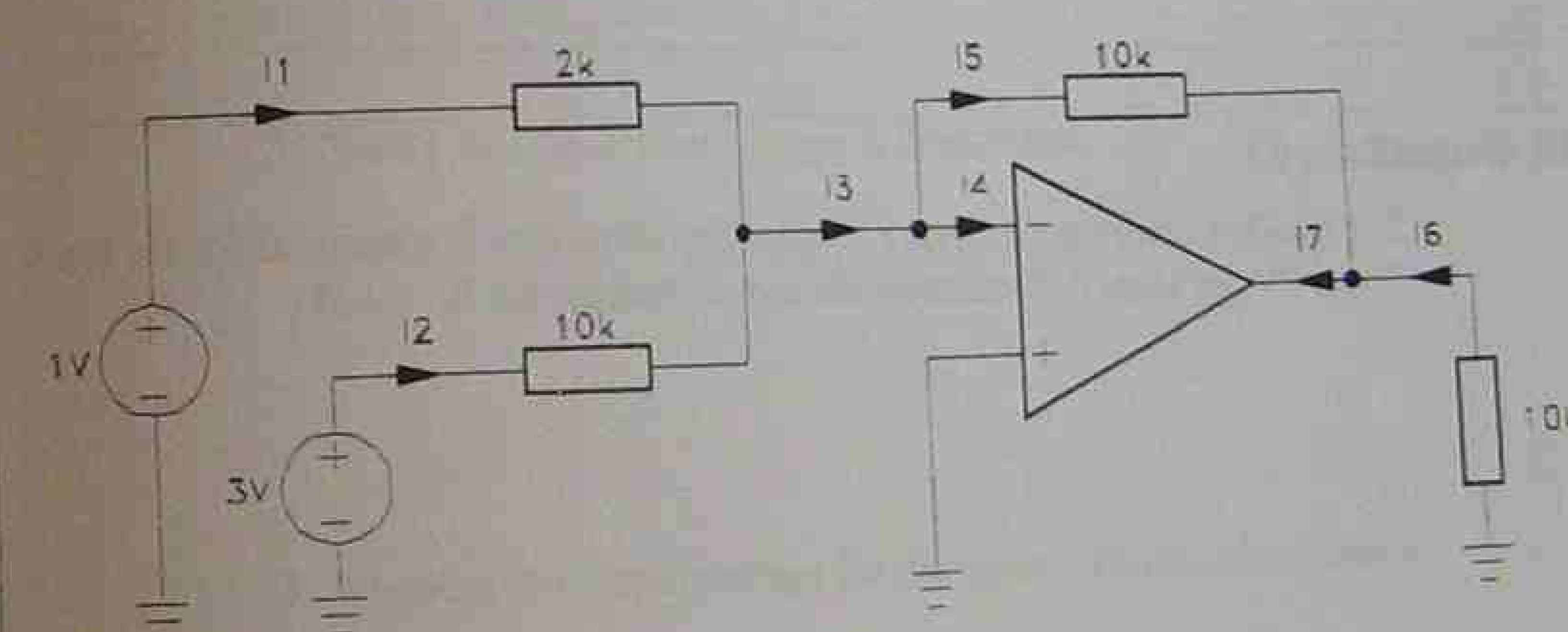
Drill Question 14

For the circuit diagram shown below, determine the output voltage.



Drill Question 15

For the circuit diagram shown below, determine the currents I1 ~ I6.



Drill Question 16

Design a summing amplifier to satisfy the following equation.

$$V_O = -0.2(V_A + 2V_B + 4V_C)$$

$R_f = 10k$

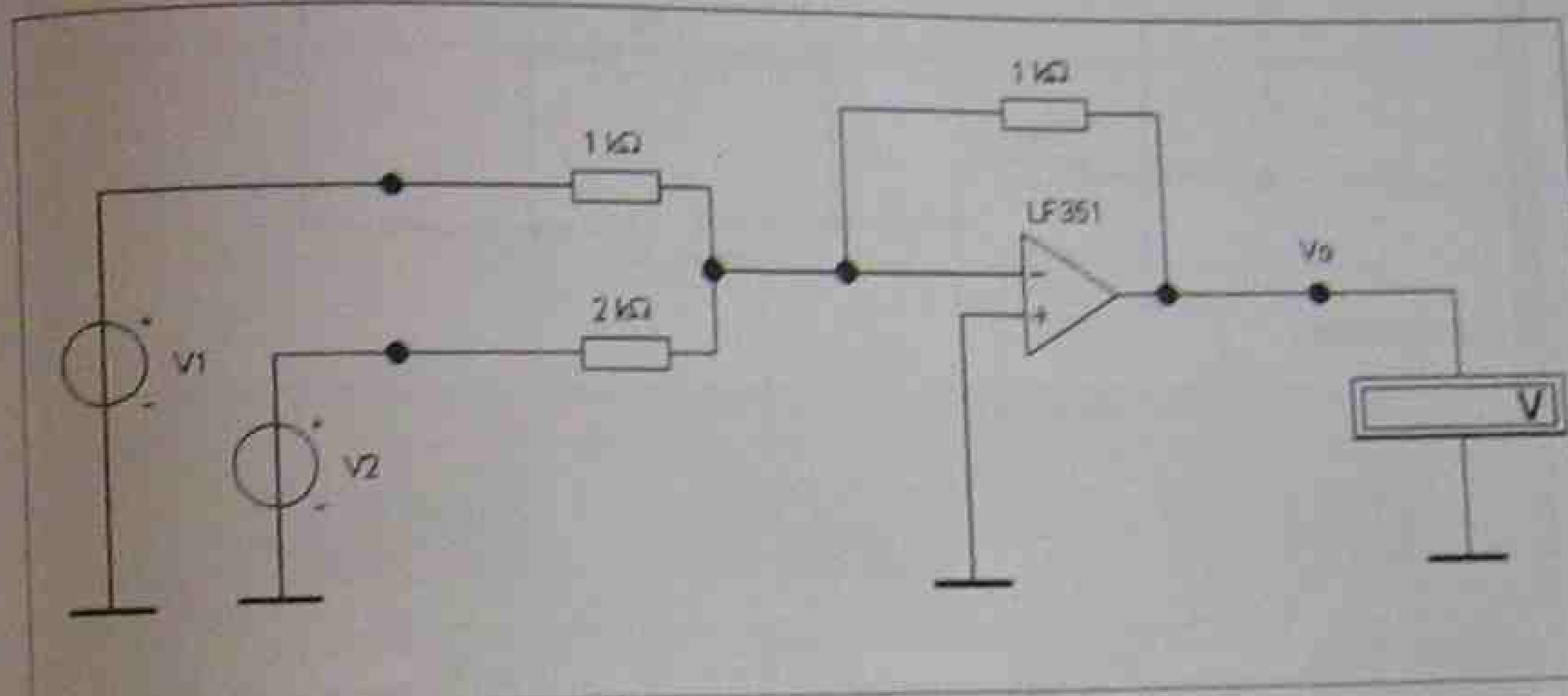
Drill Question 17

For the D/A converter in Figure 5.4, the digital voltage is measured 4V for logic 1 and 0V for logic 0. Determine the output voltage for $R_f = 2.5k$.

Skill Practice 6

Summing Amplifier

1. Construct the circuit shown below using Electronics Workbench.



2. Verify that the output voltage is determined:

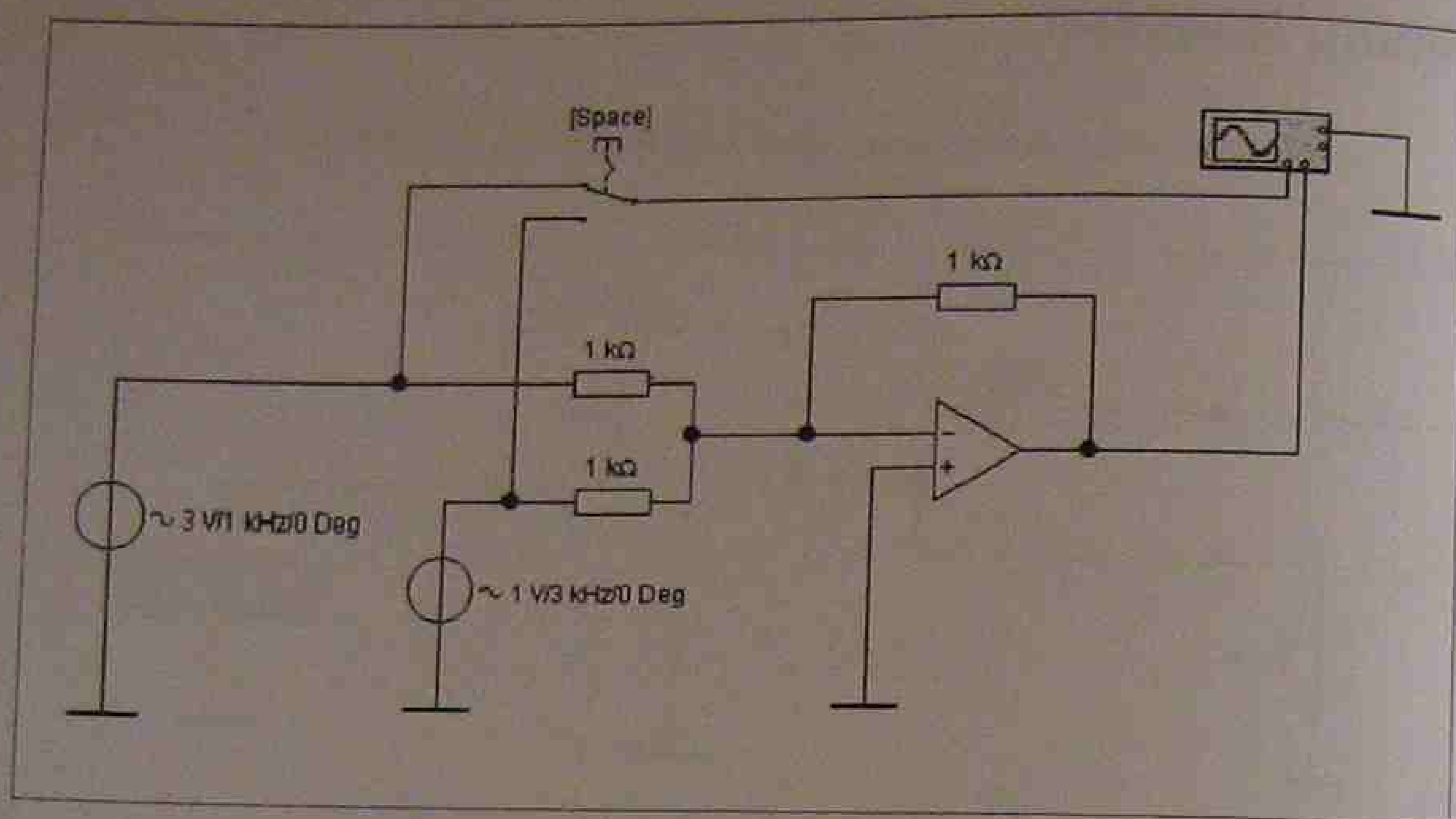
$$V_o = -(V_1 + 0.5V_2)$$

3. Measure the output voltage for the input voltages shown below.

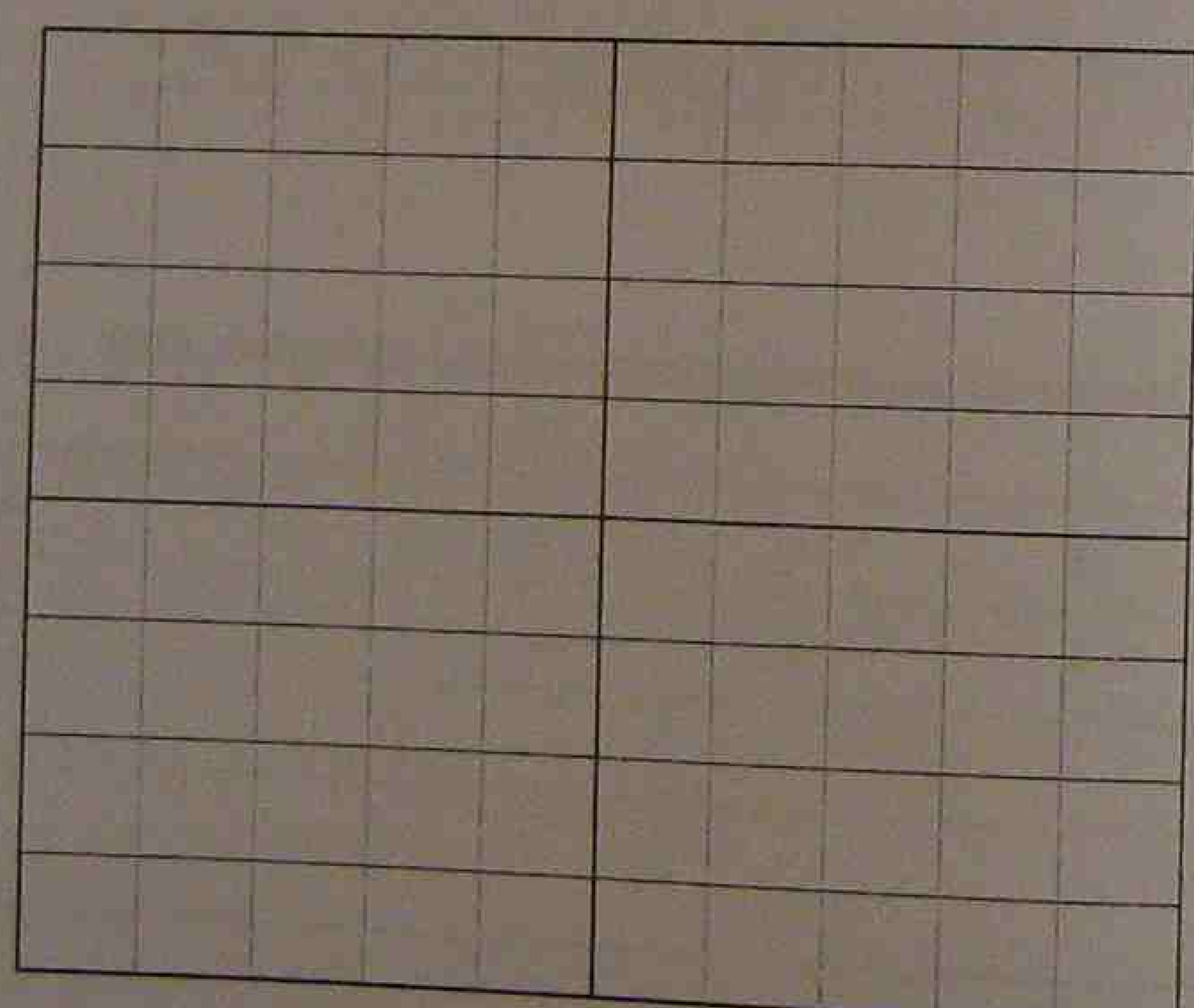
V_1 (V)	1	1	2	-2	-3
V_2 (V)	1	-2	1	2	-1
V_o (Theory)					
V_o (Measured)					

Signal Mixer

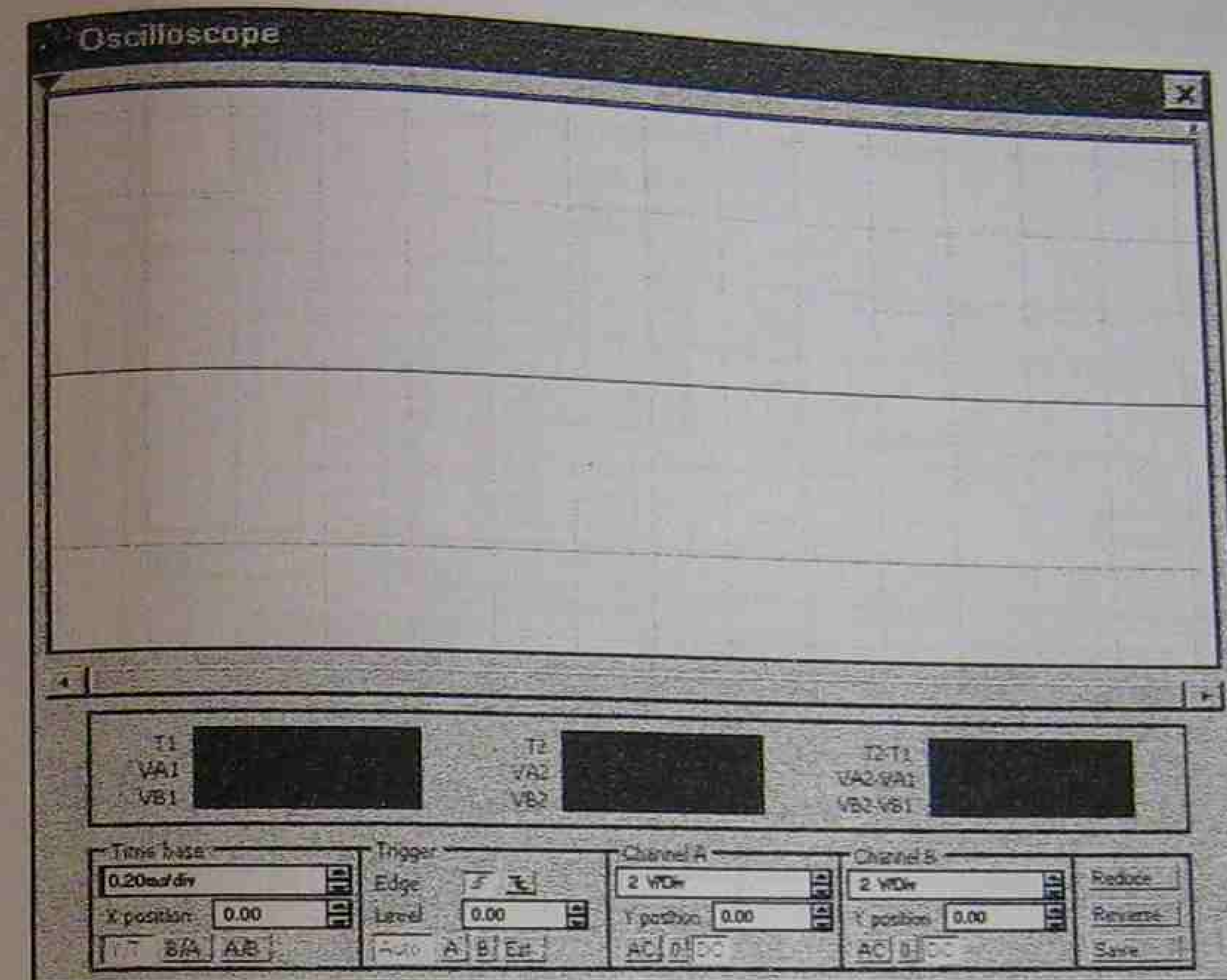
1. Construct the circuit shown below using Electronics Workbench.



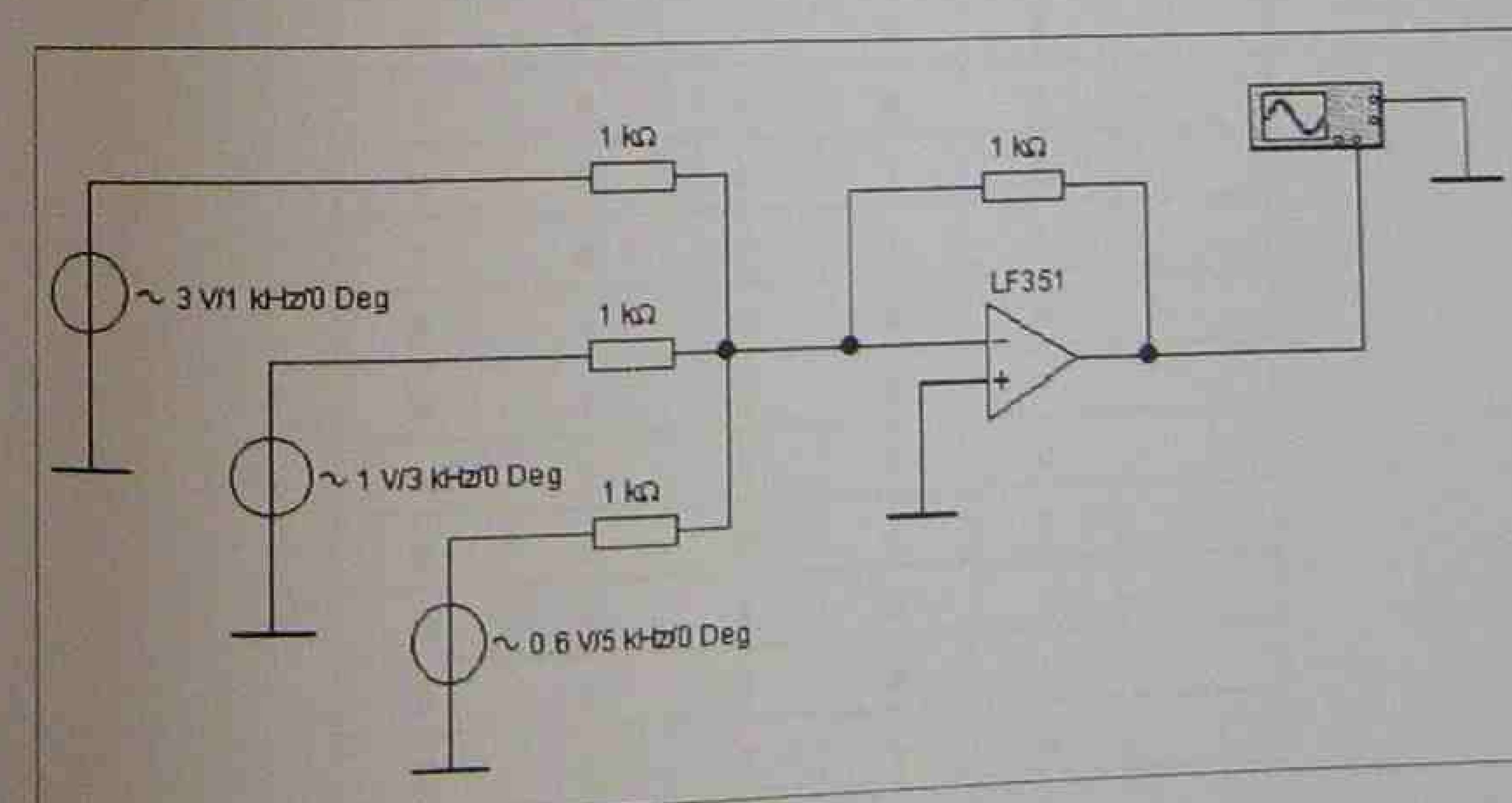
2. Sketch the expected input/output waveforms.



3. Sketch the input/output waveforms using Electronics Workbench.

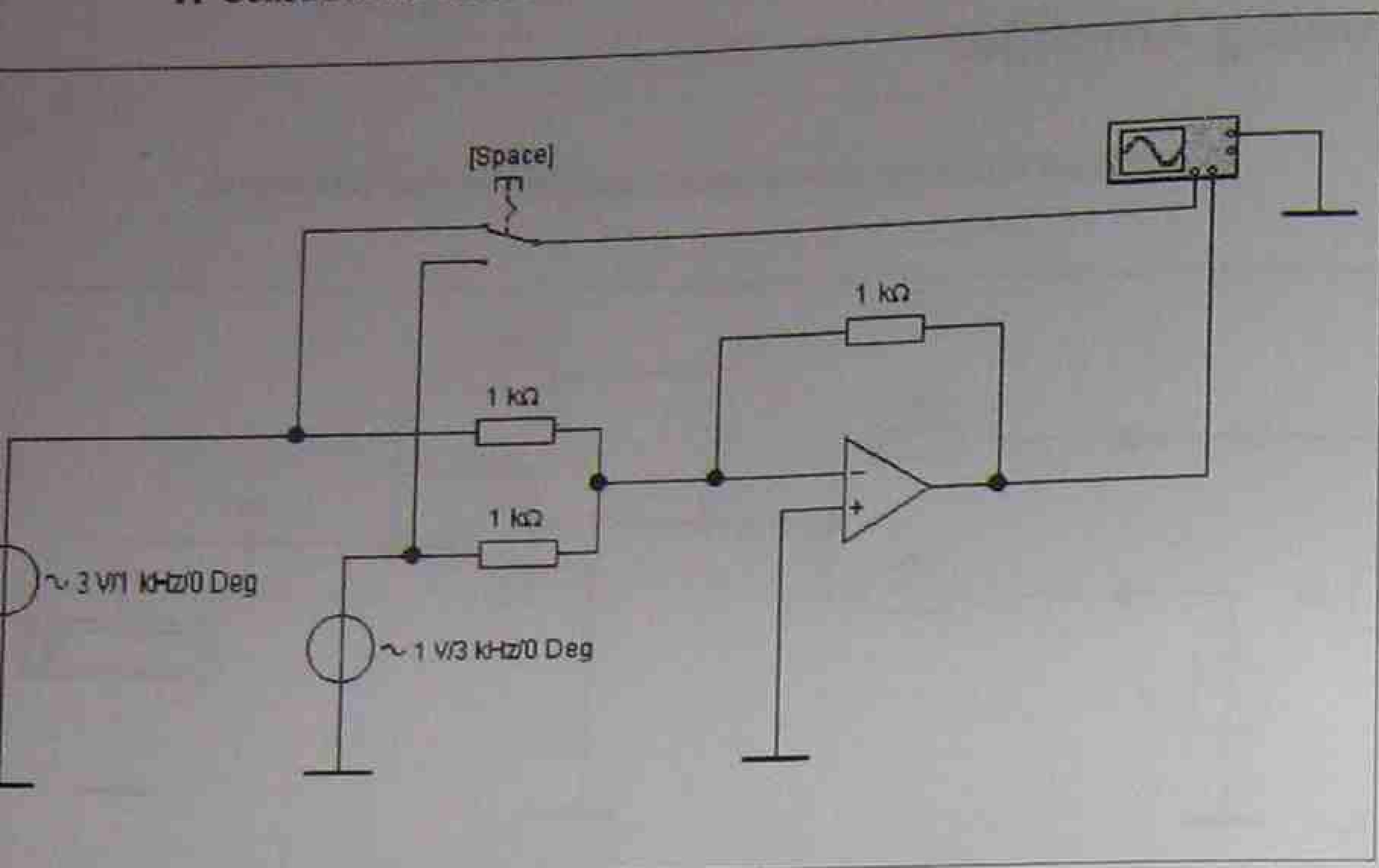


4. Add one more signal source as shown below and observe the output waveform.



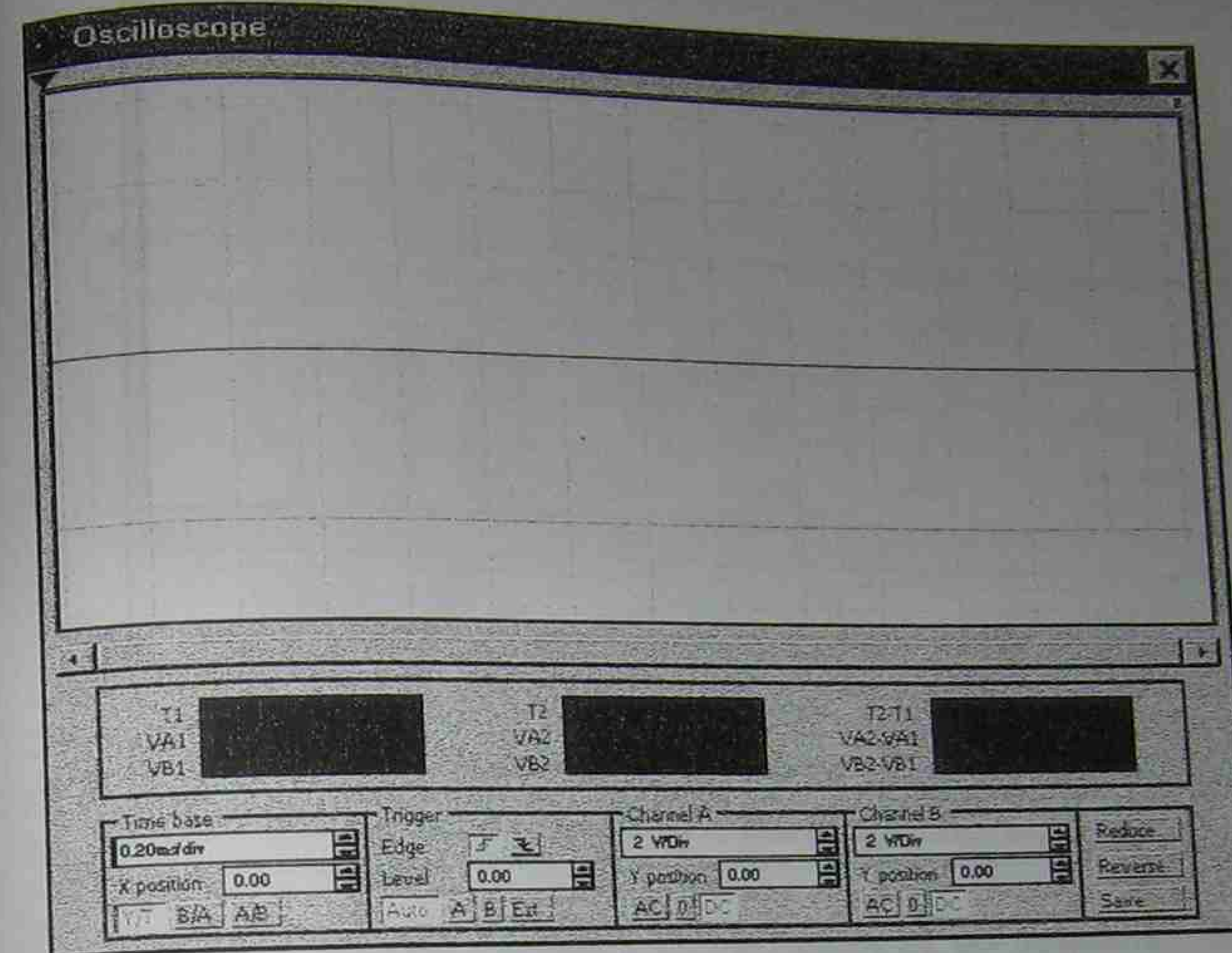
gnal Mixer

1. Construct the circuit shown below using Electronics Workbench.

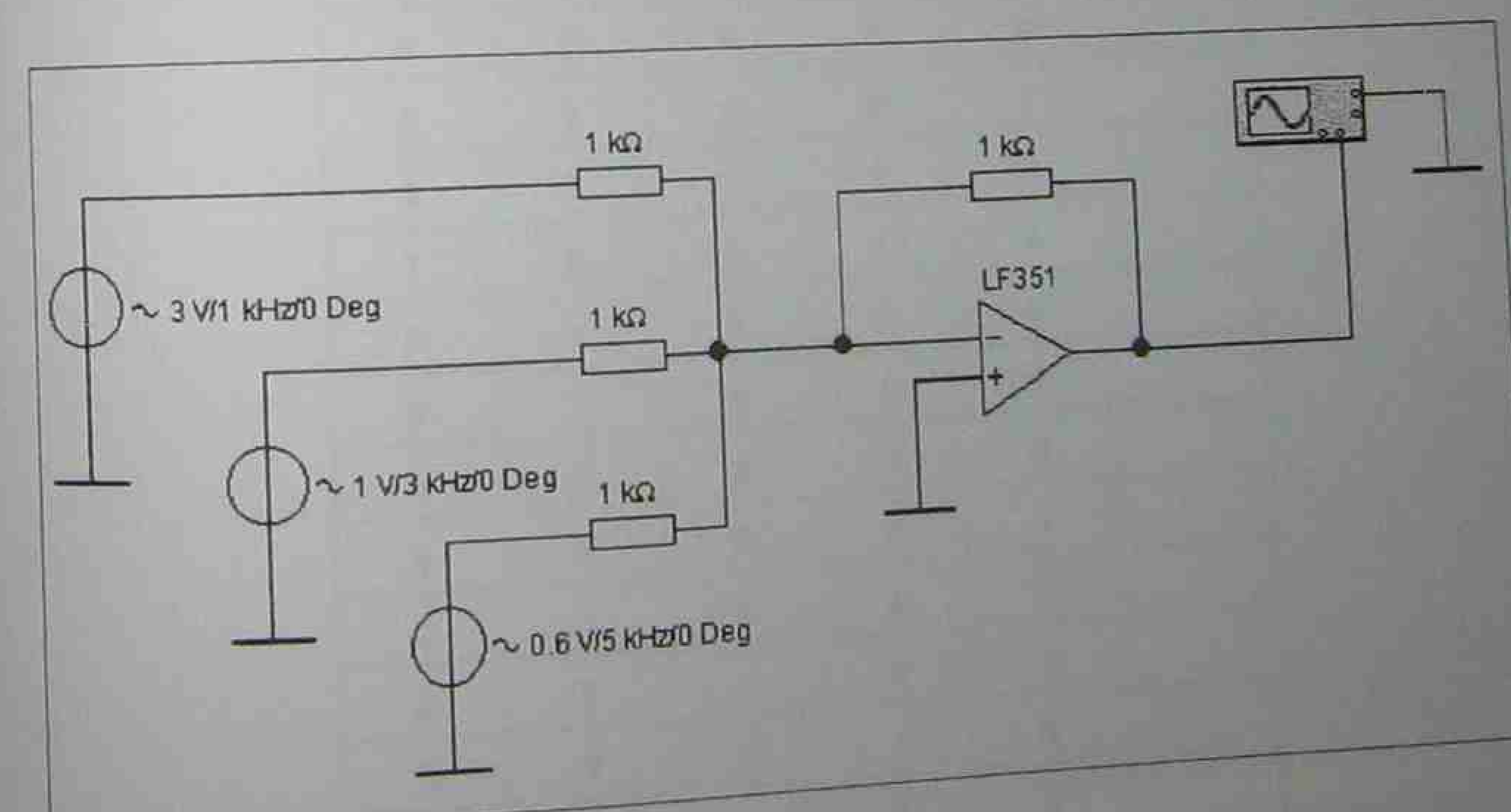


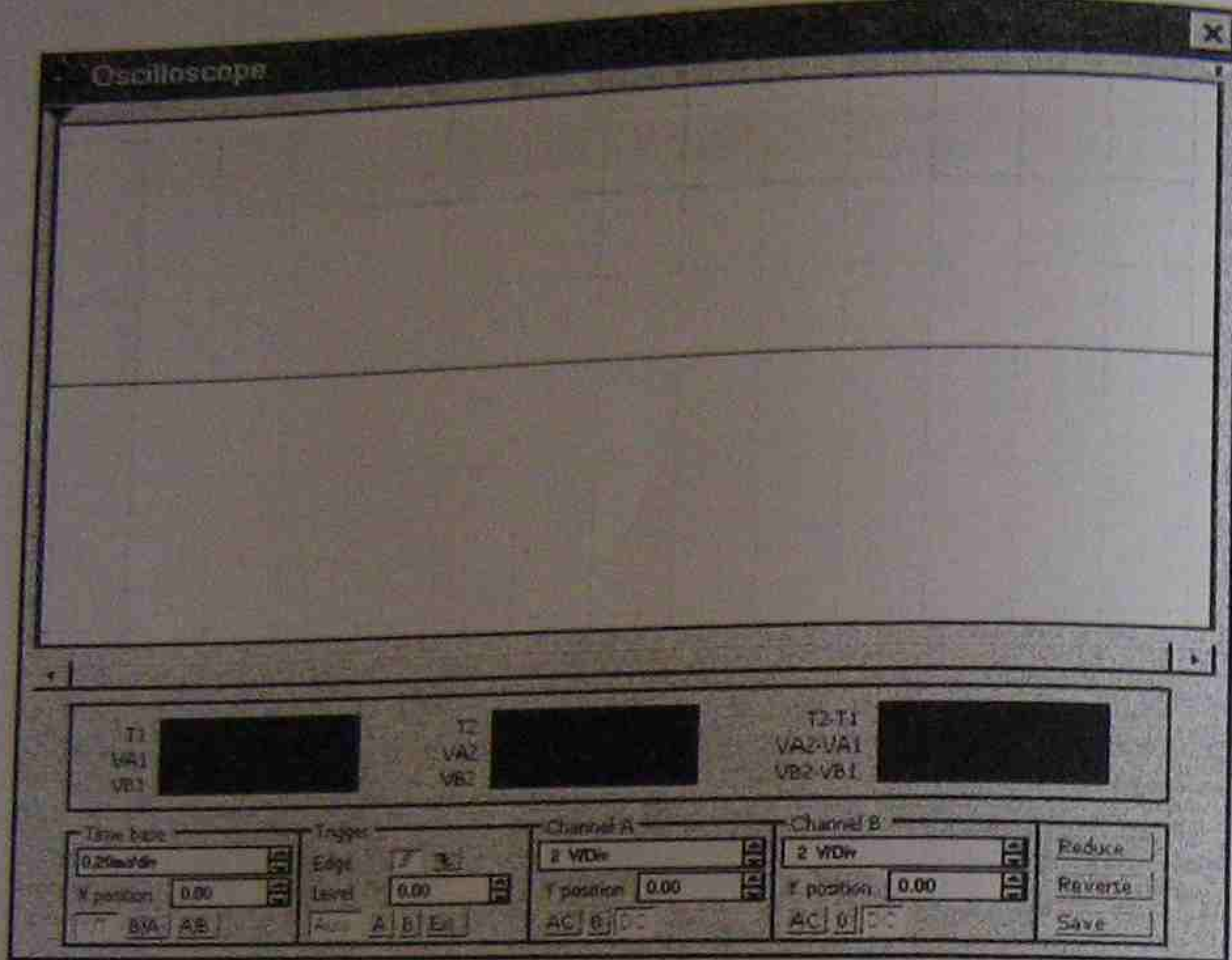
2. Sketch the expected input/output waveforms.

3. Sketch the input/output waveforms using Electronics Workbench.



4. Add one more signal source as shown below and observe the output waveform.

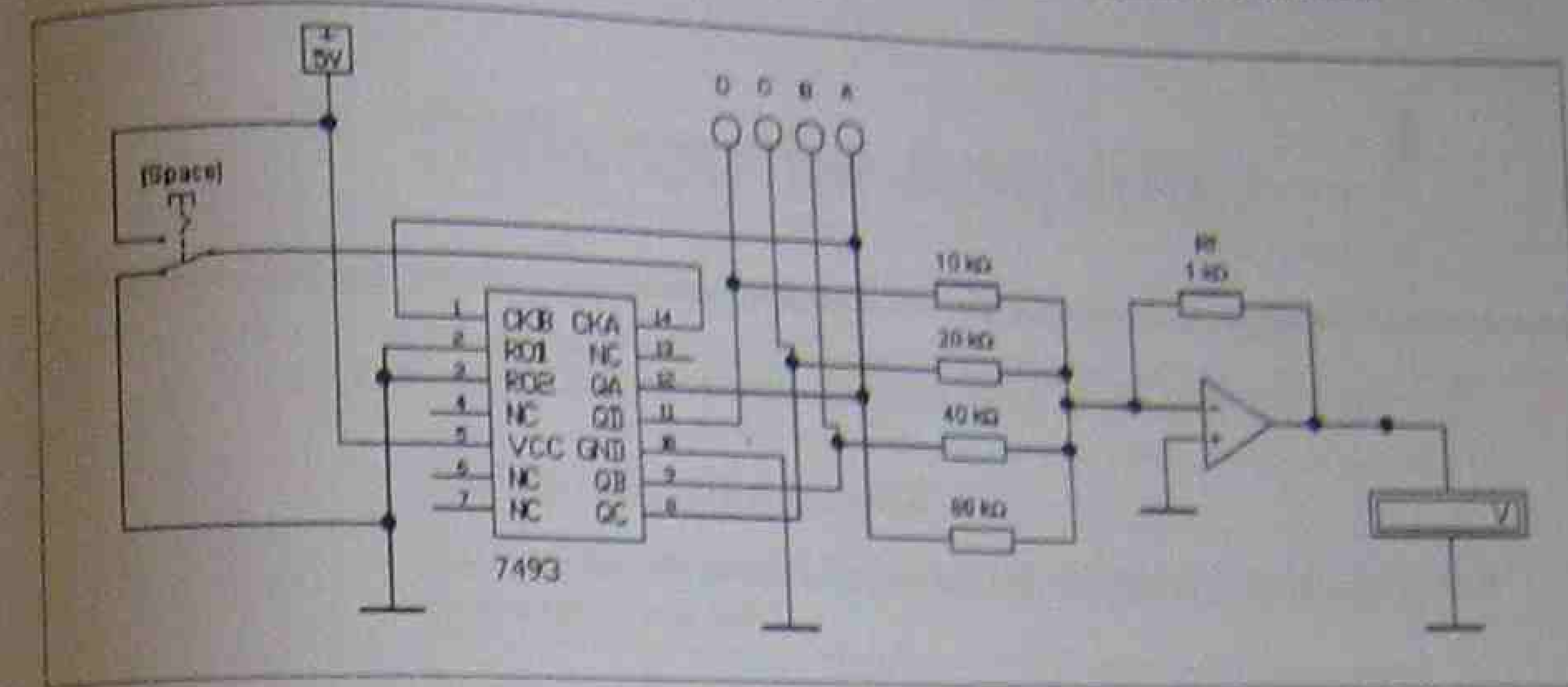




5. Explain why the voltage followers (buffers) are employed for the signal mixer in Fig. 5.3.

D/A Converter

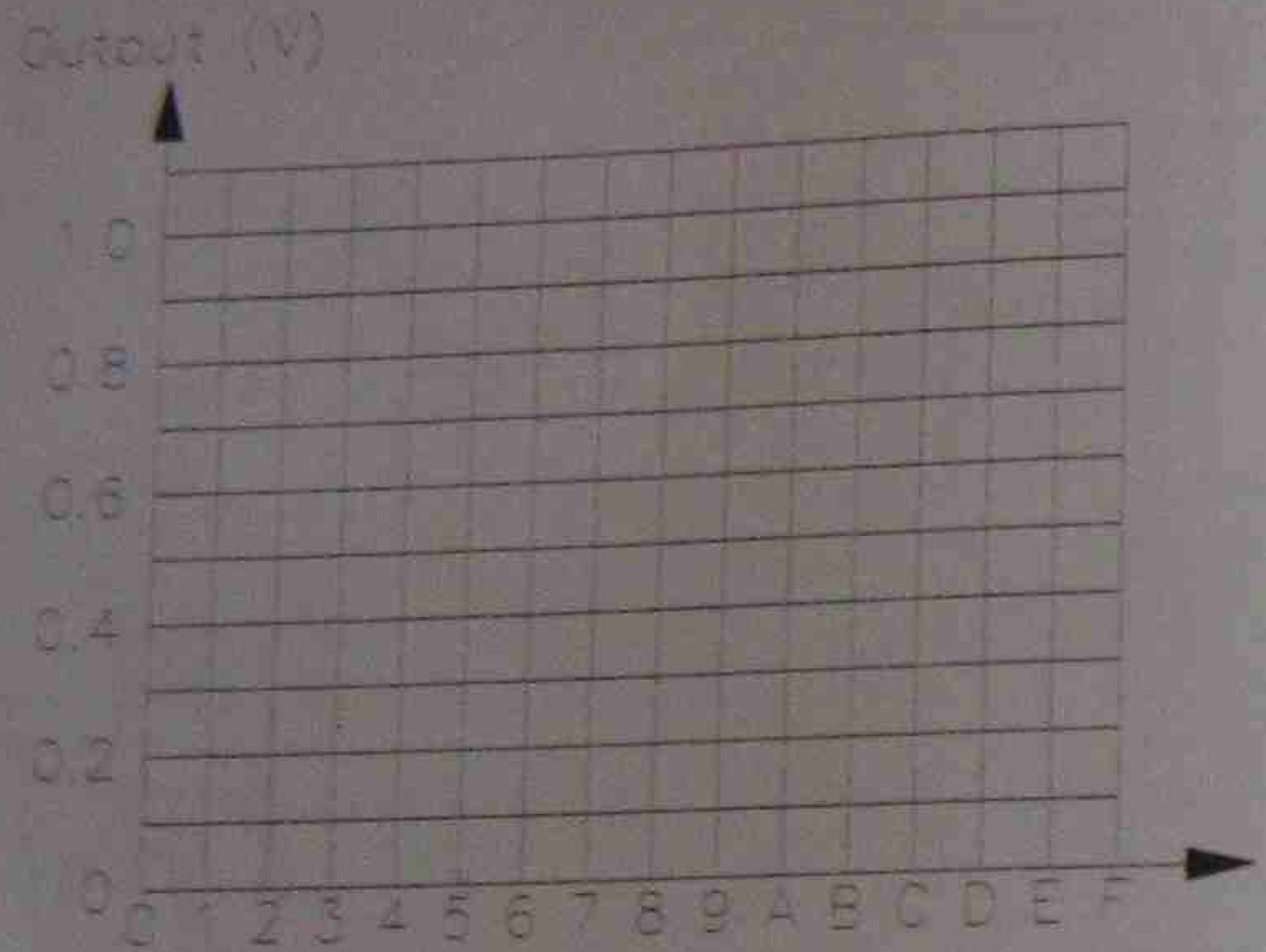
1. Construct the circuit shown below using Electronics Workbench.



2. Measure the output voltage for the signals shown below. (Press the space key twice to make a single pulse for the counter input. You can see the change of logic through LED indicators which are named A, B, C and D respectively.)

	D	C	B	A	Vo(mV)
0	0	0	0	0	
1	0	0	0	1	
2	0	0	1	0	
3	0	0	1	1	
4	0	1	0	0	
5	0	1	0	1	
6	0	1	1	0	
7	0	1	1	1	
8	1	0	0	0	
9	1	0	0	1	
A	1	0	1	0	
B	1	0	1	1	
C	1	1	0	0	
D	1	1	0	1	
E	1	1	1	0	
F	1	1	1	1	

3. Plot output-voltage / input-number characteristic on the graph paper provided below



6

Differential Amplifier

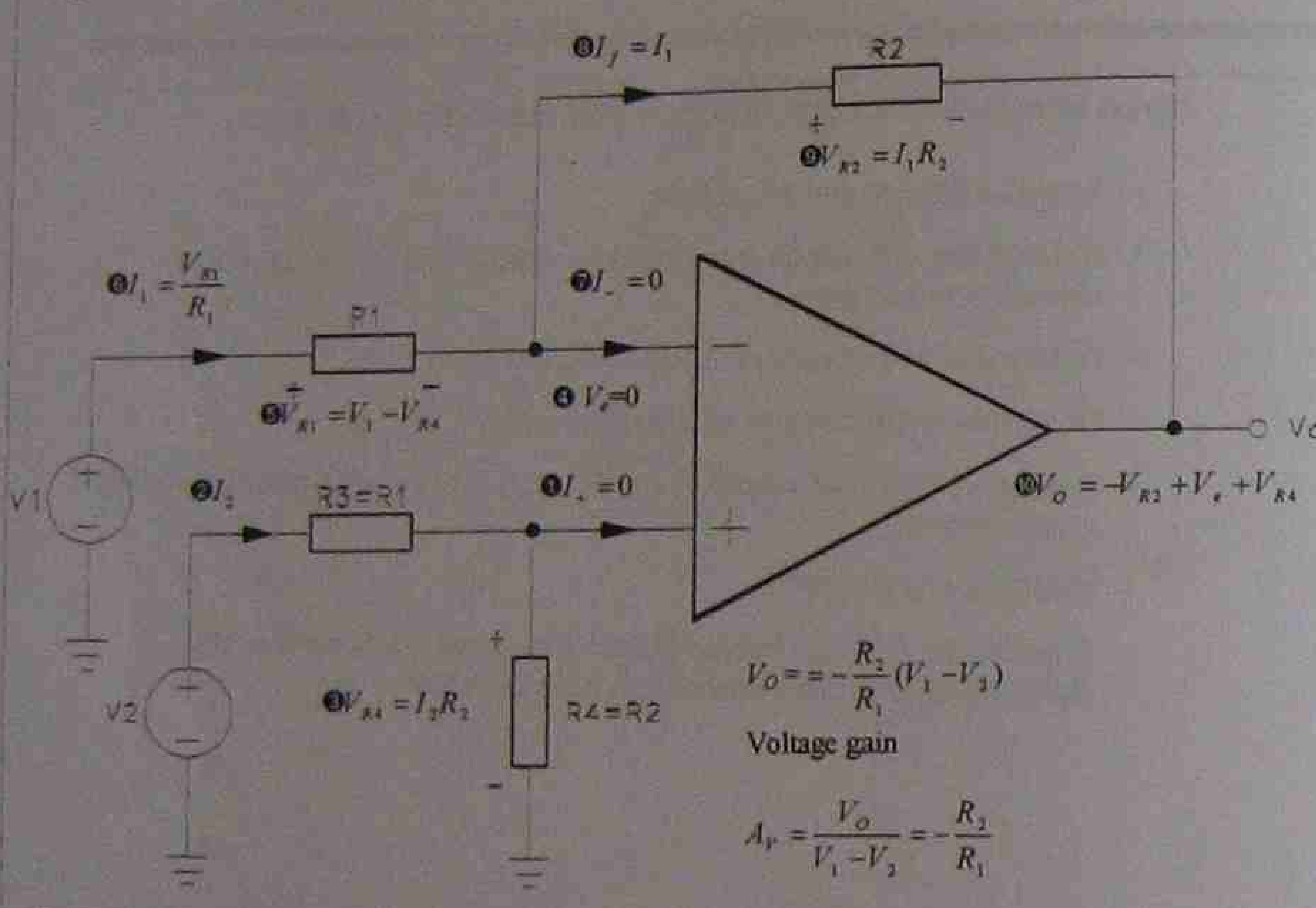
Upon completion of this chapter, you should be able to:

- Analyse differential amplifiers.
- Explain the difference between the differential mode gain and common mode gain.
- Determine the common mode rejection ratio (CMRR).
- Describe the necessity of the differential amplifier.
- Design differential amplifiers including CMRR adjustment for your applications.
- Measure the CMRR.
- Explain the CMRR change due to the input resistance of the differential amplifier and signal source resistance.

6.1 Differential Amplifier

Differential amplifier amplifies the voltage difference between two inputs. That is, this amplifier subtrac t voltages from each other. Operational amplifier itself is a differential amplifier with extremely high voltage gain. Therefore additional circuit is required to obtain a requested gain. Figure 6.1 shows a differential amplifier using op amp.

Figure 6.1 Differential Amplifier



❶ There is no current through non-inverting input.

❷ By the Ohm's law, the input current I_2 is:

$$I_2 = \frac{V_2}{R_1 + R_2}$$

❸ Therefore, the voltage drop across R_4 is:

$$V_{R4} = I_2 R_2 = \frac{V_2}{R_1 + R_2} R_2 = \frac{R_2}{R_2 + R_1} V_2$$

❹ There is no voltage difference between the non-inverting input and the inverting input.

❺ Therefore, the voltage drop across R_1 is.

$$V_{R1} = V_1 - \frac{R_2}{R_1 + R_2} V_2 = \frac{V_1(R_1 + R_2) - R_2 V_2}{R_1 + R_2}$$

❻ The input current I_1 is:

$$I_1 = \frac{V_{R1}}{R_1} = \frac{V_1(R_1 + R_2) - R_2 V_2}{R_1(R_1 + R_2)}$$

❼ Because of the current through the non-inverting input is 0,

❽ The current through R_2 is the same current as I_1.

❾ The voltage drop across R_2 is:

$$V_{R2} = I_1 R_2 = \frac{V_1(R_1 + R_2) - R_2 V_2}{R_1(R_1 + R_2)} R_2 = \frac{V_1 R_2 (R_1 + R_2) - R_2^2 V_2}{R_1(R_1 + R_2)}$$

❿ The output voltage is determined:

$$V_O = -V_{R2} + V_e + V_{R4} = -V_{R2} + 0 + V_{R4}$$

$$= -\frac{V_1 R_2 (R_1 + R_2) - R_2^2 V_2}{R_1(R_1 + R_2)} - \frac{R_2}{R_2 + R_1} V_2$$

$$= -\frac{V_1 R_2 (R_1 + R_2) - R_2^2 V_2 - R_1 R_2 V_2}{R_1(R_1 + R_2)}$$

$$= -\frac{V_1 R_2 (R_1 + R_2) - R_2 V_2 (R_1 + R_2)}{R_1(R_1 + R_2)} = -\frac{V_1 R_2 - R_2 V_2}{R_1}$$

$$= -\frac{R_2}{R_1}(V_1 - V_2)$$

Therefore the **voltage gain** of the amplifier is:

$$A_V = \frac{V_O}{V_1 - V_2} = -\frac{R_2}{R_1}$$

This voltage gain is valid for all following conditions.

$$\frac{R_2}{R_1} = \frac{R_4}{R_3}$$

The voltage gain we determined is normally referred as a **differential mode voltage gain** A_d.

$$A_d = -\frac{R_2}{R_1}$$

Input Resistance

There are two different input resistance are existing - the input resistance seen from input 1 (R_{i1}) and the input resistance seen from input 2 (R_{i2}).

$$R_{i1} = \frac{V_1}{I_1}$$

$$R_{i2} = \frac{V_2}{I_2} = R_3 + R_4 \quad (= R_1 + R_2)$$

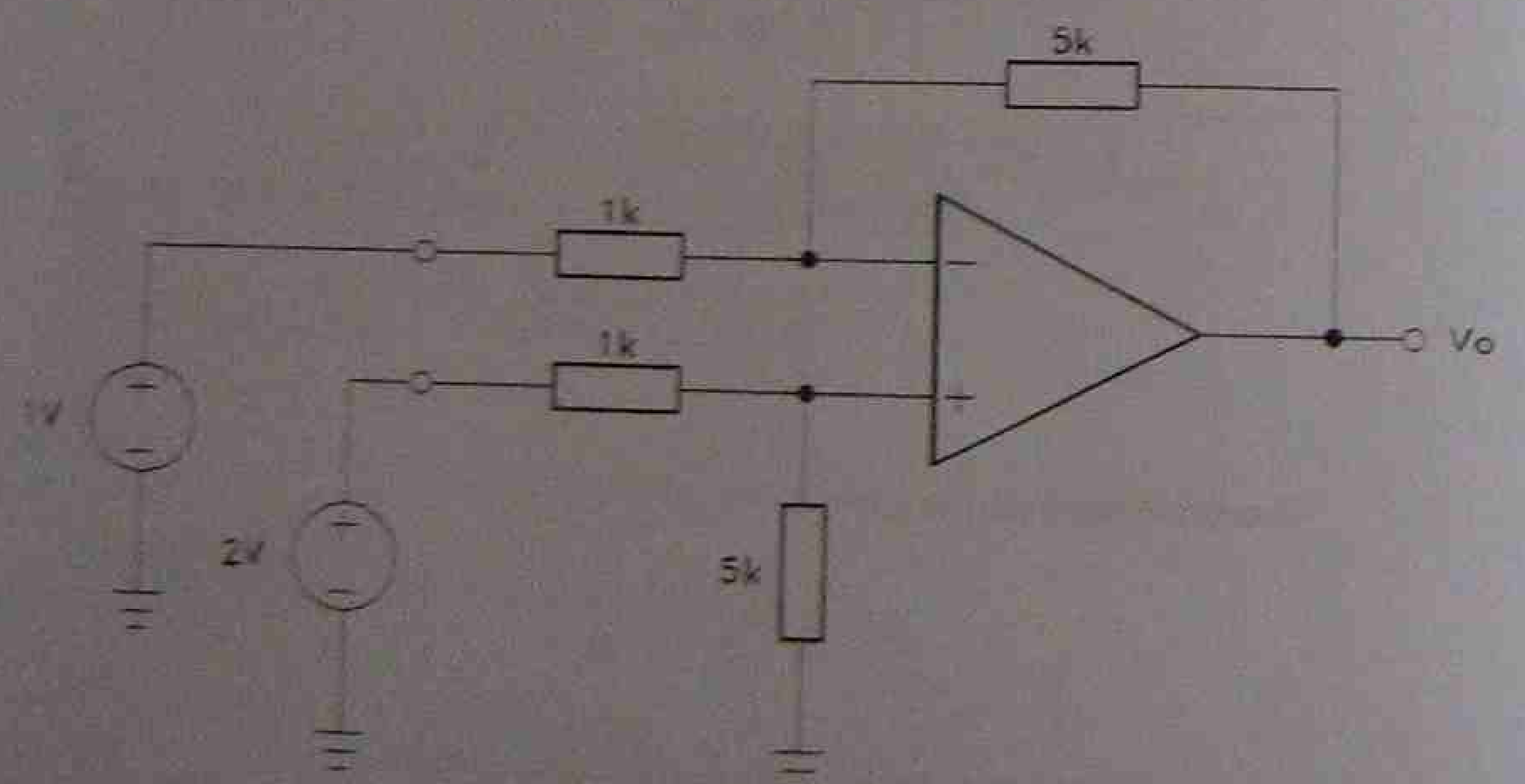
Output Resistance

The output resistance of the differential amplifier using op amp is extremely low like a inverting amplifier.

$$R_o = 0$$

Drill Question 1

For the circuit shown below, determine the output voltage and two input resistance.



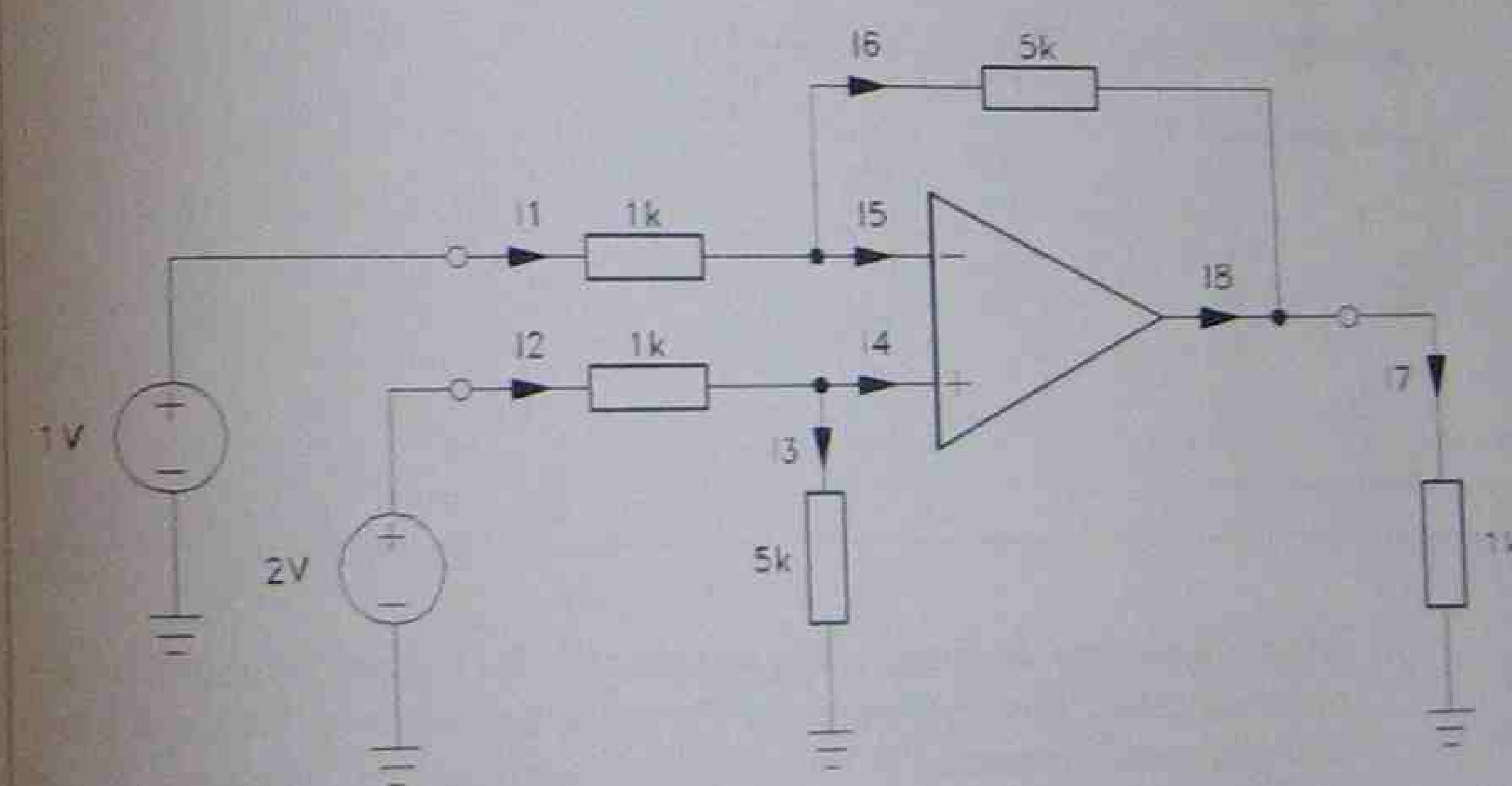
Drill Question 2

Design a differential amplifier for:

$A_V = -10$, $R_{i2} = 2.2\text{k}\Omega$

Drill Question 3

For the circuit diagram shown below, determine the currents $I_1 \sim I_8$.



Common Mode Gain

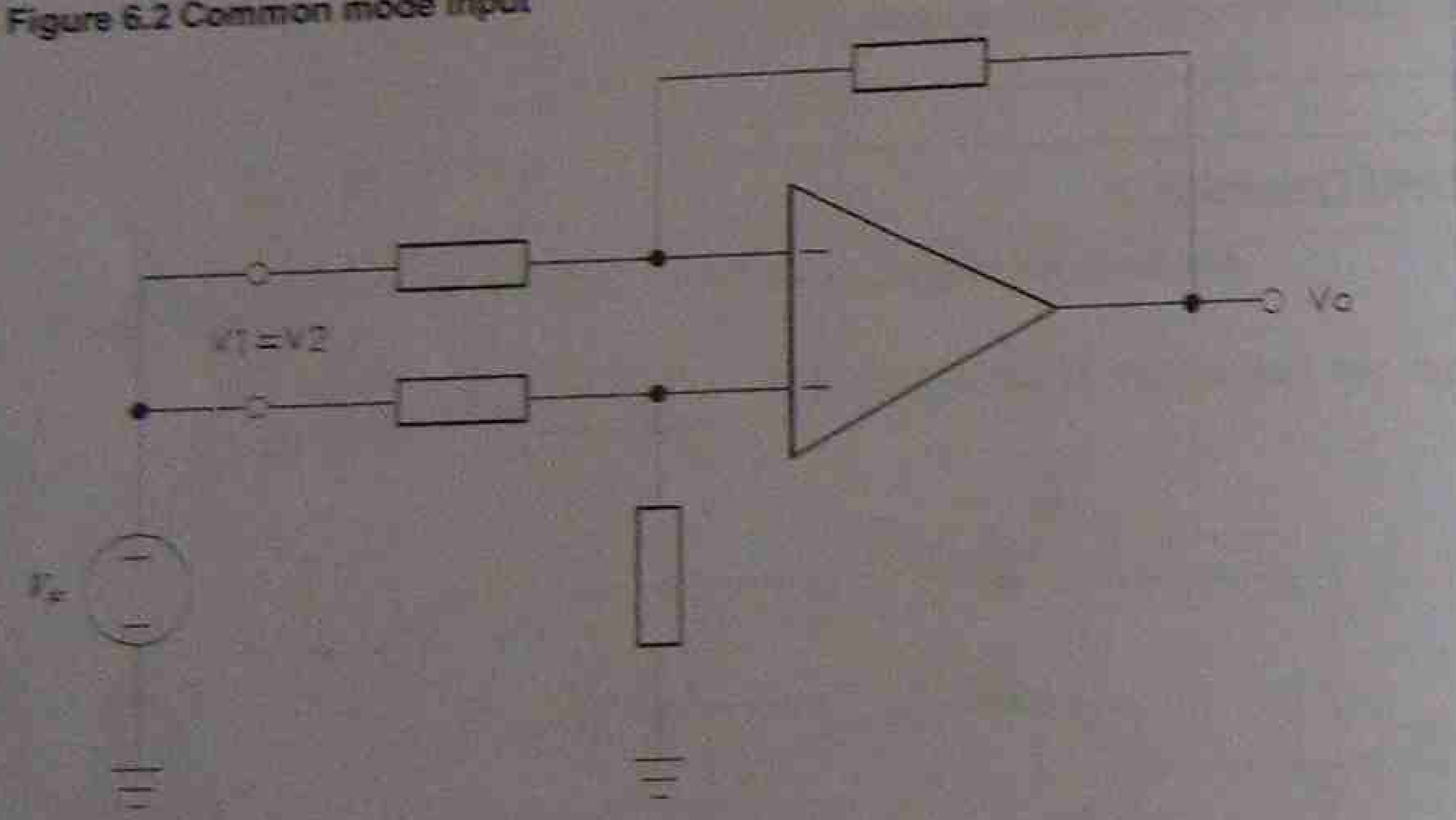
A common mode signal is one that drives both inputs of a diferential amplifier equally. Figure 4.18 shows a common mode input. Because two input voltages are same ($V_1 = V_2$), ideally the output voltage must be 0V. The common mode gain is defined as the ratio of output voltage to common mode input voltage.

$$A_c = \frac{V_O}{V_{ic}}$$

$$\text{or } A_c = 20\log\frac{V_O}{V_{ic}} \text{ (dB)}$$

The common mode gaon of an ideal differential amplifier is 0. However, because of the tolerance of resistance and non-ideal op amp actual common mode gain is not zero but it is a very small value.

Figure 6.2 Common mode input



Most interference, magnetic, static, and other kinds of undesirable pickup are common mode. What happens is this. The connecting wires on the input bases act like small antennas. If the differential amplifier is operating in an environment with a lot of electromagnetic interference, each base picks up an equal amount of unwanted interference voltage. In this case, $V_1 = V_2$. Therefore, no electromagnetic inteference voltage is appeared at the output.

Common Mode Rejection Ratio (CMMR)

The CMMR is defined as the ratio of differential voltage gain to common mode voltage gain.

$$\text{CMMR} = \frac{|A_d|}{|A_c|}$$

$$\text{or CMMR} = 20\log\frac{|A_d|}{|A_c|} \text{ (dB)}$$

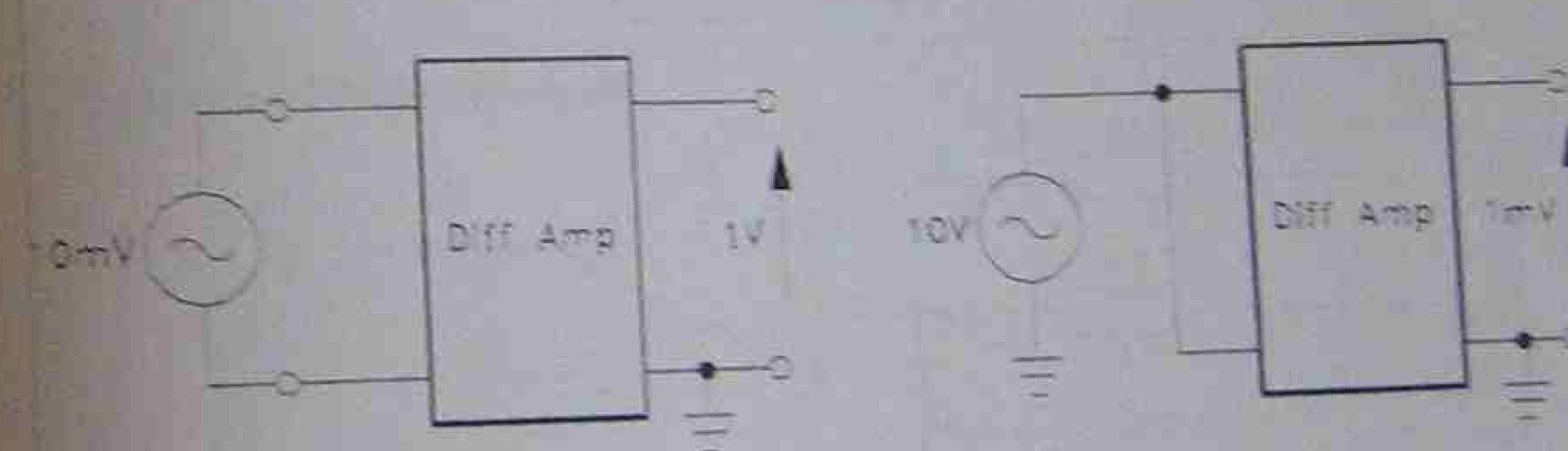
As we discussed in the common mode gain, the higher the CMMR is, the better the differential amplifier.

For the real differential amplifier, the CMMR due to the tolerance of resistance is given:

$$\text{CMMR} = \frac{R_2}{R_1} \times \frac{1 + \frac{R_4}{R_3}}{\left|\frac{R_4}{R_3} - \frac{R_2}{R_1}\right|}$$

Drill Question 4

For a differential amplifier measurement shown below, determine:



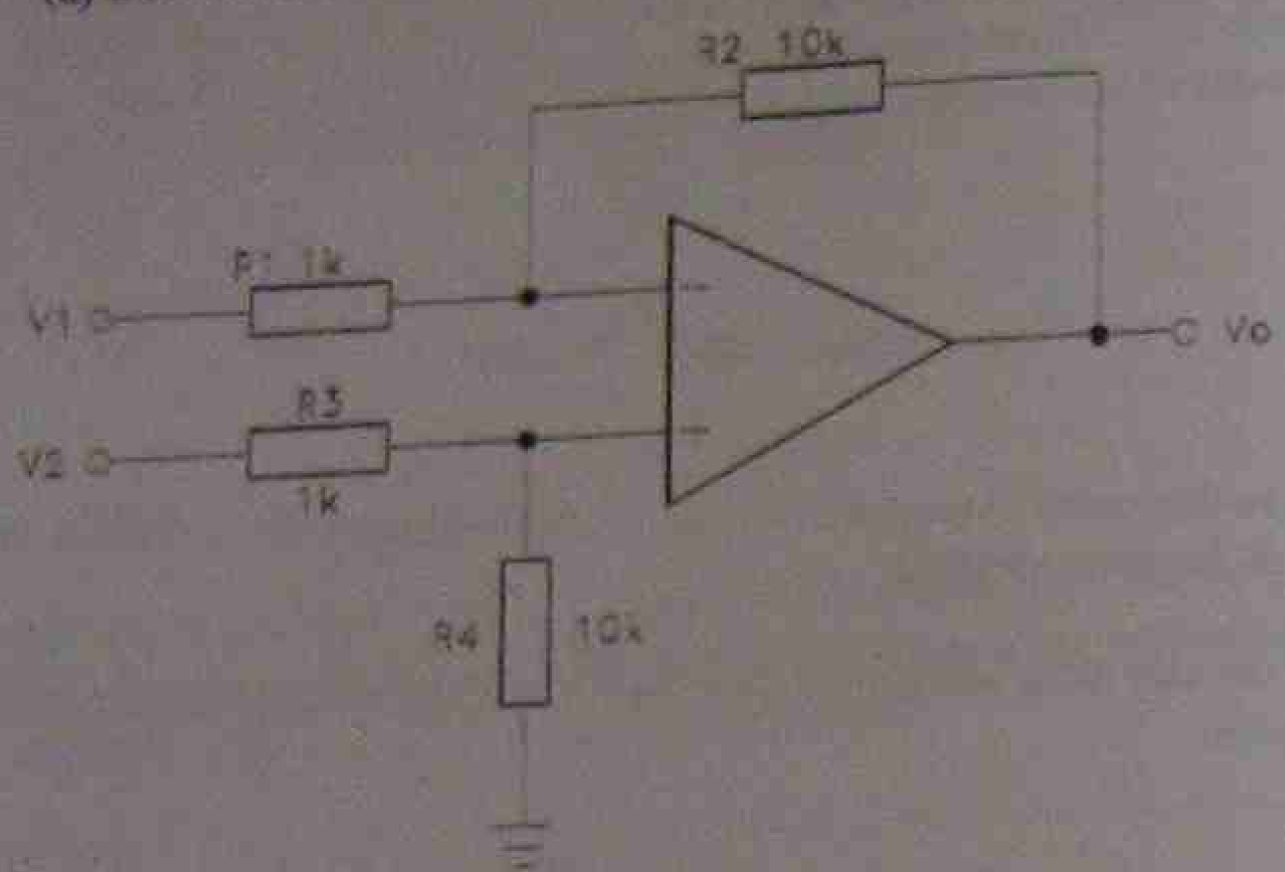
(a) Differential mode gain

(b) Commom mode gain

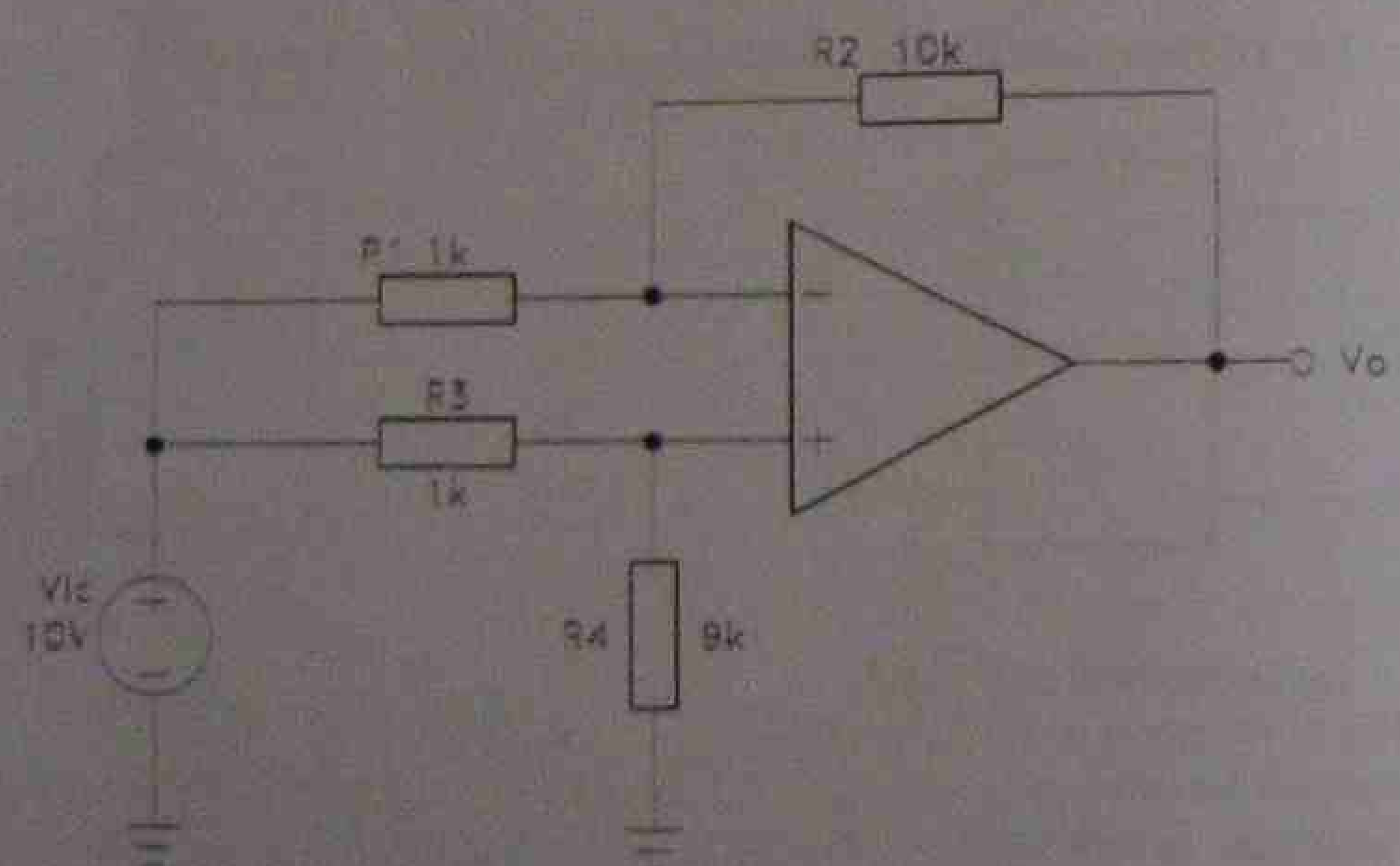
(c) CMMR

Drill Question 5

(a) Determine the differential mode voltage gain for the circuit shown below.



(b) The real value of R_4 is measured 9k instead of 10k. Determine the common mode gain using following procedure.



(i) Determine the voltage drop across R4.

Drill Question 22. continue.

(ii) Determine the voltage drop across R1. and calculate the current through R1.

(iii) Determine the current through R2.

(iv) Calculate the voltage drop across r2.

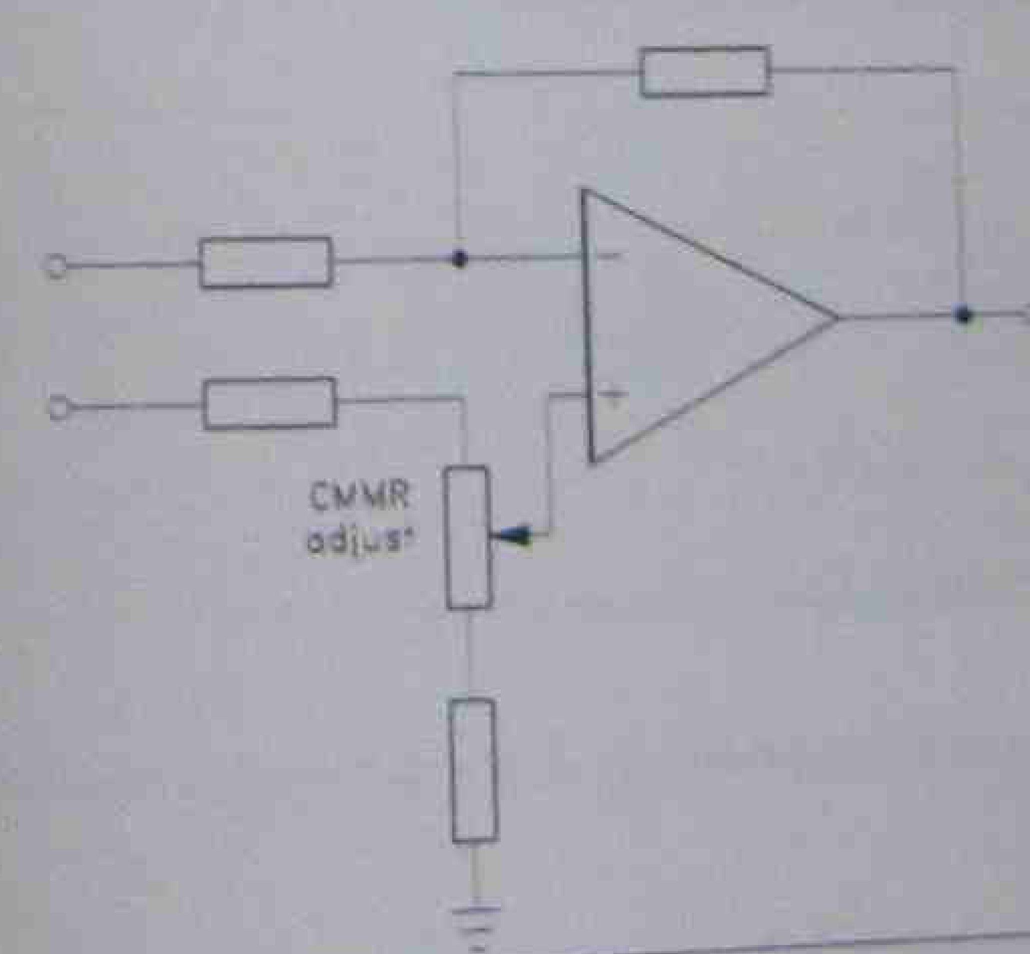
(v) Determine the output voltage.

(vi) Determine the common mode gain.

(vii) Determine the CMMR.

In the Figure 4.21, a potentiometer is employed to adjust the minimum common mode gain (maximum CMMR).

Figure 6.3 CMMR adjustment.

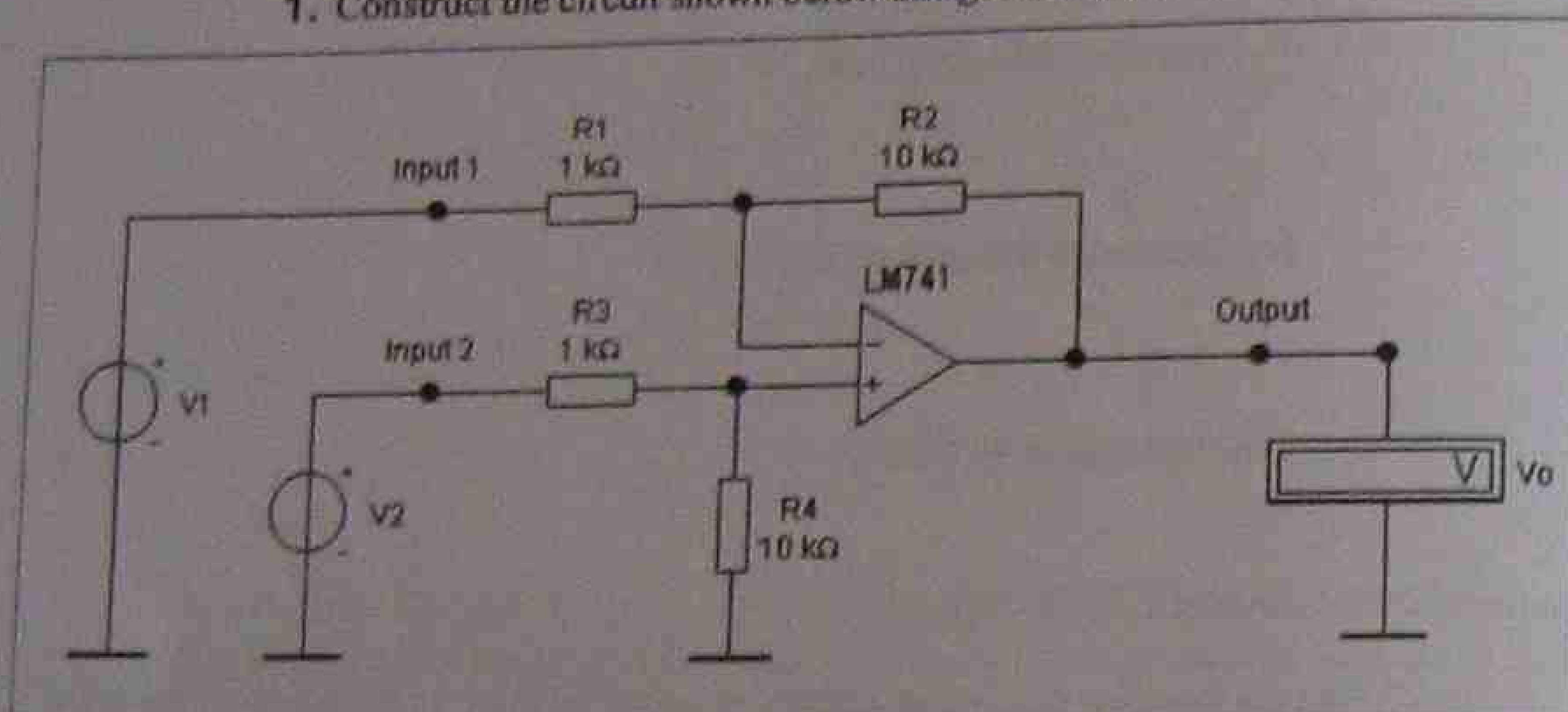


Skill Practice 7

Differential Amplifier

Differential Mode Gain

1. Construct the circuit shown below using Electronics Workbench.

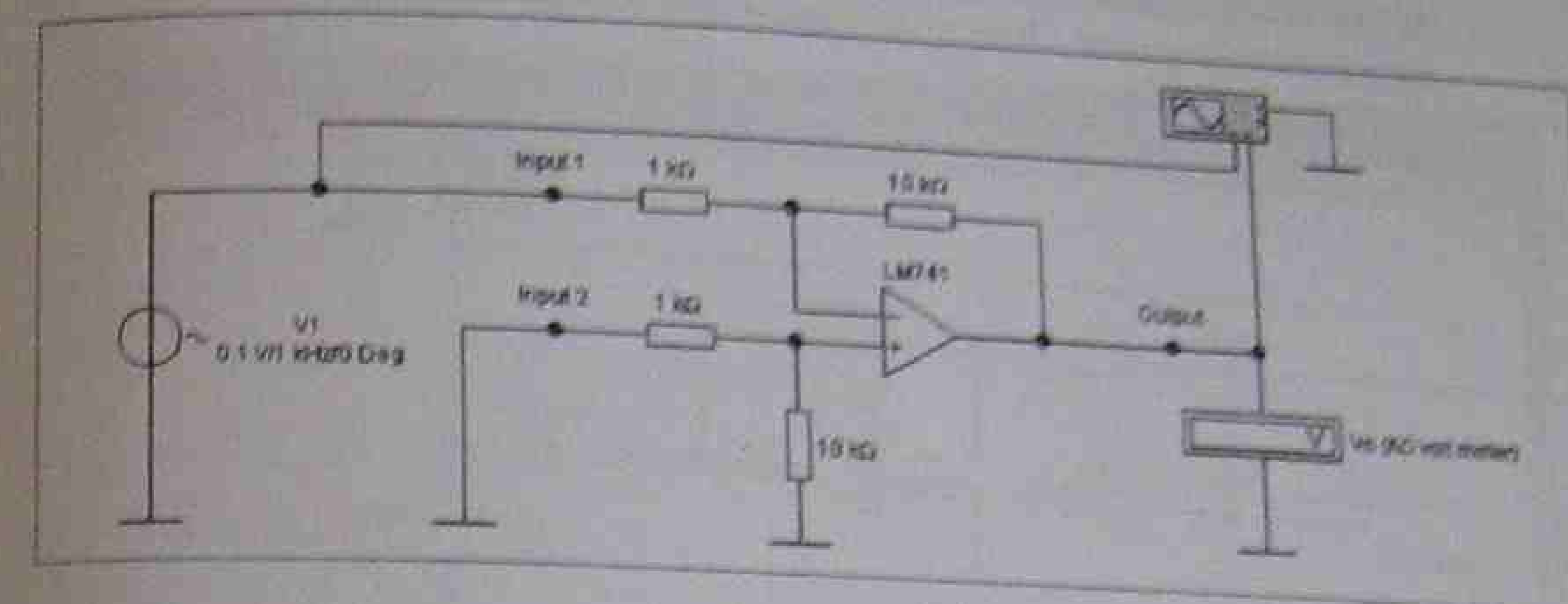


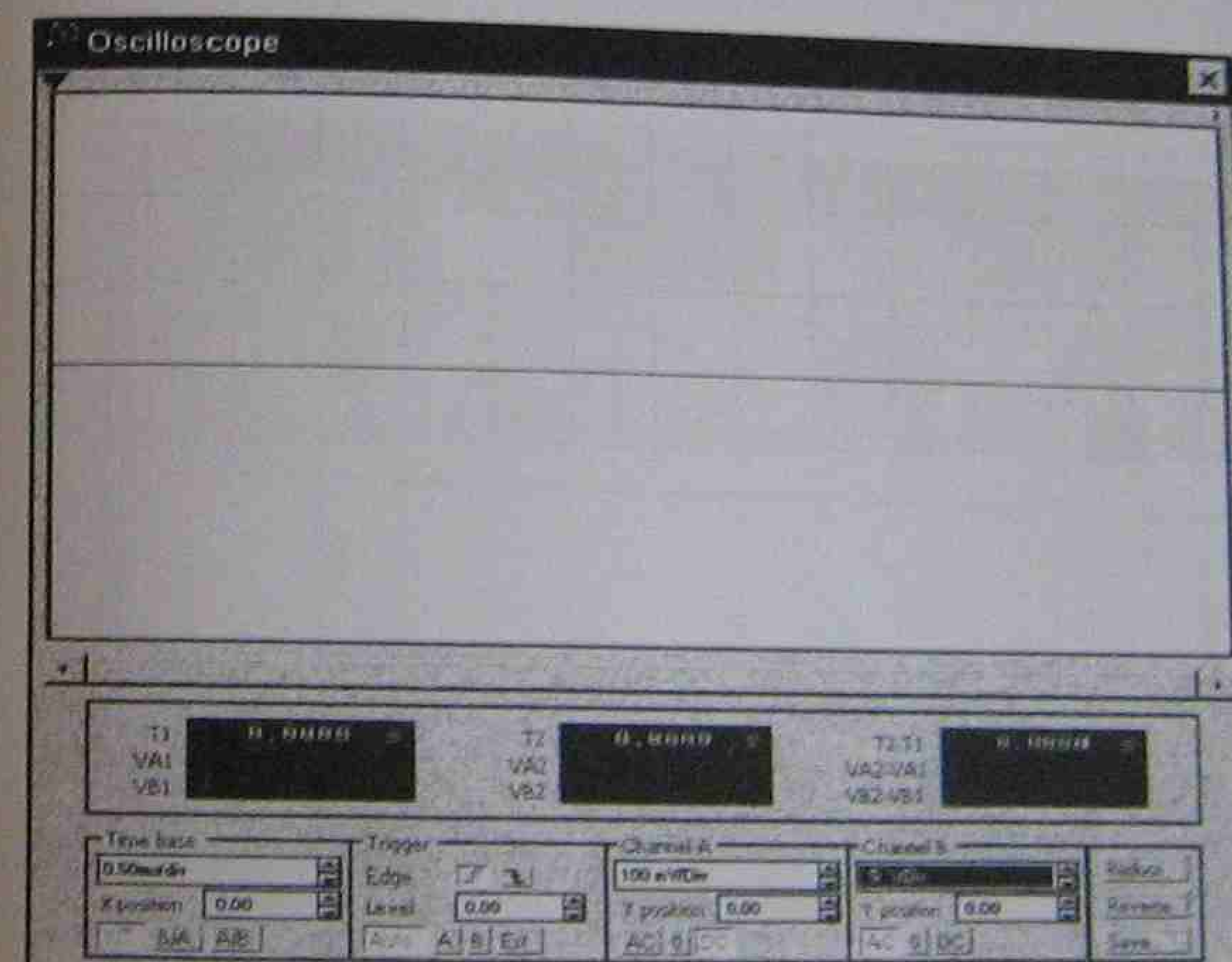
2. Measure the output voltages for the different input voltages shown below. Also, calculate the differential mode voltage gain A_d.

V_1 (V)	1	2	1	-1
V_2 (V)	2	1	1	-0.5
V_O (V)				
A_D				

3. Compare the measurement results with theoretical calculations.

4. Replace input V_1 with an 1KHz 0.1V sine wave signal source and measure the output waveform and output voltage. Also, determine the differential mode gain for this application.



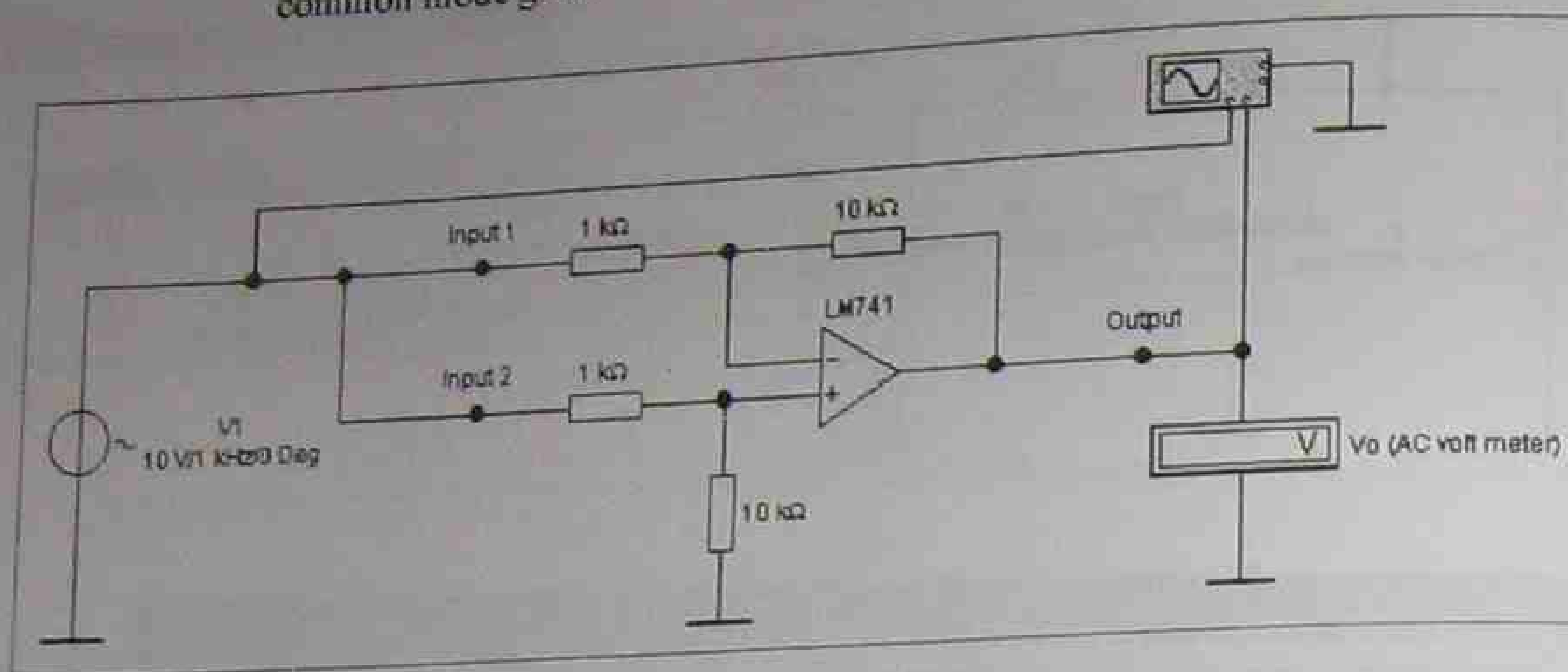


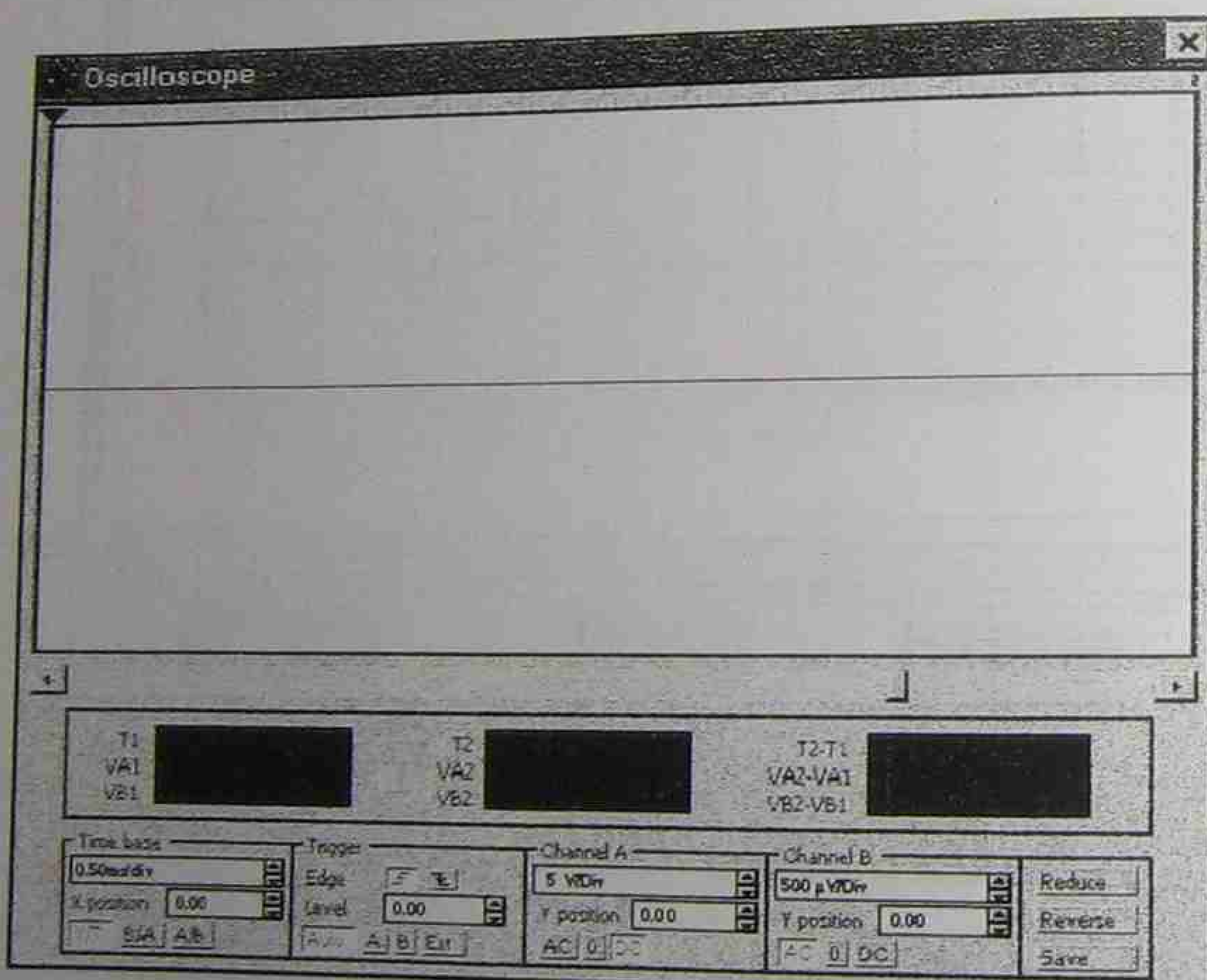
$V_O =$

$A_d =$

Common Mode Gain

1. The circuit below is working under common mode since input1 and input 2 are connected together. Observe the input/output waveforms and determine the common mode gain.

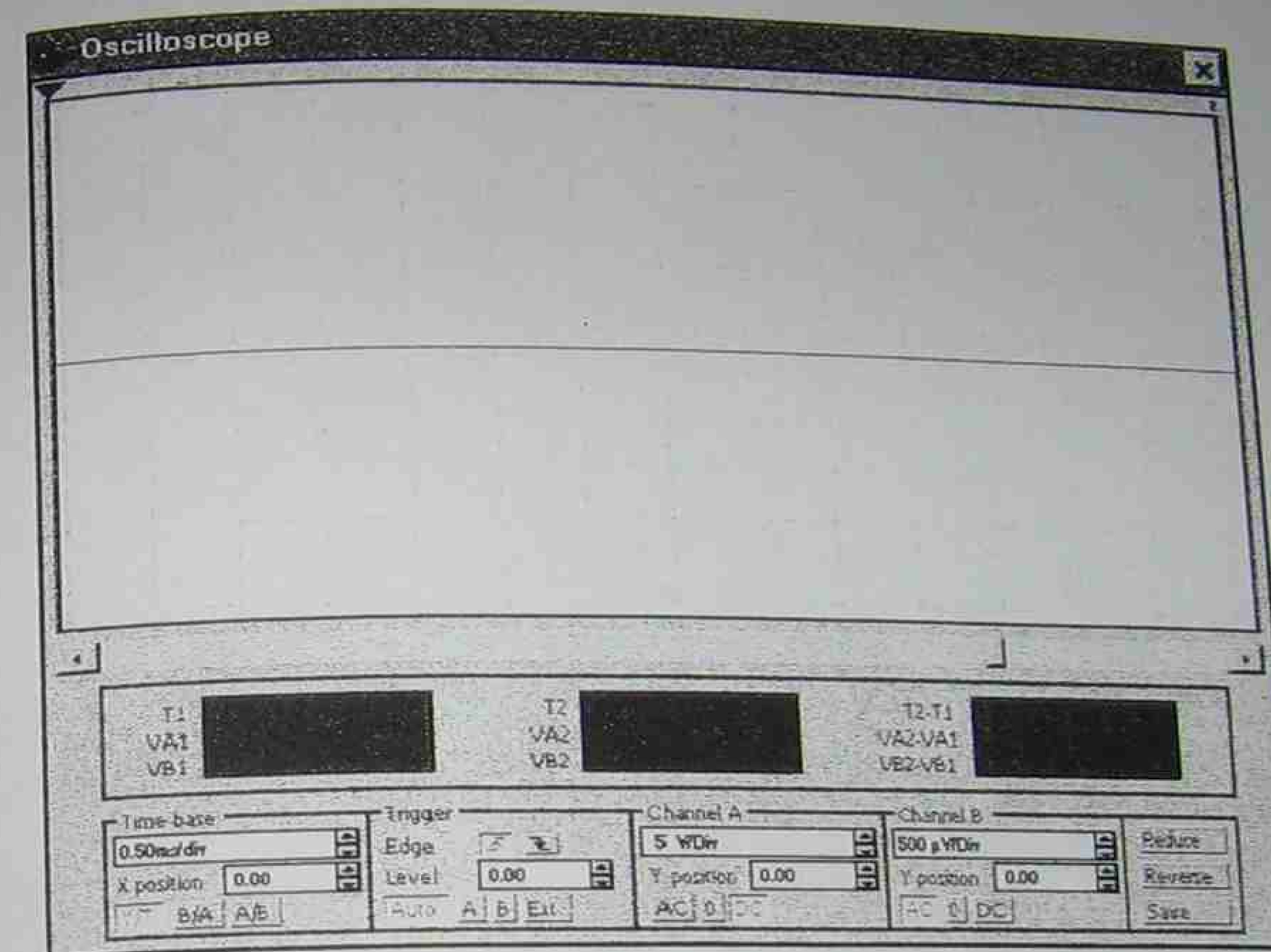




$V_O =$

Common mode gain $A_C =$

2. Replace op amp with LF351 and repeat the previous measurement.



$V_O =$

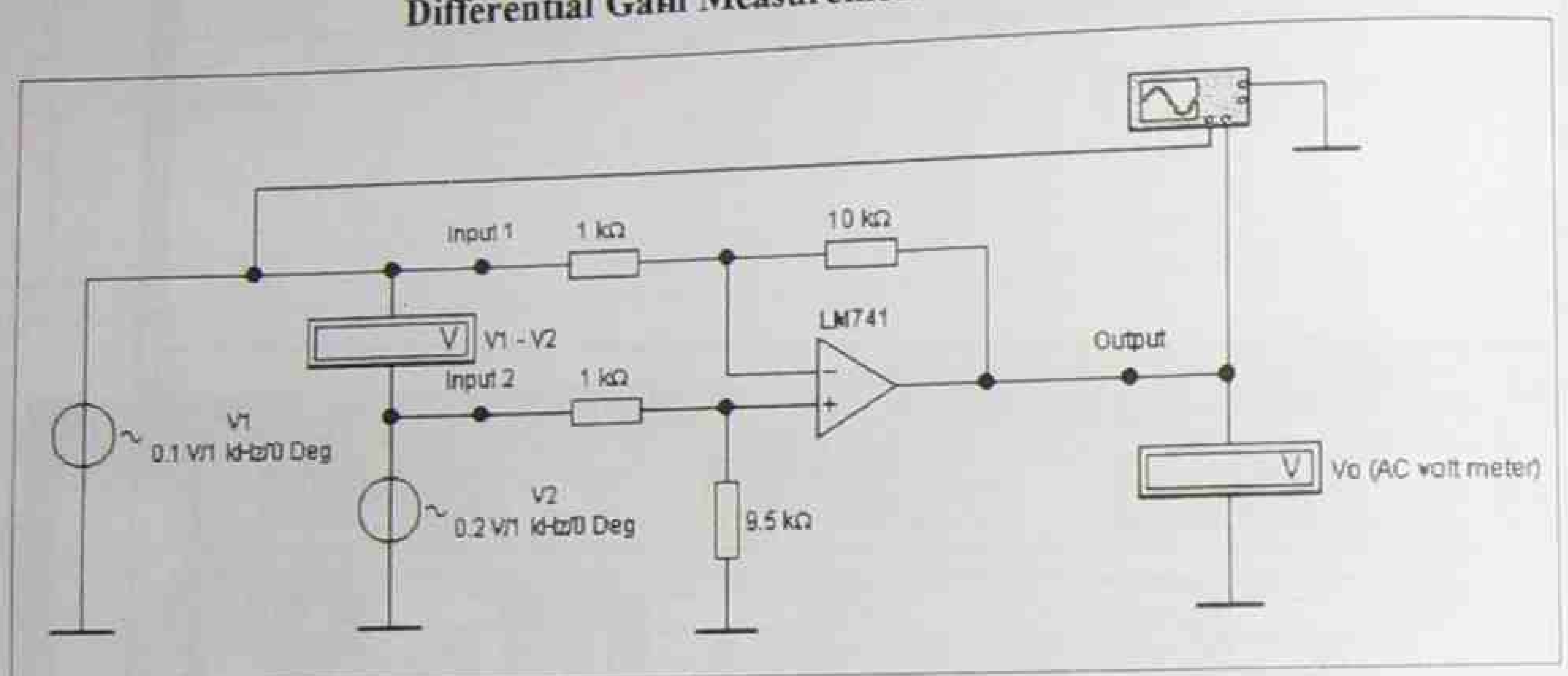
$A_C =$

Common Mode Rejection Ratio (CMRR)

1. Determine the CMRR (dB) for both LM741 and LF351 applications.

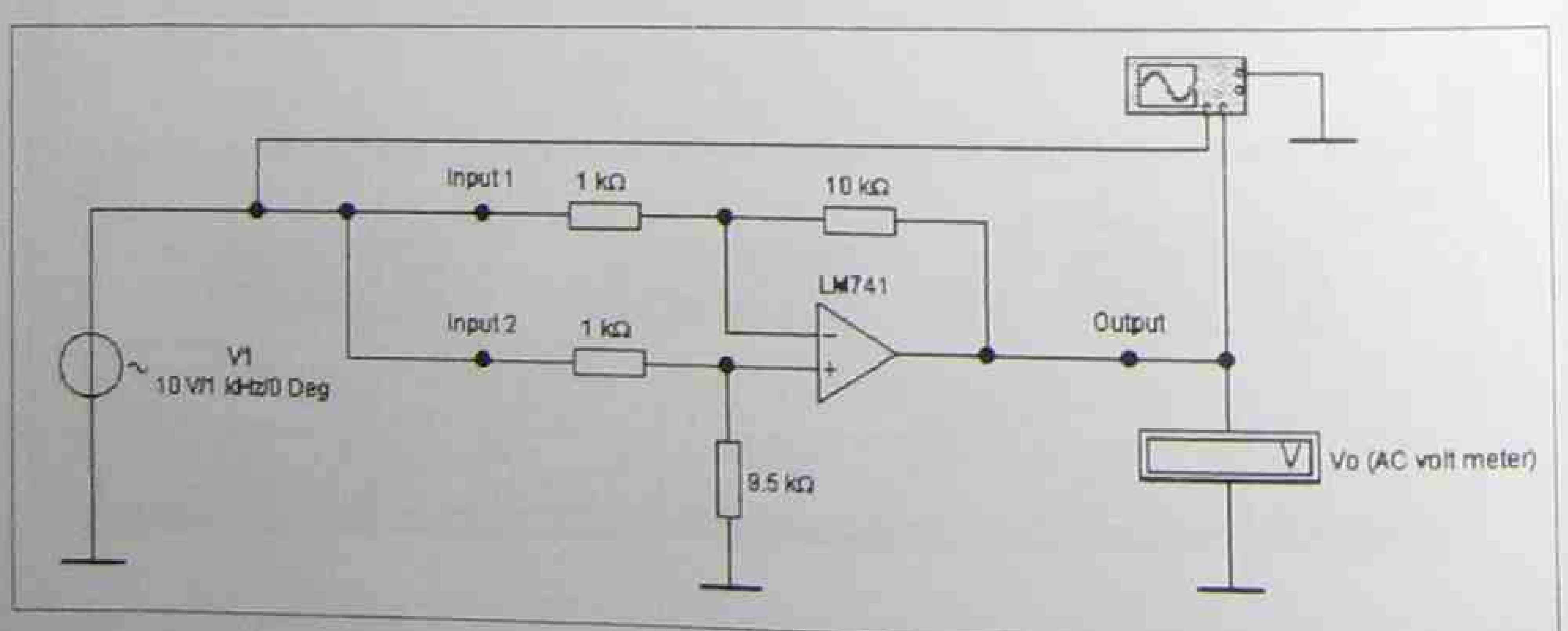
2. Replace R4 with 9.5kΩ and measure the CMRR (dB).

Differential Gain Measurement



$A_D =$

Common Mode Gain Measurement



$A_C =$

CMRR Calculation

CMMR (dB) =

ADVANCED CERTIFICATE
IN APPLIED INDUSTRIAL ELECTRONICS 6016

ELECTRONIC DEVICES 6016A
THEORY/TUTORIAL MANUAL

by

Peter Phillips

ACKNOWLEDGEMENTS

This manual has been developed with the much appreciated help of George Kriflik, Vic Ciscato, Robbie Thornton, Peter Bujack and many others too numerous to mention.

TO THE STUDENT

These notes should be read in conjunction with a text book. Reference in these notes is made to the text Electronic Devices (2nd Ed) by T.L. Floyd (Merrill) although other books as advised by the teacher can be used. It is important to recognise that this manual is a summary only and should not be regarded as the sole source of written information for this subject. This manual also contains tutorials, which are designed to ensure that the objectives for each lesson have been attained. You should attempt all questions as soon as possible after the relevant theory lesson.

TO THE TEACHER

This manual covers the syllabus content of the subject Electronic Devices, 6016A. The emphasis may vary between teaching centres and locally relevant material can be included. Each theory lesson lasts approximately one hour, although some lessons may require the theory to extend into the practical session, in the form of an integrated theory/practical presentation. These notes are relatively brief and students should be advised to purchase the recommended text book as well.

CONTENTS

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 1

REVISION - DC MEASUREMENT

OBJECTIVES: At the end of this lesson you will be able to:

- Apply Ohm's law in solving basic calculations.
- Use scientific notation and metric prefixes in calculations and measurements.
- Calculate the loading effect of a voltmeter and its effect in a circuit.

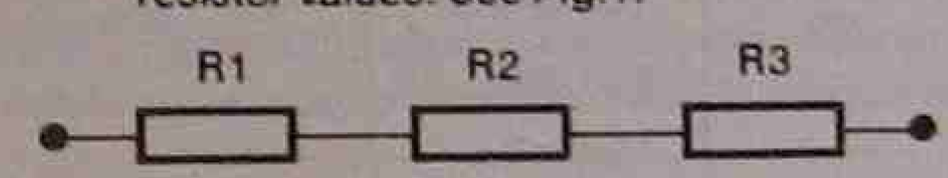
1. INTRODUCTION

Ohm's law is an essential tool in calculating voltages, currents and other values in an electronic circuit. Being able to apply Ohm's law is often more difficult than simply knowing the three basic equations that relate voltage, current and resistance. This lesson revises Ohm's law and uses these laws to show the loading effect a voltmeter has on the voltage it is measuring. All measuring instruments affect the reading they are taking, and a knowledge of the effect is important in deciding on the best type of instrument to use in a particular application.

2. OHM'S LAW IN RESISTIVE CIRCUITS

The three basic Ohm's law equations as applied to a resistive circuit are shown below. Note that the potential divider equation is often used in electronics.

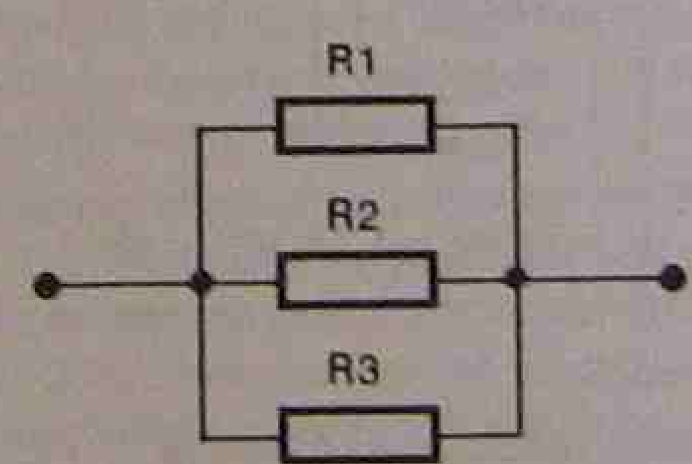
(a) The Series Circuit: The total resistance of a series circuit equals the sum of the individual resistor values. See Fig.1.



☐ $R_T = R1 + R2 + R3$

Fig.1: *The series resistive circuit*

(b) The Parallel Circuit: The total resistance of a parallel circuit can be calculated in various ways. See Fig.2.



☐ $1/R_T = 1/R1 + 1/R2 + 1/R3$

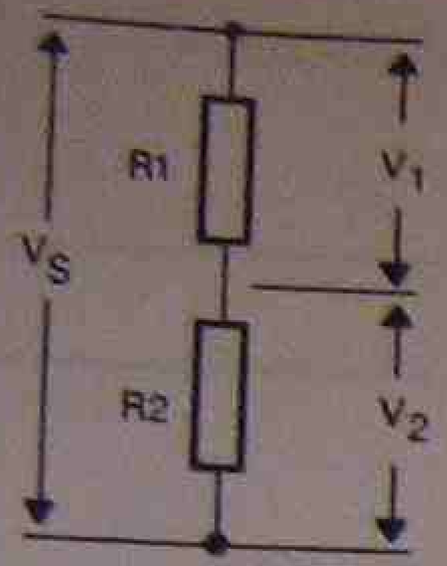
☐ $R_T = \dfrac{1}{1/R1 + 1/R2 + 1/R3}$

☐ $R_T = \dfrac{R1\,R2}{R1 + R2}$ (for 2 resistors)

Fig.2: *The parallel resistive circuit*

(c) Voltage Divider Equation: The potential divider equation is used to find the voltage across a resistor in a circuit containing two series connected resistors. Note the following points which refer to Fig.3:

- The equation is based on Ohm's law, where V = IR. The current in the circuit equals the applied voltage (Vs) divided by the total series resistance.
- The ratio of the voltage drops across both resistors equals the ratio of the resistor values. That is V1:V2 = R1:R2



$$V_1 = \left(\frac{V_s}{R1 + R2}\right) R1$$

$$V_2 = \left(\frac{V_s}{R1 + R2}\right) R2$$

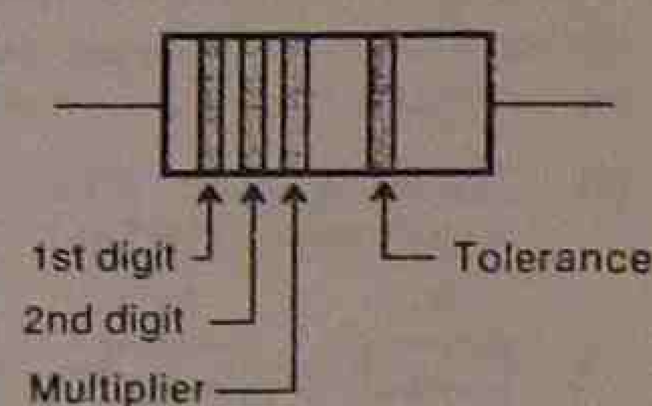
NOTE: $\frac{V_s}{R1 + R2}$ = current in R1 & R2

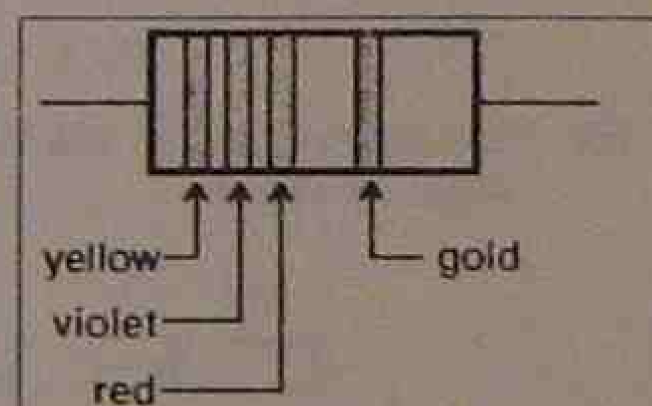
Fig.3: *The potential divider*

3. **Resistor Colour Code**

The resistor colour code is a standard used to identify the value of a resistor. Coloured bands are painted around the resistor and each colour represents a numerical value. Most resistors have four bands, in which the first two give the numerical value, the third (called the multiplier) the number of zeros that follow the numerical value and the fourth the tolerance of the resistor value. Resistors with five bands use the first three to give a three digit value and the fourth and fifth are the multiplier and tolerance bands respectively. Most resistors have a 5% tolerance. Fig.4 and the table of values show how the colour code is applied.

COLOUR	1st & 2nd Digits	Multiplier	Tolerance
Black	0	x 1	
Brown	1	x 10	1%
Red	2	x 100	2%
Orange	3	x 1k	
Yellow	4	x 10k	
Green	5	x 100k	
Blue	6	x 1M	
Violet	7	not used	
Grey	8	not used	
White	9	not used	
Gold	not used	x 0.1	5%
Silver	not used	x 0.01	10%





value = 4 7 x 100 = 4700 = 4k7

Fig.4: *The resistor colour code*

Resistor values are generally available in the 10% preferred range, in multiples of 1, 1.2, 1.5, 1.8, 2.2, 2.7, 3.3, 3.9, 4.7, 5.6, 6.8 and 8.2. The 5% range has values between those of the 10% range.

4. **Scientific Notation**

Very large or very small numbers are common in electronics, and the use of **scientific notation** makes these values easier to use. Scientific notation shortens the number by expressing the zeros as 10 raised to a power.

e.g. 2,200,000 = 2.2×10^6
0.0000047 = 4.7×10^{-6}

It is standard practice in electronics to refer to a multiplier (or power of 10) by a name or letter, usually Greek in origin. For example, 1000, which equals 10^3 is known as the **kilo**, or by the letter k. The following table lists those typically used in electronics.

Table of metric units

exa E	10^{18}		deci d	10^{-1}	
peca P	10^{15}		centi c	10^{-2}	
tera T	10^{12}		* milli m	10^{-3}	
* giga G	10^{9}		* micro μ	10^{-6}	
* mega M	10^{6}		* nano n	10^{-9}	
* kilo k	10^{3}		* pico p	10^{-12}	

Those units marked with * are commonly used in electronics. Note how the powers are all multiples of 3.

EXAMPLES

1 million ohms = 1×10^6 = **1M ohm**

1 millionth of a farad = 1×10^{-6}F = **1μF**

1 thousandth of an amp = 1×10^{-3} = **1mA**

5. **Multimeters**

Analog Multimeters have a mechanical "moving coil" as the readout. These operate on a magnetic field principle where the level of current flowing in the coil determines the strength of its magnetic field. This then tries to align the coil with its surrounding permanent magnet, against the force of a return spring. These meters must be mechanically calibrated but are relatively inexpensive.

Various resistance shunts and voltage dividers can be switched in to provide different measuring functions. The accuracy depends on the quality of the meter movement and the accuracy of reading the scale. A mirror is usually set into the scale so that the meter needle can be aligned with its reflection to minimise reading error.

Digital Multimeters have no moving parts in the readout. These meters contain a digital readout driven by an analog to digital conversion circuit. All range selection uses some form of voltage divider to reduce the input sample to the required input range. The selection switch also usually controls the readout so that the correct units are displayed.

6. **Sensitivity**

Analog moving coil meters are designed to give full scale deflection at a certain current; e.g. 100μA. This then becomes the smallest current range available for that multimeter and is an indication of its sensitivity. Sensitivity equals the **reciprocal of the full scale deflection current** and is expressed in **ohms per volt**. For example, a 100μA meter movement has a sensitivity of 1/100μA, or 10,000 ohms per volt. The current to operate the meter movement is supplied by the circuit under test, and the smaller the current (higher the sensitivity) the better, particularly when measuring volts.

The loading effect of a voltmeter is determined by calculating the equivalent resistance that meter represents for the particular voltage range it is set to. A meter with a sensitivity of 10,000 ohms per volt will represent a resistance value of 100,000 (or 100k ohms) when set to its 10V range. Ohm's law can then be used to calculate the effect on a circuit when a 100k ohm resistor is connected where the voltmeter is being used to measure a voltage.

Digital voltmeters (DVM) also take current from the circuit under test, but they represent a fixed value of resistance (usually 11M ohm) regardless of the selected voltage range. The following lists typical sensitivities for voltmeters.

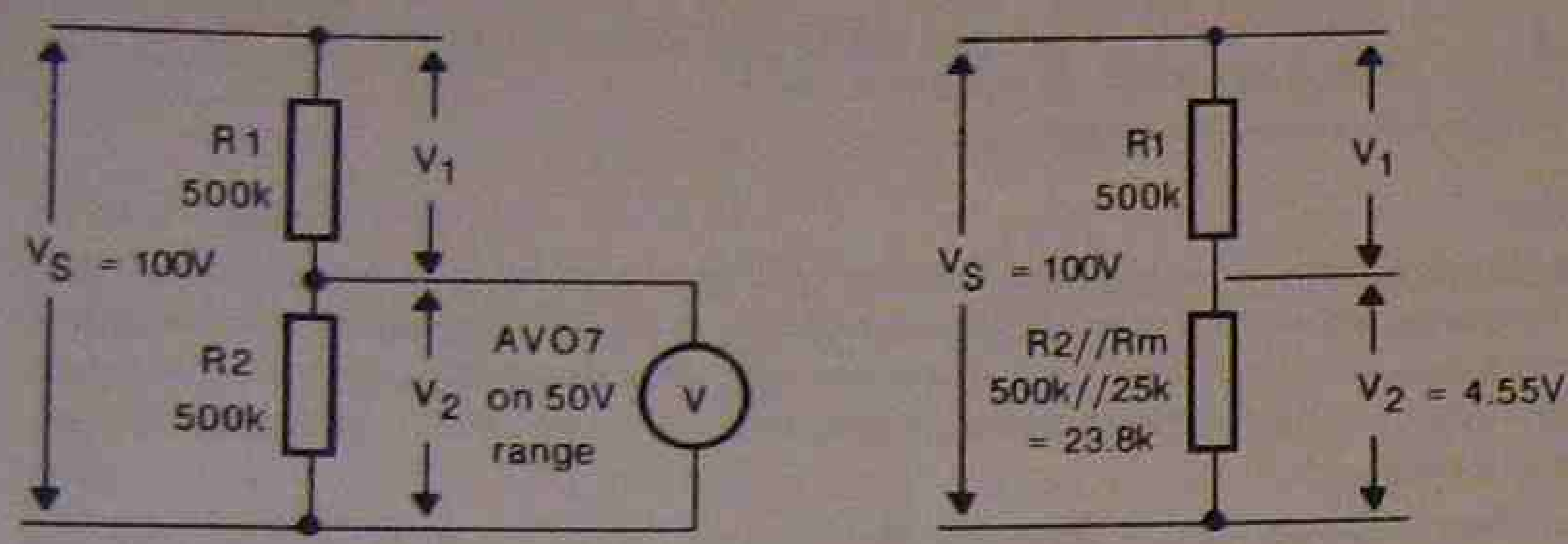
AVO7	=500 ohm/volt	(FSD current = 2mA)	On 10V range, equals a 5k resistor
AVO8	=20k ohm/volt	(FSD current = 50uA)	On 10V range, equals a 200k resistor
Arix	=100k ohm/volt	(FSD current = 10uA)	On 10V range, equals a 1M resistor
Digital	=11M ohm	(fixed resistance)	

Note that an Arix meter on its 100V range is the equivalent to a 10M ohm resistor, which compares favourably to the resistance of a digital voltmeter. When set to its 250V range, the equivalent resistance is 25M ohms, which exceeds that of a DVM.

7. **Circuit Loading**

Connecting a meter in a circuit effectively adds a resistor in parallel with the circuit under test. If the meter has a low internal resistance (low sensitivity) it can substantially reduce the voltage it is reading. This is referred to as voltmeter *loading*.

In the circuit of Fig.5(a), an AVO7 on its 50V range is connected to the circuit as shown. The AVO will have an equivalent resistance of 500 ohms/volt x 50V, which equals 25k ohms. Before the meter is connected the voltage across R2 will be 50V, or half the supply voltage as R1 equals R2.



(a) measuring voltage in a resistive circuit (b) effect caused by meter loading

Fig.5: *Meter loading*

When the meter is connected, the combined resistance of the meter and R2 equals 23.8k as shown in Fig.5(b). Using the potential divider equation, a voltage of 4.55 volts is now present across R2, and is the value that will be shown by the voltmeter. Using a meter with a much higher sensitivity is essential in a circuit that contains high value resistors, and even a DVM will cause some loading, effecting the voltage across R2. As a general rule, the meter resistance should be at least 10 times greater than the equivalent resistance of the circuit under test.

Ammeters also affect the current value being measured as an ammeter is effectively a resistance in series with the circuit under test. An ammeter contains a low value resistor, called a **shunt** that is connected in parallel with the meter movement in an analog meter or in parallel with the analog to digital converter (ADC) in a digital multimeter (DMM). The value of the shunt resistor depends on the FSD current of the meter movement (or ADC circuit in a DMM) and the lower the FSD current requirement the lower the value of the shunt resistor. A shunt is part of a current divider, through which all current other than that required by the meter movement passes. A shunt therefore has a finite value of resistance, though it is often as low as a few milliohms.

8. **Accuracy and resolution**

The accuracy of a meter is a function of its manufacture, and depends on the stability and quality of its components. Accuracy should not be confused with sensitivity or resolution. A 0.5% specification for accuracy, with a resolution of 1% is better than a meter with an accuracy of 5% but with a resolution of 0.004%. Digital meters have a much better resolution than analog meters, but may not always have the highest accuracy. Sensitivity is not a guarantee of high accuracy, although the higher the sensitivity the less the loading effect. The reading given by a high sensitivity meter will therefore be more likely to be correct, limited only by the accuracy and resolution of the meter.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 1

1. For the circuit of Fig.1, calculate the resistance between points A and B.

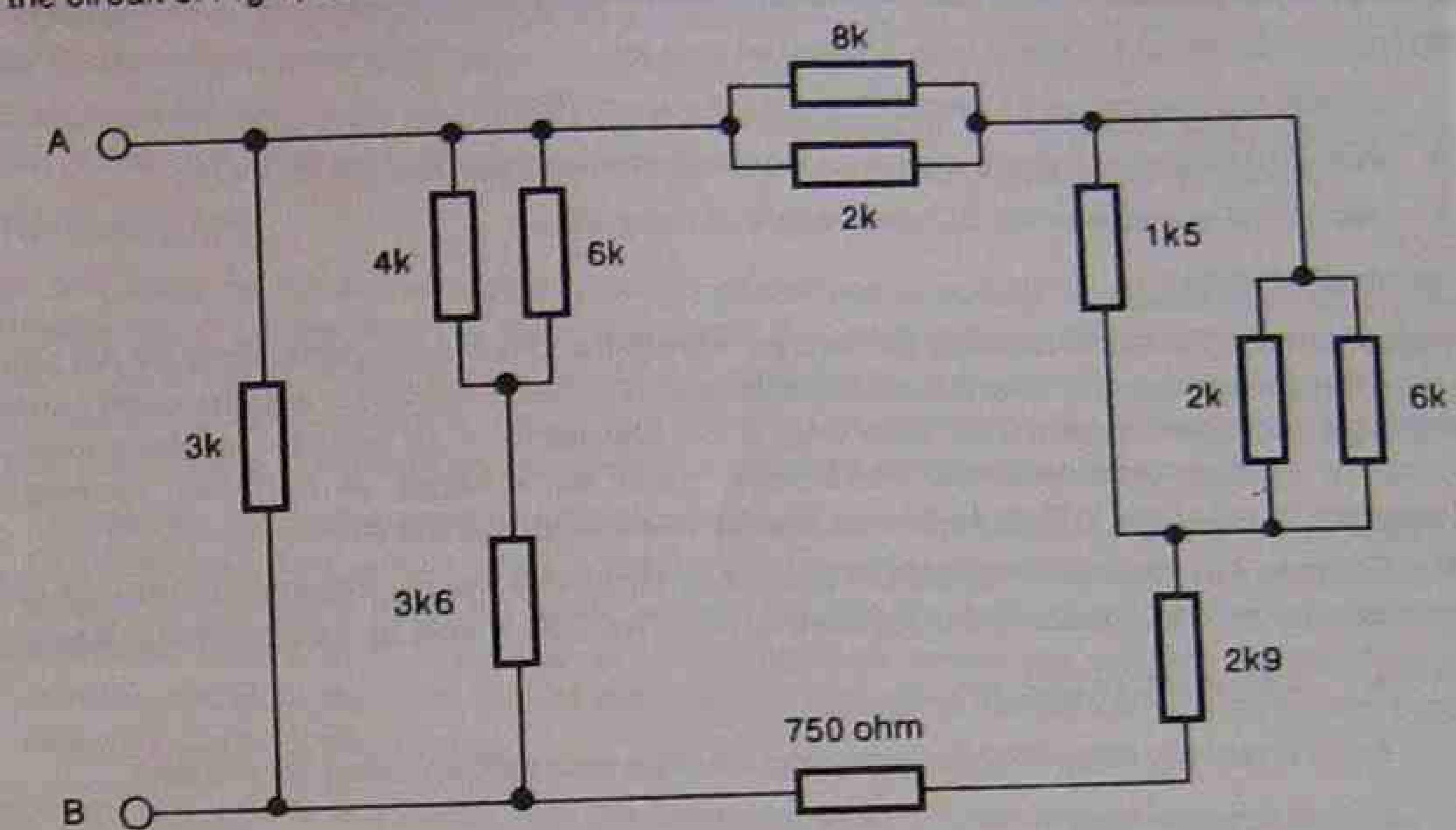


Fig.1

2. For the circuit of Fig.2, calculate the values of V1 and V2.

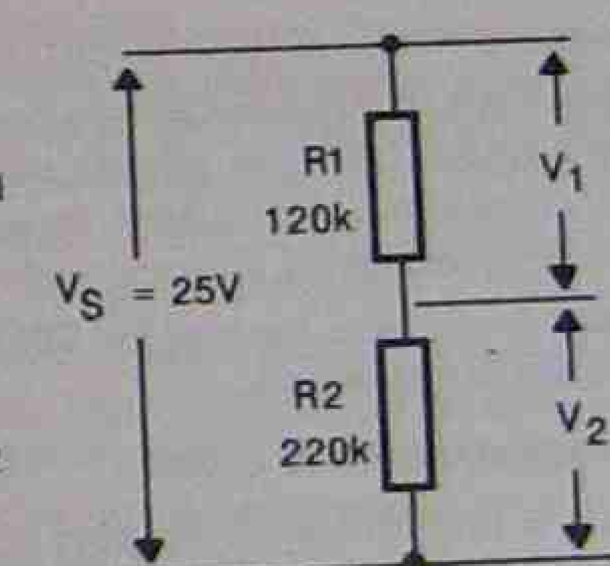


Fig.2

3. For the circuit of Fig.2, calculate the reading that will be shown by a voltmeter with a sensitivity of 10k ohms/volt set to its 10V scale.

4. Determine the values of the resistors shown in Fig.3.

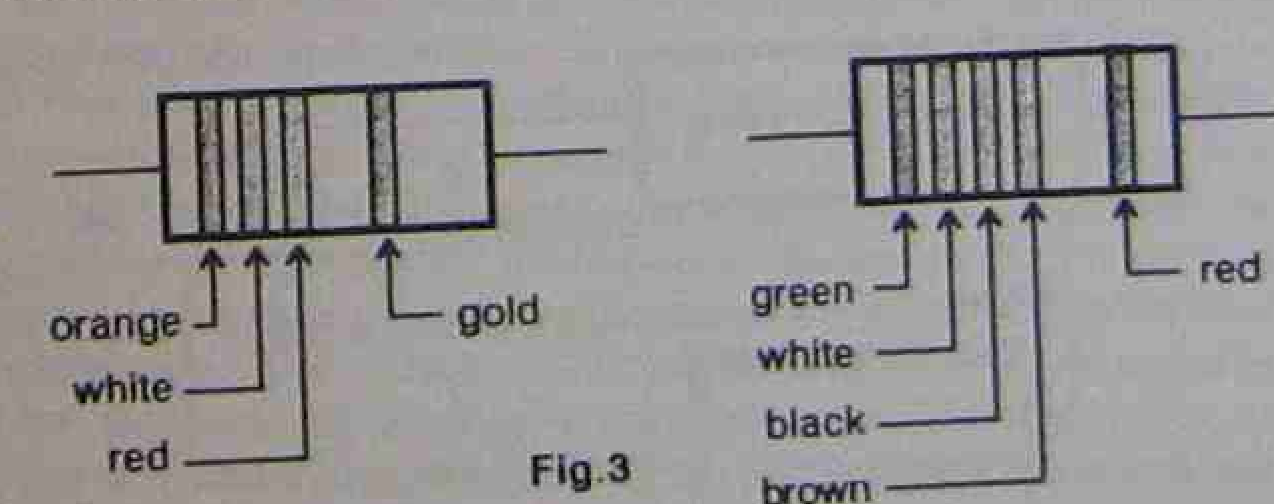


Fig.3

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 2

THE CATHODE RAY OSCILLOSCOPE

OBJECTIVES: At the end of this lesson you will be able to:

- Identify the controls of a Cathode Ray Oscilloscope. (CRO)
- Measure voltage, current, frequency and phase shift using a dual trace CRO.
- Use a CRO to measure the DC component of a signal.

1. INTRODUCTION

A multimeter is limited in its ability to measure voltages and currents, particularly for AC signals. The frequency range of a typical multimeter is limited to less than 1kHz, and the scale is usually calibrated for RMS values, and then only if the waveform is sinusoidal. In electronics it is common to have non-sinusoidal waveforms, either as voltages or currents. As well, the frequency of the signal is likely to be over the 1kHz limit of a multimeter.

The Cathode Ray Oscilloscope, (CRO) is a more expensive measuring instrument and is less portable, but in most situations it is far more useful. The CRO is able to measure the following:

- voltage (AC or DC)
- current (by measuring voltage drop across a resistorthen using Ohm's law)
- period
- frequency (by calculation from the period measurement)
- phase difference (between two waveforms)

These values can be measured for any type of waveform at frequencies only limited by the CRO itself and a typical CRO can provide useful measurements for signals up to 20MHz. Some instruments extend beyond 100MHz. Because the signal is actually displayed, the shape of the waveform can be examined, allowing more accurate analysis of the circuit behaviour.

These notes briefly describe how the CRO operates and how it is used to measure the various characteristics of a waveform.

2. BLOCK DIAGRAM OF THE CRO

Fig.1 shows a simplified block diagram of a CRO. The basic sections are the CRO tube, the power supply, the vertical (or Y) amplifier and the time base (X). Most CROs are dual trace, meaning they can display two signals simultaneously. However, for the purposes of explanation, a single trace CRO is assumed.

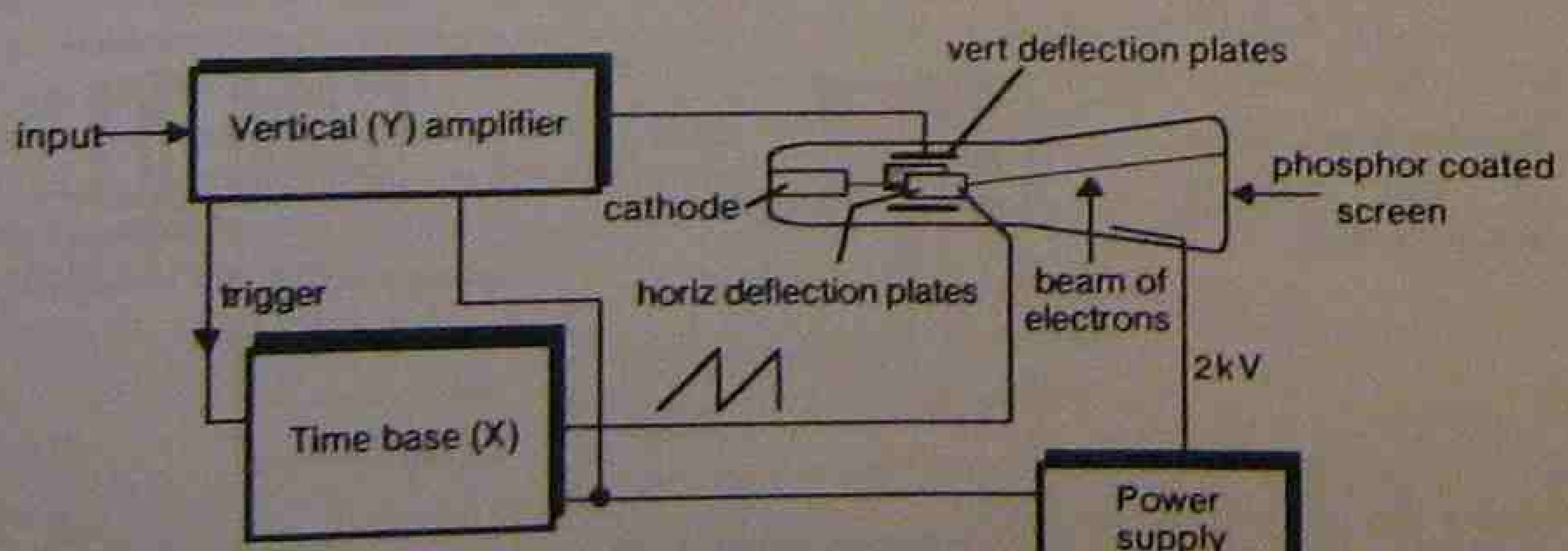


Fig.1: Block diagram of a CRO

3. THE CATHODE RAY TUBE (CRT)

The screen of an oscilloscope is at the anode end of a CRT. The CRT is a type of thermionic valve and has a coating of phosphor on the inside of the anode, which glows when struck by an electron beam. The electron beam originates from the cathode which is heated by a filament so that it emits electrons. The beam is attracted to the anode by a high positive DC potential (2kV or so) relative to the cathode. Deflection plates are placed inside the tube so that the beam can be deflected sideways (X direction) or vertically (Y direction). Deflection is caused when one plate has a different potential to the other (electrostatic deflection), unlike a TV set which uses magnetic deflection. Other electrodes are used to focus the beam and to accelerate it. These electrodes usually have a DC potential of several hundred volts to over 1kV.

To allow waveforms to be measured, a graticule consisting of a 1cm grid is either etched on the front of the tube, or drawn on a transparent panel placed at the viewing end of the tube.

4. THE TIME BASE

The time base, or horizontal section consists of a sweep generator (oscillator) and a horizontal amplifier which connects to the deflection plates that control the sideways (X) movement of the beam. The beam is moved across the screen from left to right at a preselected speed, then is returned to the left hand side (retrace) very quickly. During retrace, the beam is turned off so the retrace is invisible. The required waveform is therefore a sawtooth, as shown in Fig.1.

- The sweep generator provides the time base control for viewing signals, by producing a sawtooth waveform which controls the horizontal movement of the beam on the screen. The sawtooth waveform must be linear so that the scan rate is constant. The period of the sawtooth waveform is adjustable in calibrated steps which are selected by the Time/Div control. Because the scan rate of the beam is calibrated the period of a waveform can be directly read from the screen.

An important control associated with the sweep generator is the **trigger** control. The idea is to synchronise the sweep generator with the input signal. Typically a trigger pulse is generated by the input signal when the signal passes through zero volts or through some preset voltage value. If the trigger control is set to accept only the positive trigger pulses, the sweep generator will produce one cycle of the sawtooth waveform whenever a positive trigger pulse occurs. This causes the waveform to appear stationary on the face of the CRT. To allow a trace to appear on the screen in the absence of an input signal, another control is used to turn off the trigger input allowing the sweep generator to 'free run'. On some CROs, this control is labelled 'AUTO', and if it is not selected there will be no trace on the screen, regardless of any other settings.

- The horizontal amplifier drives the deflection plates by amplifying the output of the sweep generator to a level suitable for deflecting the beam. Most CROs have a facility to connect an external signal to the horizontal amplifier instead of the output of the sweep generator. This allows **Lissajous** patterns to be displayed on the screen, in which circles, the ABC logo and other interesting patterns are produced by feeding two sinewaves (or other waveshapes) to the CRO, one to the horizontal (X) amplifier and the other to the vertical (Y) amplifier.

5. THE VERTICAL SECTION

The vertical section (Y) accepts the input signal and after suitable amplification, applies it to the vertical deflection plates. The beam will therefore move up and down at a speed depending on the instantaneous potential difference between the deflection plates. To create the potential difference between the plates, one plate is fed with a signal that is 180 degrees out of phase with the other. In the absence of any input signal, (assuming auto trigger), the beam will only be deflected in the horizontal direction, giving a straight line display on the screen. If a DC signal is applied, the beam will be deflected vertically, and will appear as a straight line, but moved from its previous position, depending on the value of the input DC voltage. If the input signal contains an AC and a DC component, the trace will show as a waveform moved from the zero, or reference line. Thus, the shape of a waveform is produced by the beam being moved vertically by the Y amplifier and horizontally by the X amplifier.

Fig.2 shows a simplified circuit of the input switching for the signal prior to its application to the vertical (Y) amplifier. It contains a calibrated attenuator (SW2) and a non-calibrated attenuator (VR1) that are both used to reduce the level of the signal applied to the vertical amplifier. This is necessary to keep the display on the screen from exceeding the available display height. The calibrated attenuator is a switch that selects the volts/division and consists of a number of series connected resistors that form a potential divider. For example, if the setting is 10V/division on the graticule, a waveform that extends for three divisions has a peak to peak voltage of 30V.

The potentiometer VR1 can further reduce the amplitude of the signal but if is used, the settings of the main attenuator are no longer calibrated and the value of the input voltage cannot be read from the screen.

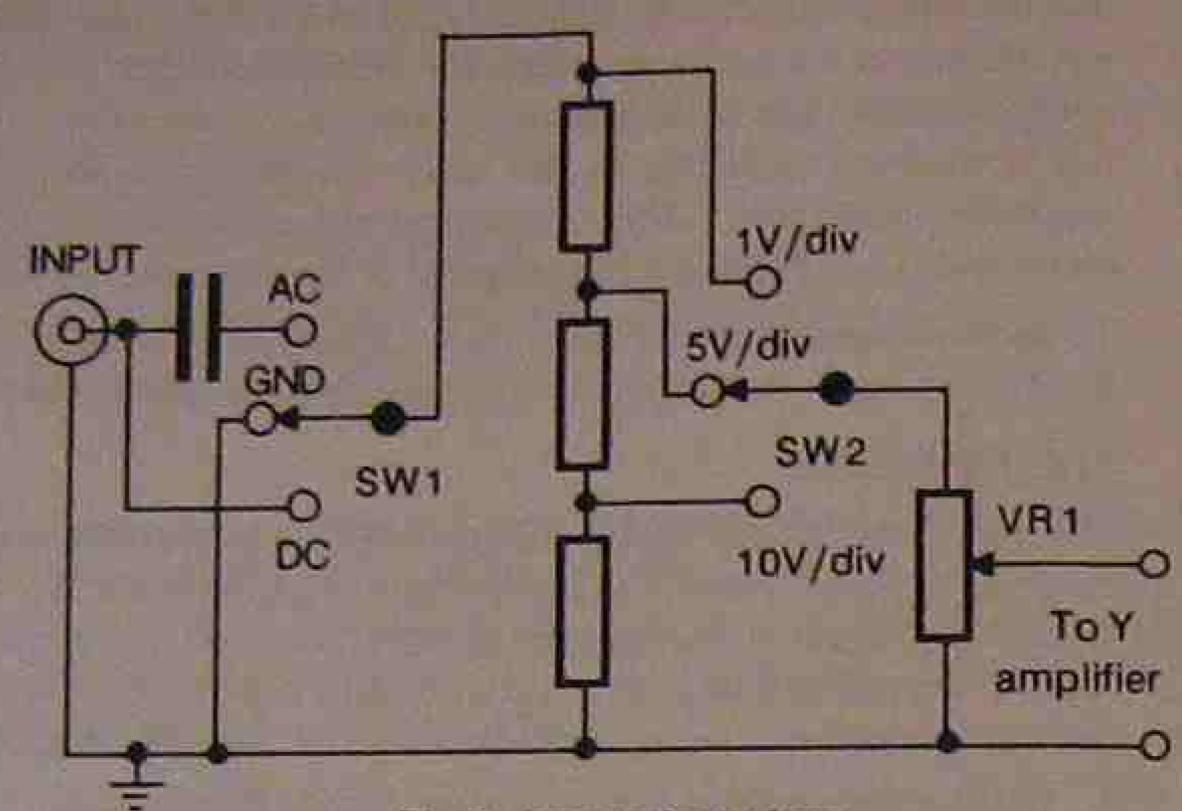


Fig.2: Input attenuators

Other vertical input controls include the vertical position control, and a three pole input coupling switch (SW1). The three positions of this switch are:

- AC: this position connects the input signal via a coupling capacitor and allows the AC component of a waveform to be viewed while blocking the DC component. For example, a small AC ripple on top of a large DC voltage can be viewed.
- GND: in this position a zero reference voltage (ground) is applied to the input of the vertical amplifier so that it can be used as a reference point for measurements. The beam is usually positioned centrally on the screen with the Y shift control.
- DC: this position is the direct coupled mode and both the AC and the DC component of a waveform are passed to the attenuators. When measuring a DC voltage, this switch setting is required.

6. SIGNAL WAVEFORMS

The CRO is used to observe the shape of a waveform, to measure its peak to peak voltage and its period. If the CRO has two beams, it can also be used to measure the phase difference between two signals. There are many types of waveforms in electronics, in which the sinewave is the most basic. The relationships between the various voltage values for a sine wave are shown below.

> $V_{max} = V_{p\text{-}p}/2$ (Vmax is the voltage from 0V to the maximum of the waveform)
>
> $V_{RMS} = 0.71 V_{max}$
>
> $V_{average} = V_{DC} =$ 0V (for whole cycle)

The period of any waveform is the time between two identical points of the waveform. Usually, the period is measured between two adjacent positive or two negative going zero crossing points, but it can be measured between the positive peaks, the negative peaks and so on.

Fig.3 shows how a CRO can measure the peak to peak voltage and the period of a waveform.

Vp-p = **8V** as the V/div setting = 2V/div and the height of the wave is 4 divisions.

The period is **20mS** as the time/div setting is 5mS, and a cycle covers 4 divisions.

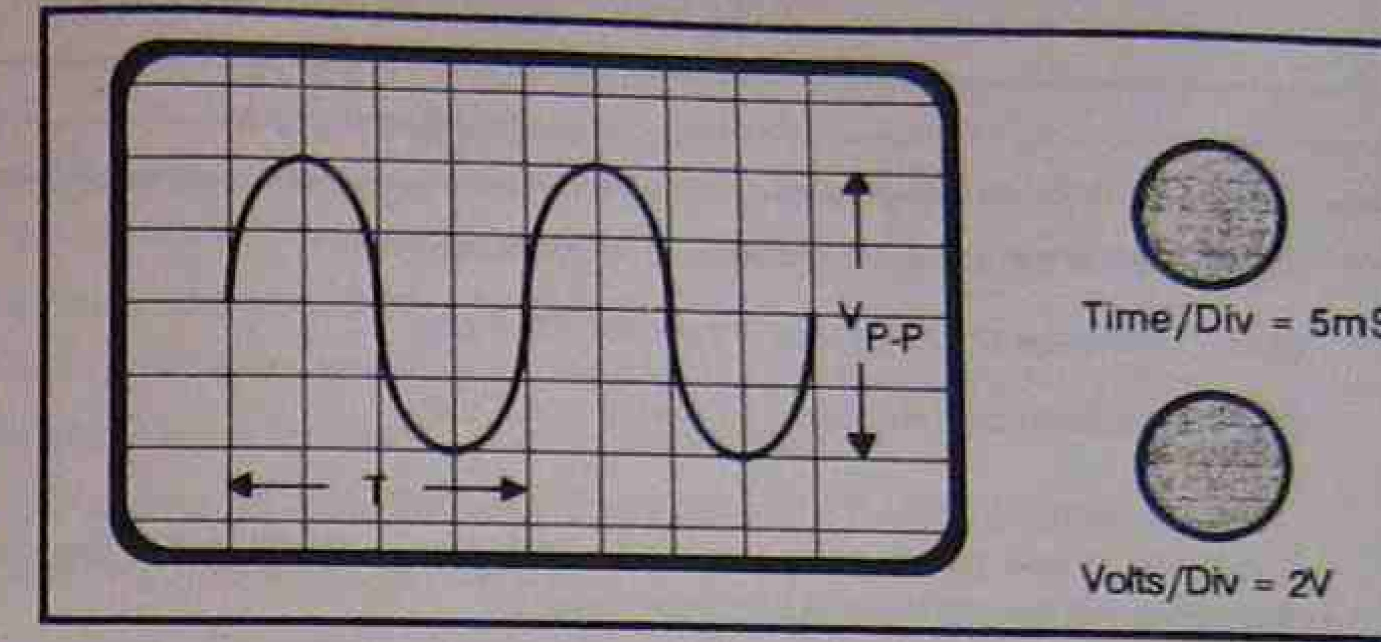


Fig.3: *Measuring period and peak to peak voltage on a CRO*

The frequency of any waveform is determined by dividing the period of the waveform into 1. That is, **frequency equals the reciprocal of period**. For Fig.3, the frequency is the reciprocal of 20ms which equals 50Hz

To determine the RMS value of the waveform shown in Fig.3, the maximum voltage is either measured or derived from the peak to peak value. In Fig.3, V_{max} = 4V and the RMS voltage therefore equals 0.71 x 4 which gives 2.84V. Because the waveform is symmetrical around 0V, the average, or DC value is 0V. If the waveform was superimposed on a DC voltage, the display would have been shifted up (for positive voltages) or down (for negative voltages), indicating that a DC component was present. This would give a waveform that was no longer symmetrical around the zero volts line, and the average value of the waveform would equal the DC component present in the signal.

For waveforms other than sinewaves, the average and the RMS voltages need to be determined mathematically in which:

- the average value is determined by summing a fixed range of instantaneous voltage values over one cycle with an equal number of values from both the positive and the negative half cycles. The average value equals the total of all the instantaneous values, divided by the number of values taken.
- RMS values are calculated over one cycle by adding the squares of the amplitude per time unit, dividing this sum by the number of time units, and taking the square root. (RMS = Root Mean Squared).

It is unusual to have to calculate these values for non-sinusoidal waveforms, and for the purposes of this subject, sine waves will be assumed.

7. WAVEFORMS WITH A DC COMPONENT

In electronics it is common to have an AC waveform superimposed on a DC voltage. In Fig.4, an AC signal is coupled through a capacitor to a potential divider connected to 20V DC. Because both resistor values are equal, a 10V DC potential will be added to the AC input signal. The resulting waveform is the AC signal with a 10V DC component. The AC waveform is now symmetrical around the 10V line rather than the 0V line. On a CRO, the waveform would be shifted vertically on the screen, unless AC coupling was selected.

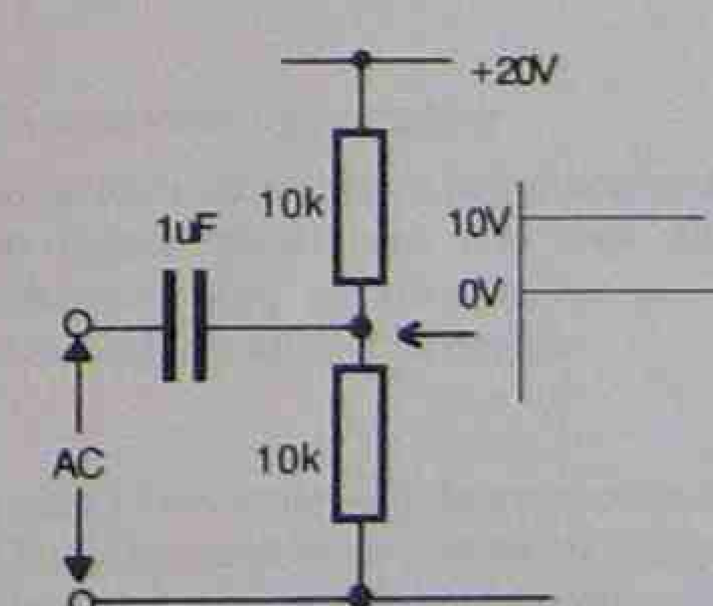


Fig.4: *AC combined with DC*

****

THEORY ASSIGNMENT 2

1. Calculate the frequencies of the following waveforms:
   (a) Timebase on 20 μs/cm 1 cycle = 2.5 cm
   (b) Timebase on 1ms/cm 1 cycle = 4.1 cm

2. A 2.5kHz, 2 volt p-p square wave is applied to a CRO. If the timebase is set to 100μs/cm, determine the length (in cms) of one cycle.

3. Determine the following from the display shown in Fig.1:
   (a) Peak to peak voltage for both waveforms if the volts/division switch is set to:
      i) 5V/div
      ii) 200mV/div
      iii) 40V/div
   (b) Calculate the phase difference between the two waveforms and indicate which is lagging.

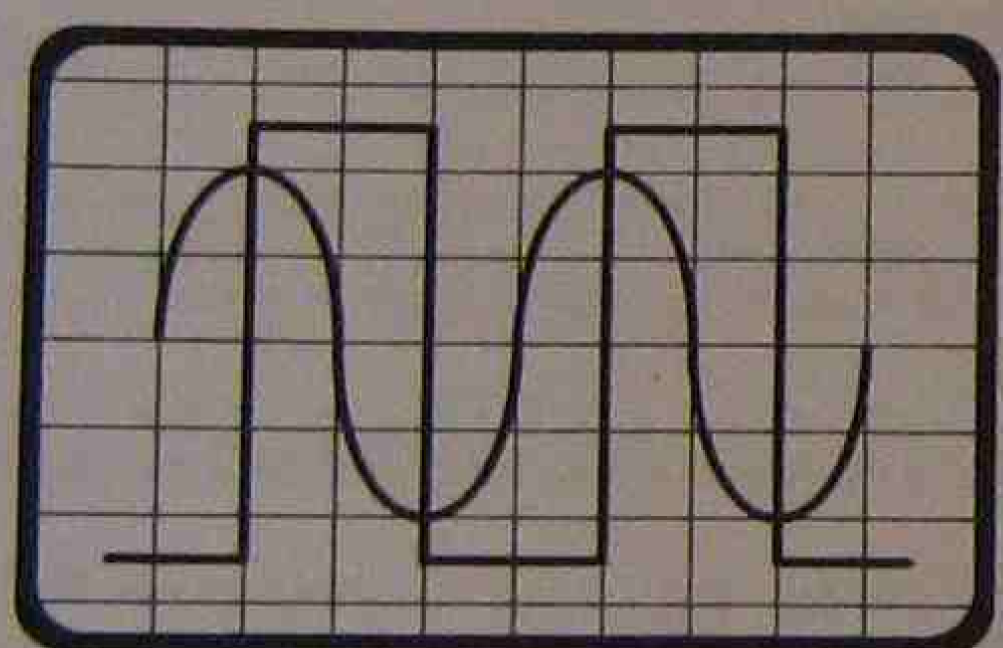


Fig.1

****

PRINCIPLES OF AMPLIFICATION AND TRANSISTOR CHARACTERISTICS

OBJECTIVES At the end of this lesson you should be able to:

- Sketch the equivalent circuit of an amplifier that shows its operating characteristics.
- Define the term amplification as applied to electronic amplifiers.
- Calculate the voltage gain for an amplifier given the output and input voltages.
- List the basic characteristics of a bipolar transistor.

1. INTRODUCTION

An amplifier is the basic building block of analog electronics. Its function is to amplify an electrical quantity, such as voltage, current or power. All amplifiers have certain characteristics and these need to be known if an amplifier is to be used in a particular application. These notes describe the basic characteristics of an amplifier and their effect in a circuit. The equivalent circuit of an amplifier is also described.

The transistor is the basic amplifying device and these notes also briefly describe some of the operating characteristics of the transistor.

2. AMPLIFICATION

The term amplification in electronics is used to describe the action of a circuit that produces an output signal larger than the input signal. An ideal amplifier will produce an output signal that has exactly the same shape as the input, except it will be larger. The term amplification is derived from the term amplitude, which refers to the height of a waveform. Most amplifiers accept a voltage as the input signal, but the electrical quantities of power and current can also be amplified. Amplifiers are not necessarily confined to electronics, and other types of amplifiers include the magnetic amplifier, hydraulic, pneumatic and mechanical types.

Fig.1 shows the general form of an amplifier, in which the input signal is used to control a power source. The output signal is therefore produced by the power source, and the amplifier section has the task of controlling the power source. In an ideal amplifier the power or energy source has an infinite capacity.

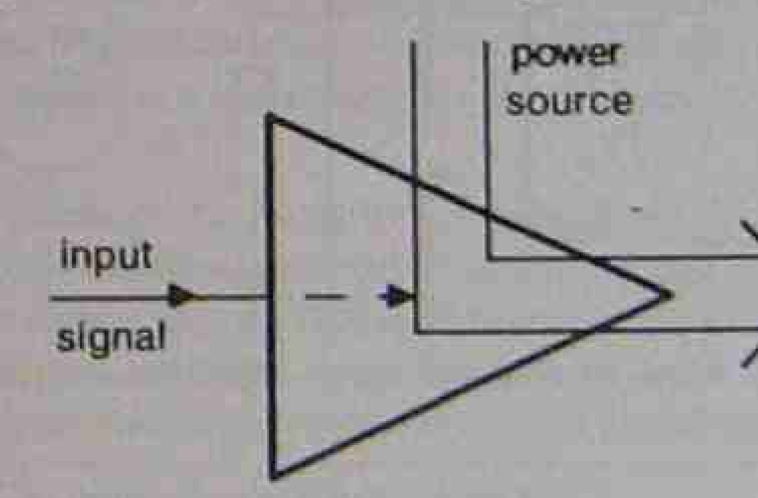


Fig.1: General form of an amplifier

An amplifier is typically used to amplify the electrical signal from some type of signal source and to drive an output load. These devices are often called transducers and their basic characteristics need to be explained so that the characteristics of an amplifier can be understood. Transducers are more fully described in other subjects within this course.

3. INPUT AND OUTPUT TRANSDUCERS

A transducer is a device that converts from one energy form to another, and in electronics one of the energy forms will be an electrical quantity. An input transducer produces an electrical signal as a result of heat, mechanical movement, chemical action, magnetism and other energy forms. Typical input signal sources are tacho generators, thermocouples, tape recorder heads, strain gauges, record player pickups, pH cells, Hall effect devices and so on. An output transducer converts an electrical signal into another form of energy. Typical output transducers are motors, a loudspeaker, optical cable drivers and so on.

Fig.2 shows the equivalent circuit of an input transducer, also called a signal source, in which an internal voltage source connects to the output terminal of the device through a series resistor Rs. The important aspect of Fig.2 is the value of the source resistance Rs.

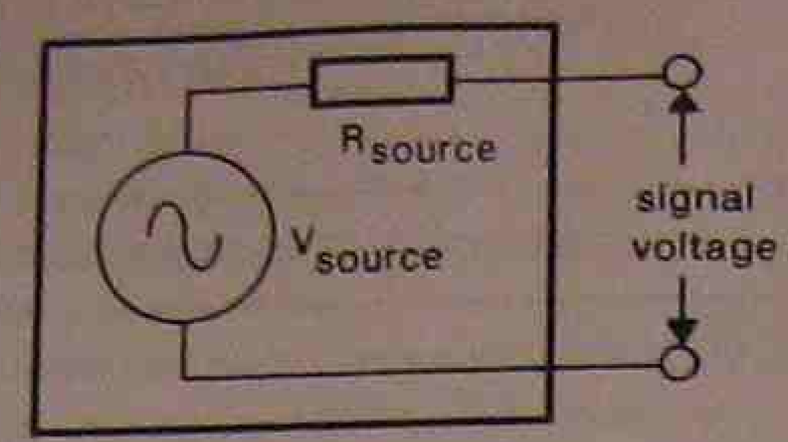


Fig.2: Equivalent circuit of a signal source

If the signal source is connected to a CRO, the signal that will be displayed should equal the voltage being produced by the internal voltage source of the device. However, if a resistance equal to the source resistance of the device is connected across its output terminals, the voltage displayed on the CRO will drop to half its previous reading. This is similar to the loading effect described in the notes for week 1. Therefore, if a signal source is connected to an amplifier, it is important that the amplifier itself doesn't load the signal source.

An output transducer is the load for the amplifier, and the important characteristic to be considered is the value of resistance the load represents. In most cases, an output transducer will have a low resistance. Motors or loudspeakers all have an impedance measuring a few ohms, and these devices need to be driven by a power amplifier. A voltage amplifier cannot supply power, and their output load needs to have a much higher resistance.

4. AMPLIFIER CHARACTERISTICS

As for any electrical circuit, an amplifier has a number of operating characteristics which collectively define how the amplifier can be used.

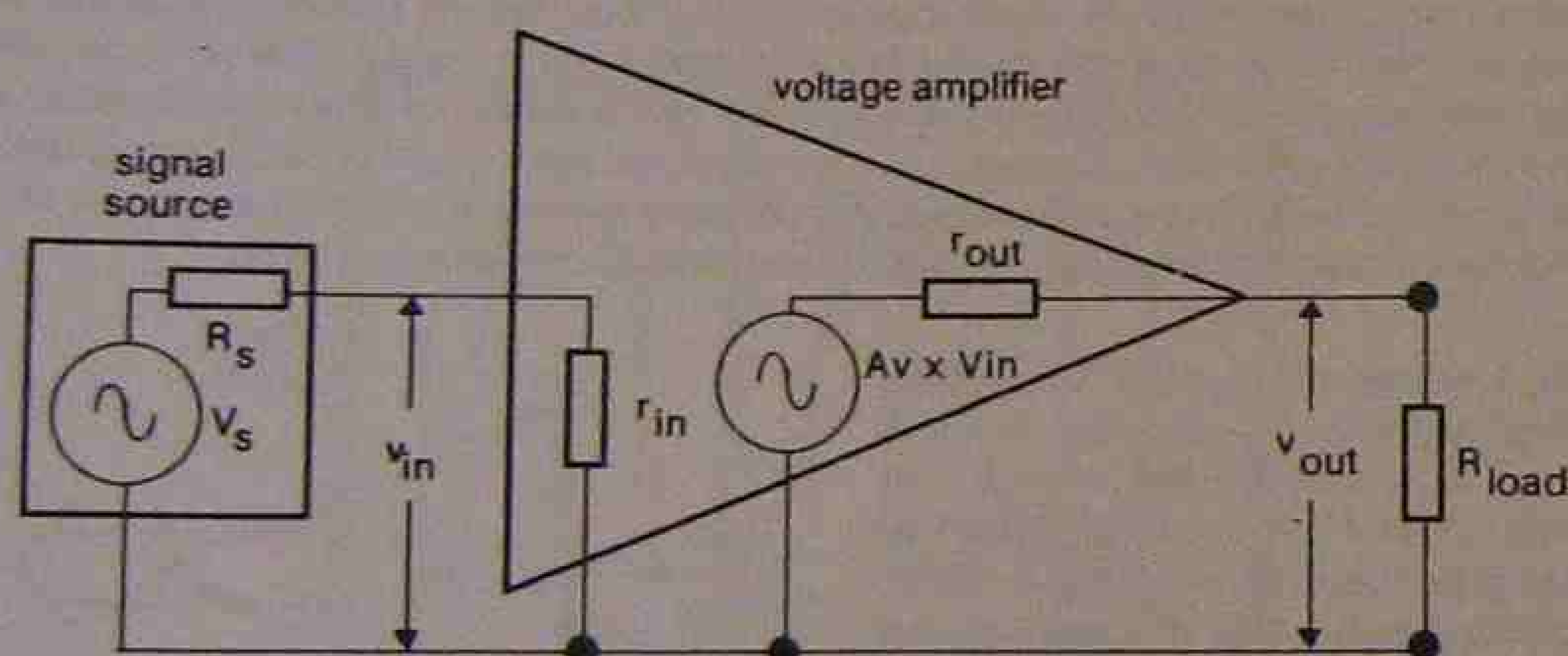


Fig.3: Equivalent circuit of a voltage amplifier

Fig.3 shows the equivalent circuit of a voltage amplifier connected to a signal source and a load. Note that this diagram can be modified to describe a power amplifier or a current amplifier.

The most important characteristic of an amplifier is its gain which is defined as the **ratio of its output to its input**. Thus:

- Voltage gain (Av) is the ratio of the output voltage of an amplifier to its input voltage. In Fig.3, the voltage gain is shown by the internal voltage generator labelled Av x vin. This generator represents that part of the amplifier circuit causing the input signal to be amplified.
- Current gain (Ai) is the ratio of the output current and the input signal current to the amplifier.
- Power gain (Ap) is the ratio of the output power of the amplifier to the power applied to its input.

$$\square \quad Av = \frac{Vo}{Vin} \qquad Ai = \frac{Io}{Iin} \qquad Ap = \frac{Po}{Pin}$$

The next two characteristics are the input impedance and the output impedance of the amplifier. The term impedance is used to describe any capacitance or inductance present with the resistance value. In these notes, the only electrical quantity that will be considered is the resistance, and the terms input resistance and output resistance will be used. However, a practical amplifier will have capacitance and inductance which can affect its performance at particular frequencies.

- Input resistance (rin) is the resistance present between the input terminals of an amplifier. It cannot be measured with an ohmmeter, as the resistance comprises the effects of all the components within the amplifier. Instead it has to be determined by measuring the input signal voltage and the input signal current, then calculated with Ohm's law. That is:

$$\square \quad rin = \frac{Vin}{Iin}$$

The input resistance of an ideal amplifier is infinity, or an open circuit. Practical amplifiers have input resistance values ranging from a few ohms to hundreds of megohms. A typical transistor amplifier has an input resistance of several thousand ohms.

Input resistance is important as it determines the type of signal source that can be connected to the amplifier. If the amplifier has a low input resistance, the signal source connected to it needs to have a correspondingly low source resistance. An amplifier with a high input resistance can be connected to any type of signal source, as the amplifier will not load the source.

- Output resistance (ro) is the resistance value between the internal voltage source of the amplifier and its output terminal. The ideal value of output resistance is zero, and a practical amplifier will have an output resistance anywhere from a few ohms to several thousand ohms.

In fact, an amplifier can be seen as a form of signal source itself, although it needs an electrical input signal to operate. The value of the output resistance determines the type of load that can be connected to the amplifier. If the output resistance is low, then a low resistance load can be connected, assuming the amplifier is able to produce the required load current. A voltage amplifier with a low output resistance cannot drive a low resistance load as it is generally unable to supply the high value of current required without overheating. An amplifier with a high output resistance can only drive high resistance loads, as a low resistance load will cause the output of the amplifier to be reduced.

Like input resistance, output resistance needs to be determined experimentally. This can be achieved in a number of ways, and a commonly used method is to first measure the unloaded output voltage of the amplifier. A variable resistor is then connected across the output terminals and its value adjusted until the output of the amplifier drops to half its previous value. The resistance of the potentiometer will now equal the output resistance of the amplifier.

Summary of input and output resistance

☐ Input resistance (rin):	ideal	=	infinity,	practical:	>10 x Rs
☐ Output resistance (ro):	ideal	=	zero,	practical:	$< \frac{ro}{10}$

5. THE TRANSISTOR

These notes assume students are already familiar with the basic operation of a transistor. For further details, refer to a recommended text. The following summarises some aspects of the operation of a transistor.

A transistor contains two PN junctions and has three terminals labelled base, collector and emitter. This type of transistor is referred to as a bipolar junction transistor (BJT) and other transistors include the field effect type (FET). For the purposes of these notes, the term transistor refers to the BJT.

CONTENTS

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 1

REVISION - DC MEASUREMENT

OBJECTIVES: At the end of this lesson you will be able to:

- Apply Ohm's law in solving basic calculations.
- Use scientific notation and metric prefixes in calculations and measurements.
- Calculate the loading effect of a voltmeter and its effect in a circuit.

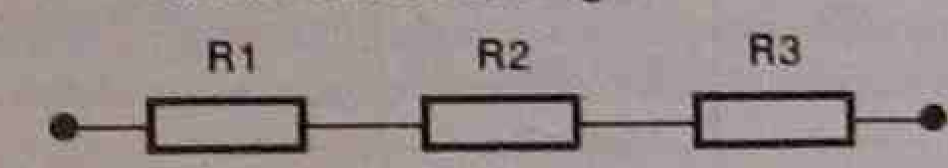
1. INTRODUCTION

Ohm's law is an essential tool in calculating voltages, currents and other values in an electronic circuit. Being able to apply Ohm's law is often more difficult than simply knowing the three basic equations that relate voltage, current and resistance. This lesson revises Ohm's law and uses these laws to show the loading effect a voltmeter has on the voltage it is measuring. All measuring instruments affect the reading they are taking, and a knowledge of the effect is important in deciding on the best type of instrument to use in a particular application.

2. OHM'S LAW IN RESISTIVE CIRCUITS

The three basic Ohm's law equations as applied to a resistive circuit are shown below. Note that the potential divider equation is often used in electronics.

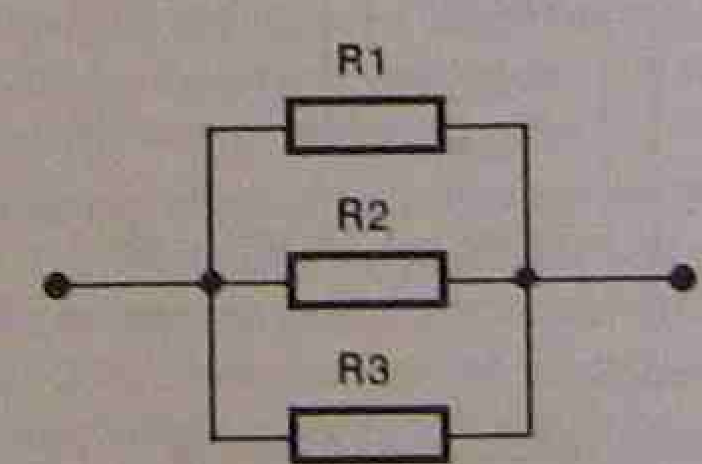
(a) The Series Circuit: The total resistance of a series circuit equals the sum of the individual resistor values. See Fig.1.



☐ $R_T = R1 + R2 + R3$

Fig.1: *The series resistive circuit*

(b) The Parallel Circuit: The total resistance of a parallel circuit can be calculated in various ways. See Fig.2.



☐ $1/R_T = 1/R1 + 1/R2 + 1/R3$

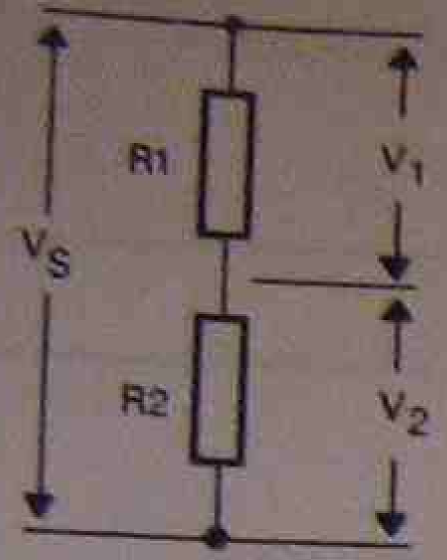
☐ $R_T = \dfrac{1}{1/R1 + 1/R2 + 1/R3}$

☐ $R_T = \dfrac{R1\,R2}{R1 + R2}$ (for 2 resistors)

Fig.2: *The parallel resistive circuit*

(c) Voltage Divider Equation: The potential divider equation is used to find the voltage across a resistor in a circuit containing two series connected resistors. Note the following points which refer to Fig.3:

- The equation is based on Ohm's law, where V = IR. The current in the circuit equals the applied voltage (Vs) divided by the total series resistance.
- The ratio of the voltage drops across both resistors equals the ratio of the resistor values. That is V1:V2 = R1:R2



☐ $V_1 = \left(\frac{Vs}{R1 + R2}\right) R1$

☐ $V_2 = \left(\frac{Vs}{R1 + R2}\right) R2$

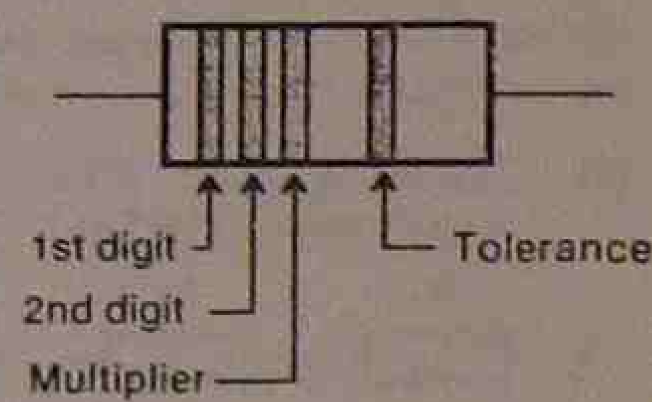
NOTE: $\frac{Vs}{R1 + R2}$ = current in R1 & R2

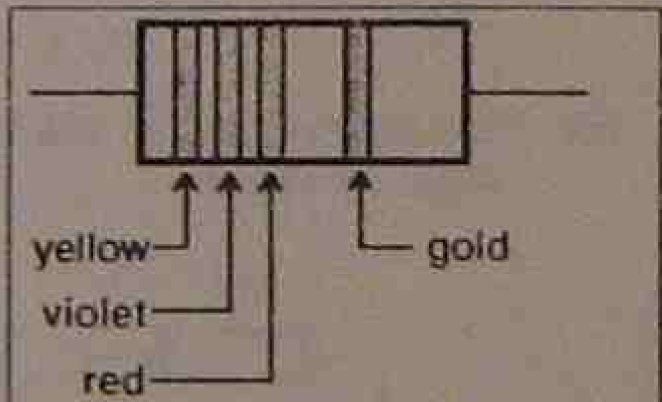
Fig.3: *The potential divider*

3. Resistor Colour Code

The resistor colour code is a standard used to identify the value of a resistor. Coloured bands are painted around the resistor and each colour represents a numerical value. Most resistors have four bands, in which the first two give the numerical value, the third (called the multiplier) the number of zeros that follow the numerical value and the fourth the tolerance of the resistor value. Resistors with five bands use the first three to give a three digit value and the fourth and fifth are the multiplier and tolerance bands respectively. Most resistors have a 5% tolerance. Fig.4 and the table of values show how the colour code is applied.

COLOUR	1st & 2nd Digits	Multiplier	Tolerance
Black	0	x 1	
Brown	1	x 10	1%
Red	2	x 100	2%
Orange	3	x 1k	
Yellow	4	x 10k	
Green	5	x 100k	
Blue	6	x 1M	
Violet	7	not used	
Grey	8	not used	
White	9	not used	
Gold	not used	x 0.1	5%
Silver	not used	x 0.01	10%





value = 4 7 x 100 = 4700 = 4k7

Fig.4: *The resistor colour code*

Resistor values are generally available in the 10% preferred range, in multiples of 1, 1.2, 1.5, 1.8, 2.2, 2.7, 3.3, 3.9, 4.7, 5.6, 6.8 and 8.2. The 5% range has values between those of the 10% range.

4. Scientific Notation

Very large or very small numbers are common in electronics, and the use of **scientific notation** makes these values easier to use. Scientific notation shortens the number by expressing the zeros as 10 raised to a power.

e.g. $2,200,000 = 2.2 \times 10^6$
$0.0000047 = 4.7 \times 10^{-6}$

It is standard practice in electronics to refer to a multiplier (or power of 10) by a name or letter, usually Greek in origin. For example, 1000, which equals 10^3 is known as the **kilo**, or by the letter k. The following table lists those typically used in electronics.

Table of metric units

exa	E	10^{18}	deci	d	10^{-1}
peca	P	10^{15}	centi	c	10^{-2}
tera	T	10^{12}	* milli	m	10^{-3}
* giga	G	10^{9}	* micro	μ	10^{-6}
* mega	M	10^{6}	* nano	n	10^{-9}
* kilo	k	10^{3}	* pico	p	10^{-12}

Those units marked with * are commonly used in electronics. Note how the powers are all multiples of 3.

EXAMPLES

1 million ohms = 1×10^6 = **1M ohm**

1 millionth of a farad = 1×10^{-6}F = **1μF**

1 thousandth of an amp = 1×10^{-3} = **1mA**

5. Multimeters

Analog Multimeters have a mechanical "moving coil" as the readout. These operate on a magnetic field principle where the level of current flowing in the coil determines the strength of its magnetic field. This then tries to align the coil with its surrounding permanent magnet, against the force of a return spring. These meters must be mechanically calibrated but are relatively inexpensive.

Various resistance shunts and voltage dividers can be switched in to provide different measuring functions. The accuracy depends on the quality of the meter movement and the accuracy of reading the scale. A mirror is usually set into the scale so that the meter needle can be aligned with its reflection to minimise reading error.

Digital Multimeters have no moving parts in the readout. These meters contain a digital readout driven by an analog to digital conversion circuit. All range selection uses some form of voltage divider to reduce the input sample to the required input range. The selection switch also usually controls the readout so that the correct units are displayed.

6. Sensitivity

Analog moving coil meters are designed to give full scale deflection at a certain current; e.g. 100μA. This then becomes the smallest current range available for that multimeter and is an indication of its sensitivity. Sensitivity equals the **reciprocal of the full scale deflection current** and is expressed in **ohms per volt**. For example, a 100μA meter movement has a sensitivity of 1/100μA, or 10,000 ohms per volt. The current to operate the meter movement is supplied by the circuit under test, and the smaller the current (higher the sensitivity) the better, particularly when measuring volts.

The loading effect of a voltmeter is determined by calculating the equivalent resistance that meter represents for the particular voltage range it is set to. A meter with a sensitivity of 10,000 ohms per volt will represent a resistance value of 100,000 (or 100k ohms) when set to its 10V range. Ohm's law can then be used to calculate the effect on a circuit when a 100k ohm resistor is connected where the voltmeter is being used to measure a voltage.

Digital voltmeters (DVM) also take current from the circuit under test, but they represent a fixed value of resistance (usually 11M ohm) regardless of the selected voltage range. The following lists typical sensitivities for voltmeters.

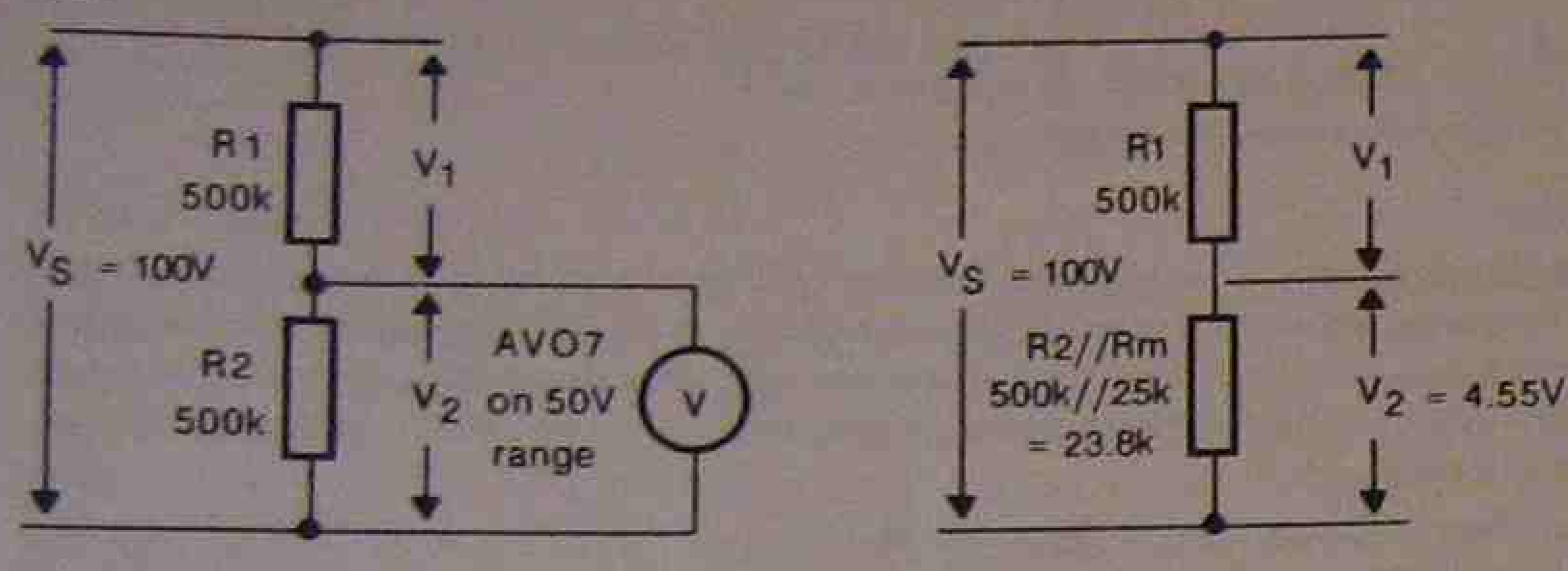
AVO7	=500 ohm/volt	(FSD current = 2mA)	On 10V range, equals a 5k resistor
AVO8	=20k ohm/volt	(FSD current = 50uA)	On 10V range, equals a 200k resistor
Arix	=100k ohm/volt	(FSD current = 10uA)	On 10V range, equals a 1M resistor
Digital	=11M ohm	(fixed resistance)	

Note that an Arix meter on its 100V range is the equivalent to a 10M ohm resistor, which compares favourably to the resistance of a digital voltmeter. When set to its 250V range, the equivalent resistance is 25M ohms, which exceeds that of a DVM.

7. **Circuit Loading**

Connecting a meter in a circuit effectively adds a resistor in parallel with the circuit under test. If the meter has a low internal resistance (low sensitivity) it can substantially reduce the voltage it is reading. This is referred to as voltmeter *loading*.

In the circuit of Fig.5(a), an AVO7 on its 50V range is connected to the circuit as shown. The AVO will have an equivalent resistance of 500 ohms/volt x 50V, which equals 25k ohms. Before the meter is connected the voltage across R2 will be 50V, or half the supply voltage as R1 equals R2.



(a) measuring voltage in a resistive circuit (b) effect caused by meter loading

Fig.5: *Meter loading*

When the meter is connected, the combined resistance of the meter and R2 equals 23.8k as shown in Fig.5(b). Using the potential divider equation, a voltage of 4.55 volts is now present across R2, and is the value that will be shown by the voltmeter. Using a meter with a much higher sensitivity is essential in a circuit that contains high value resistors, and even a DVM will cause some loading, effecting the voltage across R2. As a general rule, the meter resistance should be at least 10 times greater than the equivalent resistance of the circuit under test.

Ammeters also affect the current value being measured as an ammeter is effectively a resistance in series with the circuit under test. An ammeter contains a low value resistor, called a **shunt** that is connected in parallel with the meter movement in an analog meter or in parallel with the analog to digital converter (ADC) in a digital multimeter (DMM). The value of the shunt resistor depends on the FSD current of the meter movement (or ADC circuit in a DMM) and the lower the FSD current requirement the lower the value of the shunt resistor. A shunt is part of a current divider, through which all current other than that required by the meter movement passes. A shunt therefore has a finite value of resistance, though it is often as low as a few milliohms.

8. **Accuracy and resolution**

The accuracy of a meter is a function of its manufacture, and depends on the stability and quality of its components. Accuracy should not be confused with sensitivity or resolution. A 0.5% specification for accuracy, with a resolution of 1% is better than a meter with an accuracy of 5% but with a resolution of 0.004%. Digital meters have a much better resolution than analog meters, but may not always have the highest accuracy. Sensitivity is not a guarantee of high accuracy, although the higher the sensitivity the less the loading effect. The reading given by a high sensitivity meter will therefore be more likely to be correct, limited only by the accuracy and resolution of the meter.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 1

1. For the circuit of Fig.1, calculate the resistance between points A and B.

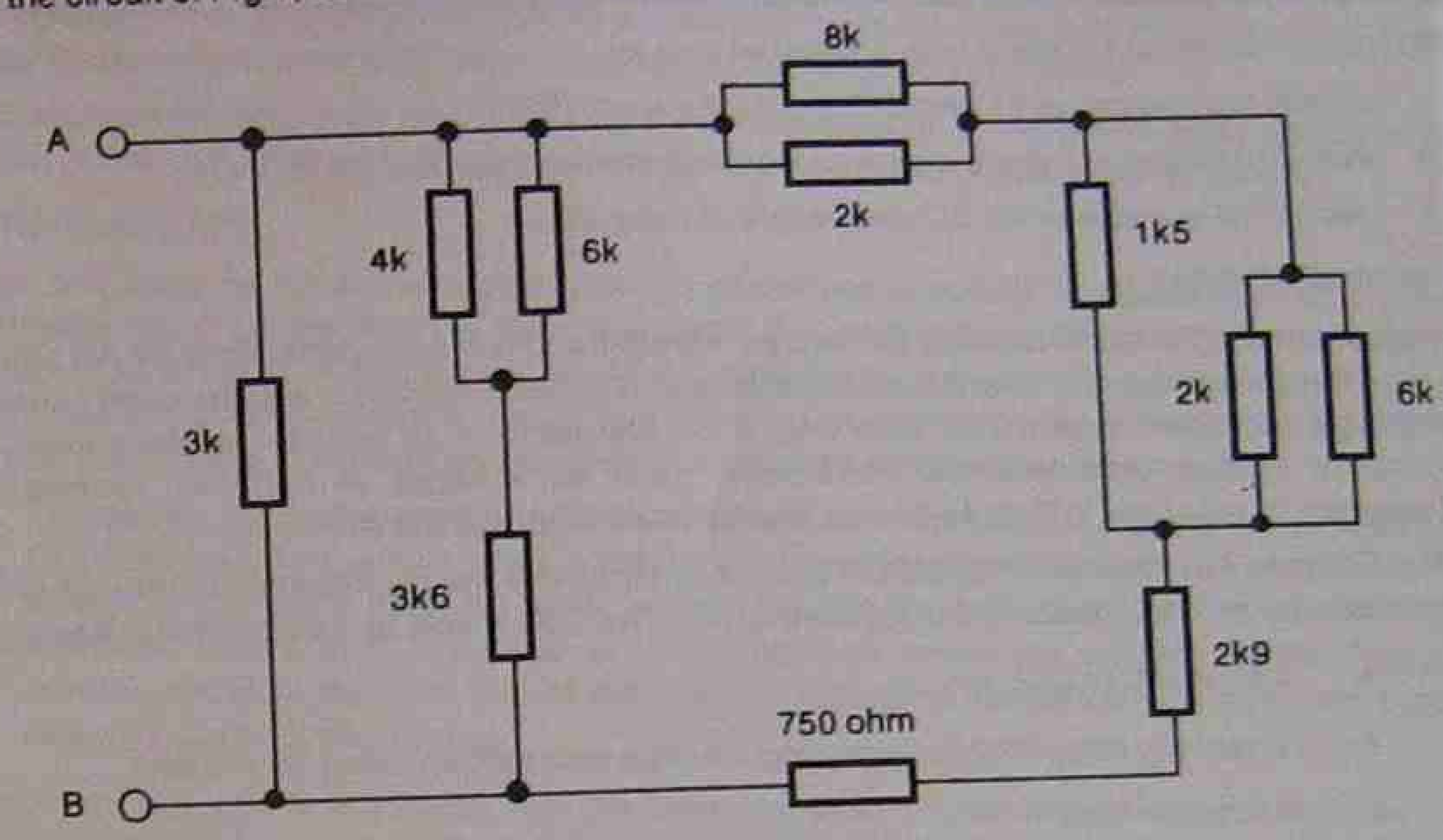


Fig.1

2. For the circuit of Fig.2, calculate the values of V1 and V2.

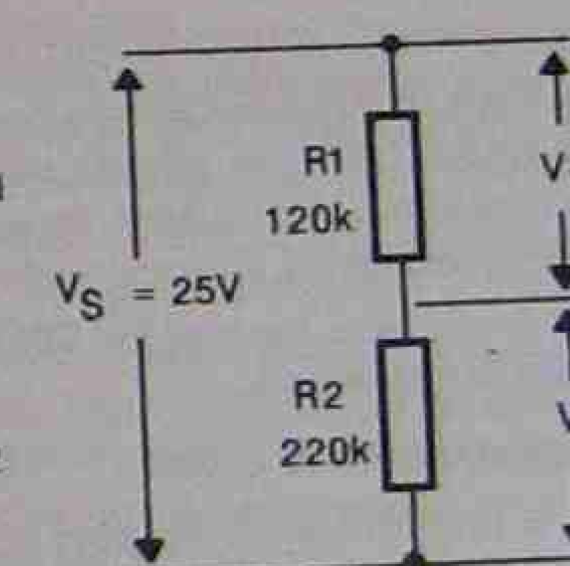


Fig.2

3. For the circuit of Fig.2, calculate the reading that will be shown by a voltmeter with a sensitivity of 10k ohms/volt set to its 10V scale.

4. Determine the values of the resistors shown in Fig.3.

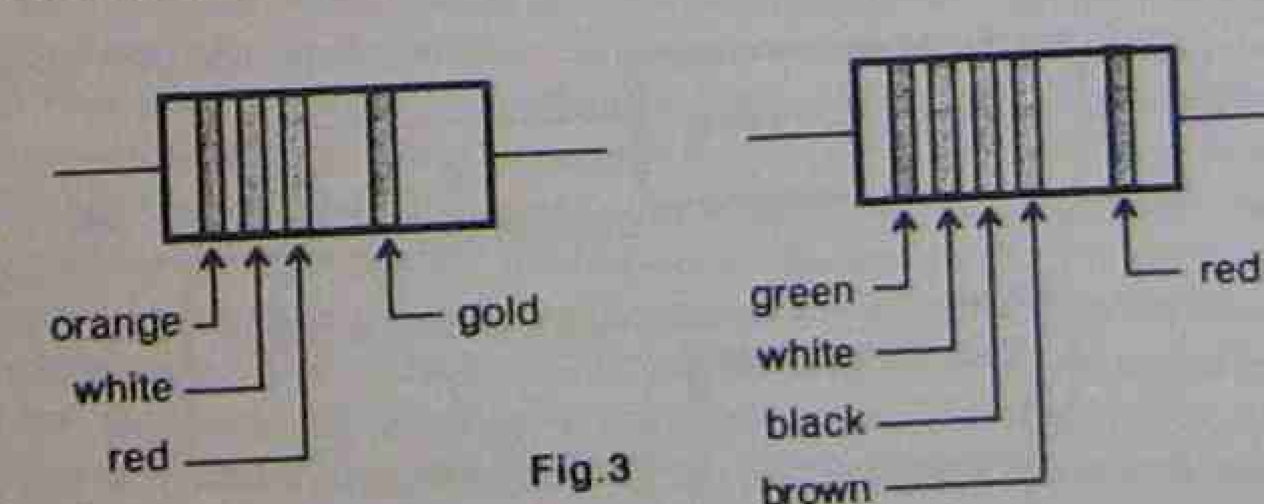


Fig.3

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 2

THE CATHODE RAY OSCILLOSCOPE

OBJECTIVES: At the end of this lesson you will be able to:

- Identify the controls of a Cathode Ray Oscilloscope. (CRO)
- Measure voltage, current, frequency and phase shift using a dual trace CRO.
- Use a CRO to measure the DC component of a signal.

1. INTRODUCTION

A multimeter is limited in its ability to measure voltages and currents, particularly for AC signals. The frequency range of a typical multimeter is limited to less than 1kHz, and the scale is usually calibrated for RMS values, and then only if the waveform is sinusoidal. In electronics it is common to have non-sinusoidal waveforms, either as voltages or currents. As well, the frequency of the signal is likely to be over the 1kHz limit of a multimeter.

The Cathode Ray Oscilloscope, (CRO) is a more expensive measuring instrument and is less portable, but in most situations it is far more useful. The CRO is able to measure the following:

- voltage (AC or DC)
- current (by measuring voltage drop across a resistorthen using Ohm's law)
- period
- frequency (by calculation from the period measurement)
- phase difference (between two waveforms)

These values can be measured for any type of waveform at frequencies only limited by the CRO itself and a typical CRO can provide useful measurements for signals up to 20MHz. Some instruments extend beyond 100MHz. Because the signal is actually displayed, the shape of the waveform can be examined, allowing more accurate analysis of the circuit behaviour.

These notes briefly describe how the CRO operates and how it is used to measure the various characteristics of a waveform.

2. BLOCK DIAGRAM OF THE CRO

Fig.1 shows a simplified block diagram of a CRO. The basic sections are the CRO tube, the power supply, the vertical (or Y) amplifier and the time base (X). Most CROs are dual trace, meaning they can display two signals simultaneously. However, for the purposes of explanation, a single trace CRO is assumed.

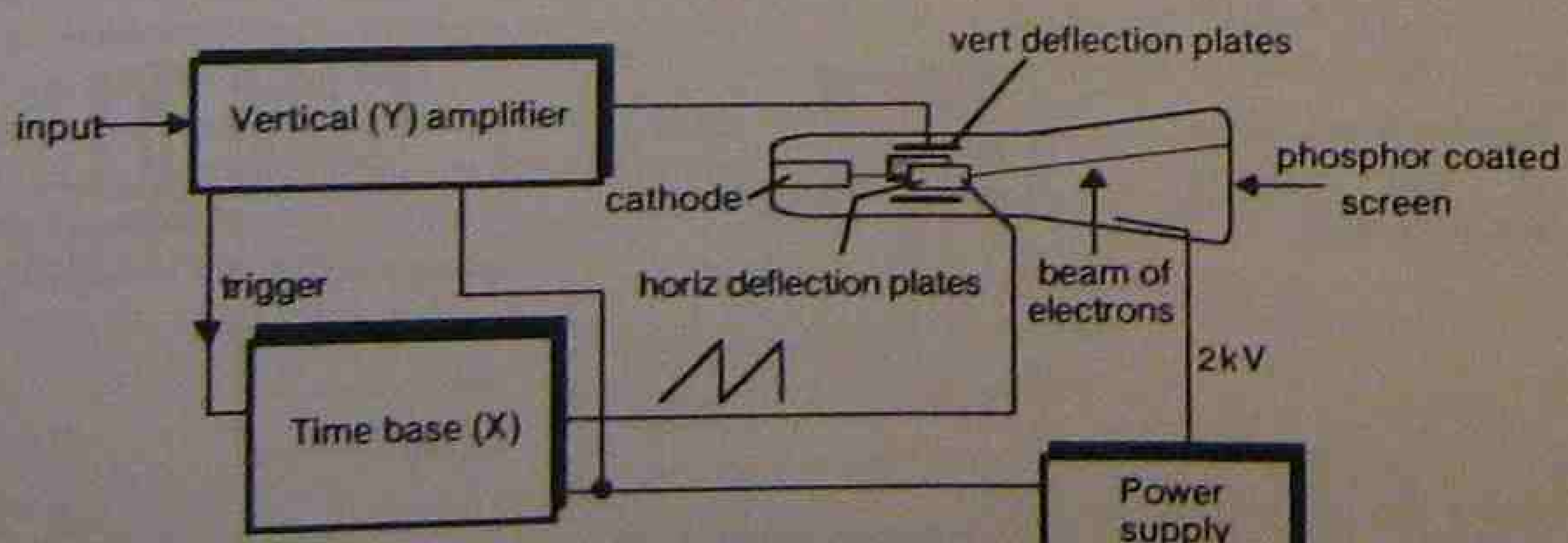


Fig.1: Block diagram of a CRO

3. THE CATHODE RAY TUBE (CRT)

The screen of an oscilloscope is at the anode end of a CRT. The CRT is a type of thermionic valve and has a coating of phosphor on the inside of the anode, which glows when struck by an electron beam. The electron beam originates from the cathode which is heated by a filament so that it emits electrons. The beam is attracted to the anode by a high positive DC potential (2kV or so) relative to the cathode. Deflection plates are placed inside the tube so that the beam can be deflected sideways (X direction) or vertically (Y direction). Deflection is caused when one plate has a different potential to the other (electrostatic deflection), unlike a TV set which uses magnetic deflection. Other electrodes are used to focus the beam and to accelerate it. These electrodes usually have a DC potential of several hundred volts to over 1kV.

To allow waveforms to be measured, a graticule consisting of a 1cm grid is either etched on the front of the tube, or drawn on a transparent panel placed at the viewing end of the tube.

4. THE TIME BASE

The time base, or horizontal section consists of a sweep generator (oscillator) and a horizontal amplifier which connects to the deflection plates that control the sideways (X) movement of the beam. The beam is moved across the screen from left to right at a preselected speed, then is returned to the left hand side (retrace) very quickly. During retrace, the beam is turned off so the retrace is invisible. The required waveform is therefore a sawtooth, as shown in Fig.1.

- The sweep generator provides the time base control for viewing signals, by producing a sawtooth waveform which controls the horizontal movement of the beam on the screen. The sawtooth waveform must be linear so that the scan rate is constant. The period of the sawtooth waveform is adjustable in calibrated steps which are selected by the Time/Div control. Because the scan rate of the beam is calibrated the period of a waveform can be directly read from the screen.

An important control associated with the sweep generator is the **trigger** control. The idea is to synchronise the sweep generator with the input signal. Typically a trigger pulse is generated by the input signal when the signal passes through zero volts or through some preset voltage value. If the trigger control is set to accept only the positive trigger pulses, the sweep generator will produce one cycle of the sawtooth waveform whenever a positive trigger pulse occurs. This causes the waveform to appear stationary on the face of the CRT. To allow a trace to appear on the screen in the absence of an input signal, another control is used to turn off the trigger input allowing the sweep generator to 'free run'. On some CROs, this control is labelled 'AUTO', and if it is not selected there will be no trace on the screen, regardless of any other settings.

- The horizontal amplifier drives the deflection plates by amplifying the output of the sweep generator to a level suitable for deflecting the beam. Most CROs have a facility to connect an external signal to the horizontal amplifier instead of the output of the sweep generator. This allows **Lissajous** patterns to be displayed on the screen, in which circles, the ABC logo and other interesting patterns are produced by feeding two sinewaves (or other waveshapes) to the CRO, one to the horizontal (X) amplifier and the other to the vertical (Y) amplifier.

5. THE VERTICAL SECTION

The vertical section (Y) accepts the input signal and after suitable amplification, applies it to the vertical deflection plates. The beam will therefore move up and down at a speed depending on the instantaneous potential difference between the deflection plates. To create the potential difference between the plates, one plate is fed with a signal that is 180 degrees out of phase with the other. In the absence of any input signal, (assuming auto trigger), the beam will only be deflected in the horizontal direction, giving a straight line display on the screen. If a DC signal is applied, the beam will be deflected vertically, and will appear as a straight line, but moved from its previous position, depending on the value of the input DC voltage. If the input signal contains an AC and a DC component, the trace will show as a waveform moved from the zero, or reference line. Thus, the shape of a waveform is produced by the beam being moved vertically by the Y amplifier and horizontally by the X amplifier.

Fig.2 shows a simplified circuit of the input switching for the signal prior to its application to the vertical (Y) amplifier. It contains a calibrated attenuator (SW2) and a non-calibrated attenuator (VR1) that are both used to reduce the level of the signal applied to the vertical amplifier. This is necessary to keep the display on the screen from exceeding the available display height. The calibrated attenuator is a switch that selects the volts/division and consists of a number of series connected resistors that form a potential divider. For example, if the setting is 10V/division on the graticule, a waveform that extends for three divisions has a peak to peak voltage of 30V.

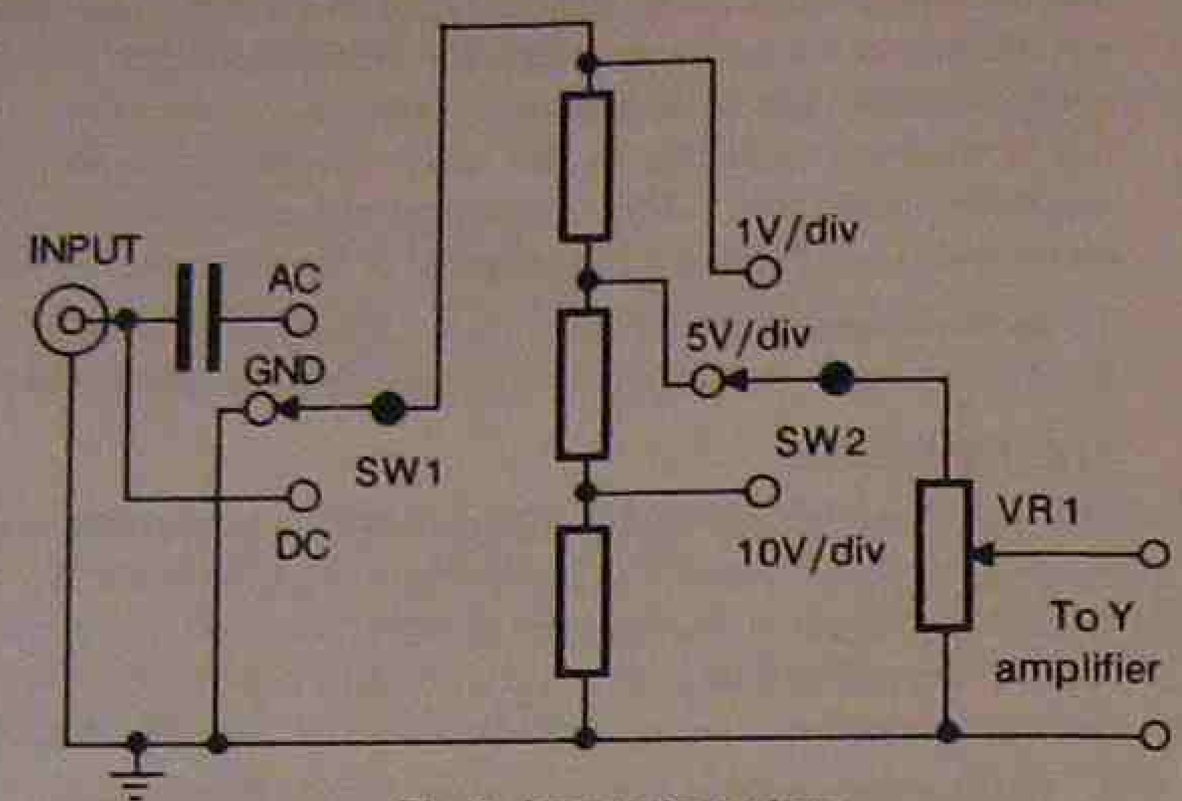


Fig.2: Input attenuators

The potentiometer VR1 can further reduce the amplitude of the signal but if is used, the settings of the main attenuator are no longer calibrated and the value of the input voltage cannot be read from the screen.

Other vertical input controls include the vertical position control, and a three pole input coupling switch (SW1). The three positions of this switch are:

- AC: this position connects the input signal via a coupling capacitor and allows the AC component of a waveform to be viewed while blocking the DC component. For example, a small AC ripple on top of a large DC voltage can be viewed.
- GND: in this position a zero reference voltage (ground) is applied to the input of the vertical amplifier so that it can be used as a reference point for measurements. The beam is usually positioned centrally on the screen with the Y shift control.
- DC: this position is the direct coupled mode and both the AC and the DC component of a waveform are passed to the attenuators. When measuring a DC voltage, this switch setting is required.

6. SIGNAL WAVEFORMS

The CRO is used to observe the shape of a waveform, to measure its peak to peak voltage and its period. If the CRO has two beams, it can also be used to measure the phase difference between two signals. There are many types of waveforms in electronics, in which the sinewave is the most basic. The relationships between the various voltage values for a sine wave are shown below.

> $V_{max} = V_{p\text{-}p}/2$ (Vmax is the voltage from 0V to the maximum of the waveform)
>
> $V_{RMS} = 0.71 V_{max}$
>
> $V_{average} = V_{DC} = 0V$ (for whole cycle)

The period of any waveform is the time between two identical points of the waveform. Usually, the period is measured between two adjacent positive or two negative going zero crossing points, but it can be measured between the positive peaks, the negative peaks and so on.

Fig.3 shows how a CRO can measure the peak to peak voltage and the period of a waveform.

Vp-p = **8V** as the V/div setting = 2V/div and the height of the wave is 4 divisions.

The period is **20mS** as the time/div setting is 5mS, and a cycle covers 4 divisions.

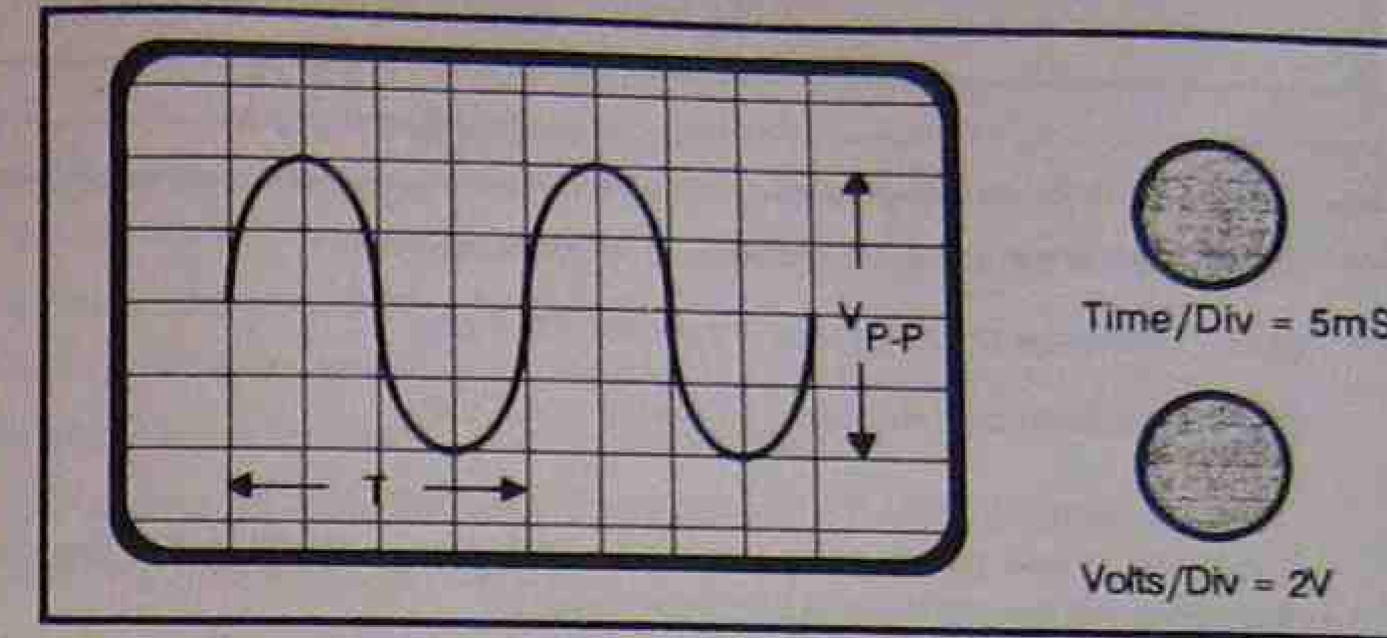


Fig.3: *Measuring period and peak to peak voltage on a CRO*

The frequency of any waveform is determined by dividing the period of the waveform into 1. That is, **frequency equals the reciprocal of period**. For Fig.3, the frequency is the reciprocal of 20ms which equals 50Hz

To determine the RMS value of the waveform shown in Fig.3, the maximum voltage is either measured or derived from the peak to peak value. In Fig.3, V_{max} = 4V and the RMS voltage therefore equals 0.71 x 4 which gives 2.84V. Because the waveform is symmetrical around 0V, the average, or DC value is 0V. If the waveform was superimposed on a DC voltage, the display would have been shifted up (for positive voltages) or down (for negative voltages), indicating that a DC component was present. This would give a waveform that was no longer symmetrical around the zero volts line, and the average value of the waveform would equal the DC component present in the signal.

For waveforms other than sinewaves, the average and the RMS voltages need to be determined mathematically in which:

- the average value is determined by summing a fixed range of instantaneous voltage values over one cycle with an equal number of values from both the positive and the negative half cycles. The average value equals the total of all the instantaneous values, divided by the number of values taken.
- RMS values are calculated over one cycle by adding the squares of the amplitude per time unit, dividing this sum by the number of time units, and taking the square root. (RMS = Root Mean Squared).

It is unusual to have to calculate these values for non-sinusoidal waveforms, and for the purposes of this subject, sine waves will be assumed.

7. WAVEFORMS WITH A DC COMPONENT

In electronics it is common to have an AC waveform superimposed on a DC voltage. In Fig.4, an AC signal is coupled through a capacitor to a potential divider connected to 20V DC. Because both resistor values are equal, a 10V DC potential will be added to the AC input signal. The resulting waveform is the AC signal with a 10V DC component. The AC waveform is now symmetrical around the 10V line rather than the 0V line. On a CRO, the waveform would be shifted vertically on the screen, unless AC coupling was selected.

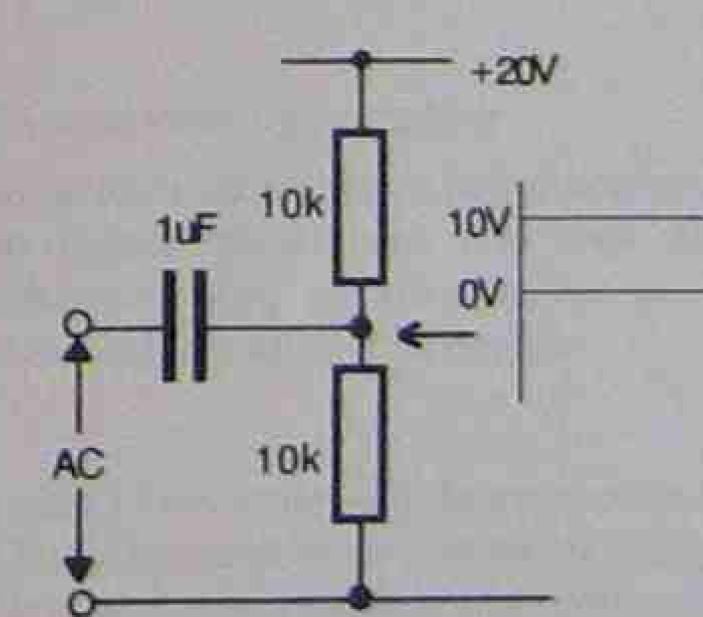


Fig.4: *AC combined with DC*

****

THEORY ASSIGNMENT 2

1. Calculate the frequencies of the following waveforms:

   (a) Timebase on 20 μs/cm 1 cycle = 2.5 cm

   (b) Timebase on 1ms/cm 1 cycle = 4.1 cm

2. A 2.5kHz, 2 volt p-p square wave is applied to a CRO. If the timebase is set to 100μs/cm, determine the length (in cms) of one cycle.

3. Determine the following from the display shown in Fig.1:

   (a) Peak to peak voltage for both waveforms if the volts/division switch is set to:

   i) 5V/div

   ii) 200mV/div

   iii) 40V/div

   (b) Calculate the phase difference between the two waveforms and indicate which is lagging.

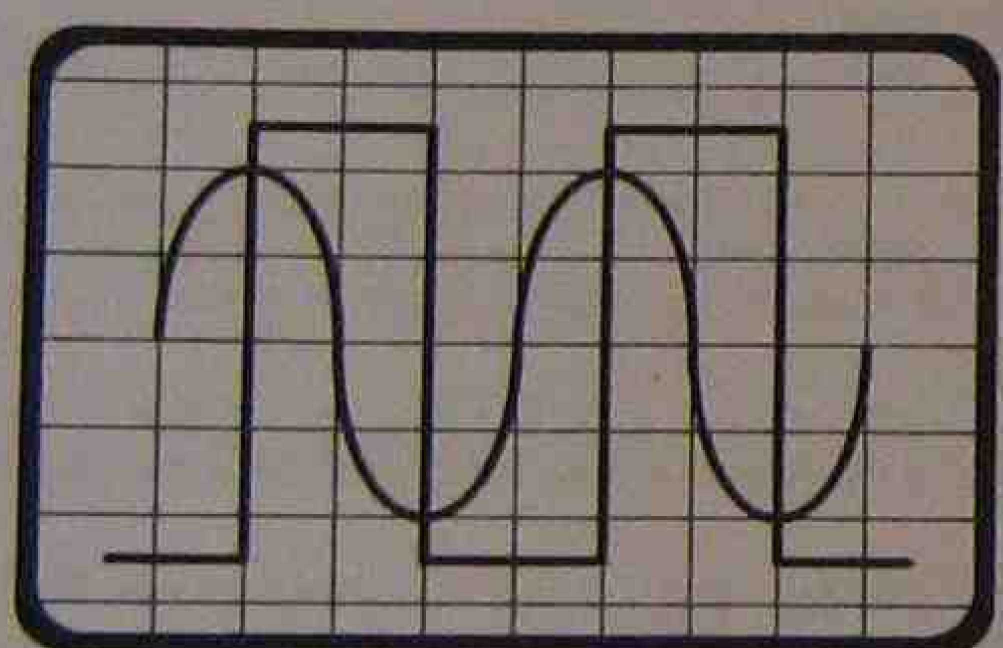


Fig.1

****

THEORY LESSON 3

PRINCIPLES OF AMPLIFICATION AND TRANSISTOR CHARACTERISTICS

OBJECTIVES At the end of this lesson you should be able to:

- Sketch the equivalent circuit of an amplifier that shows its operating characteristics.
- Define the term amplification as applied to electronic amplifiers.
- Calculate the voltage gain for an amplifier given the output and input voltages.
- List the basic characteristics of a bipolar transistor.

1. INTRODUCTION

An amplifier is the basic building block of analog electronics. Its function is to amplify an electrical quantity, such as voltage, current or power. All amplifiers have certain characteristics and these need to be known if an amplifier is to be used in a particular application. These notes describe the basic characteristics of an amplifier and their effect in a circuit. The equivalent circuit of an amplifier is also described.

The transistor is the basic amplifying device and these notes also briefly describe some of the operating characteristics of the transistor.

2. AMPLIFICATION

The term amplification in electronics is used to describe the action of a circuit that produces an output signal larger than the input signal. An ideal amplifier will produce an output signal that has exactly the same shape as the input, except it will be larger. The term amplification is derived from the term amplitude, which refers to the height of a waveform. Most amplifiers accept a voltage as the input signal, but the electrical quantities of power and current can also be amplified. Amplifiers are not necessarily confined to electronics, and other types of amplifiers include the magnetic amplifier, hydraulic, pneumatic and mechanical types.

Fig.1 shows the general form of an amplifier, in which the input signal is used to control a power source. The output signal is therefore produced by the power source, and the amplifier section has the task of controlling the power source. In an ideal amplifier the power or energy source has an infinite capacity.

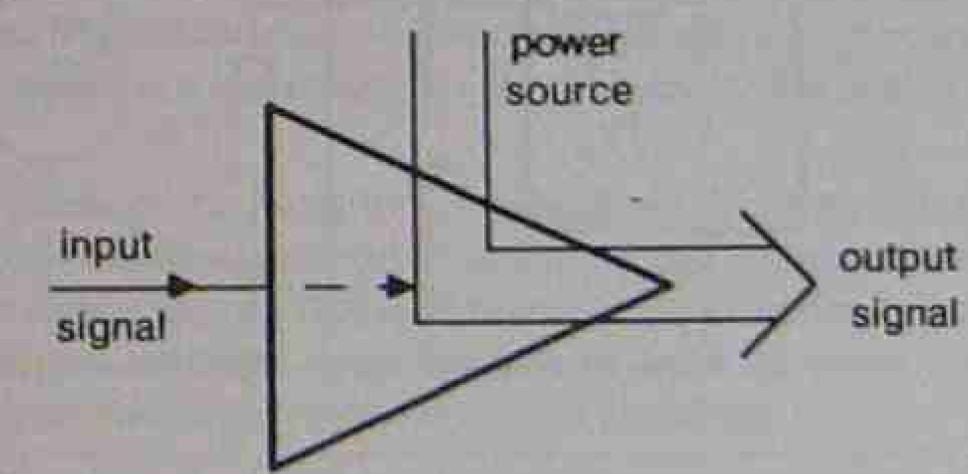


Fig.1: General form of an amplifier

An amplifier is typically used to amplify the electrical signal from some type of signal source and to drive an output load. These devices are often called transducers and their basic characteristics need to be explained so that the characteristics of an amplifier can be understood. Transducers are more fully described in other subjects within this course.

3. INPUT AND OUTPUT TRANSDUCERS

A transducer is a device that converts from one energy form to another, and in electronics one of the energy forms will be an electrical quantity. An input transducer produces an electrical signal as a result of heat, mechanical movement, chemical action, magnetism and other energy forms. Typical input signal sources are tacho generators, thermocouples, tape recorder heads, strain gauges, record player pickups, pH cells, Hall effect devices and so on. An output transducer converts an electrical signal into another form of energy. Typical output transducers are motors, a loudspeaker, optical cable drivers and so on.

Fig.2 shows the equivalent circuit of an input transducer, also called a signal source, in which an internal voltage source connects to the output terminal of the device through a series resistor Rs. The important aspect of Fig.2 is the value of the source resistance Rs.

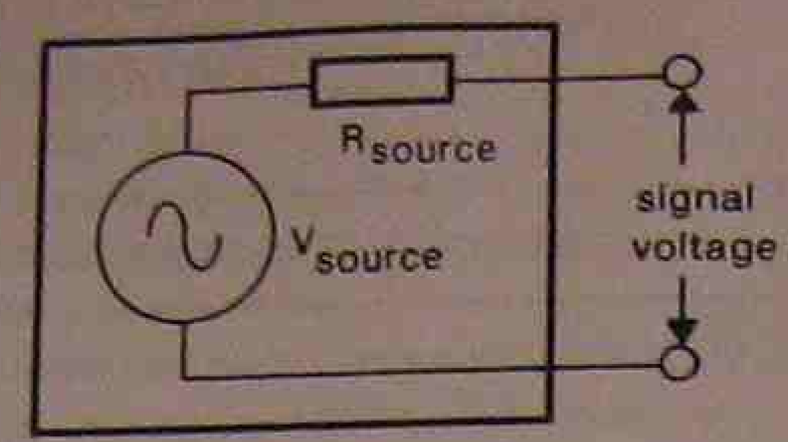


Fig.2: Equivalent circuit of a signal source

If the signal source is connected to a CRO, the signal that will be displayed should equal the voltage being produced by the internal voltage source of the device. However, if a resistance equal to the source resistance of the device is connected across its output terminals, the voltage displayed on the CRO will drop to half its previous reading. This is similar to the loading effect described in the notes for week 1. Therefore, if a signal source is connected to an amplifier, it is important that the amplifier itself doesn't load the signal source.

An output transducer is the load for the amplifier, and the important characteristic to be considered is the value of resistance the load represents. In most cases, an output transducer will have a low resistance. Motors or loudspeakers all have an impedance measuring a few ohms, and these devices need to be driven by a power amplifier. A voltage amplifier cannot supply power, and their output load needs to have a much higher resistance.

4. AMPLIFIER CHARACTERISTICS

As for any electrical circuit, an amplifier has a number of operating characteristics which collectively define how the amplifier can be used.

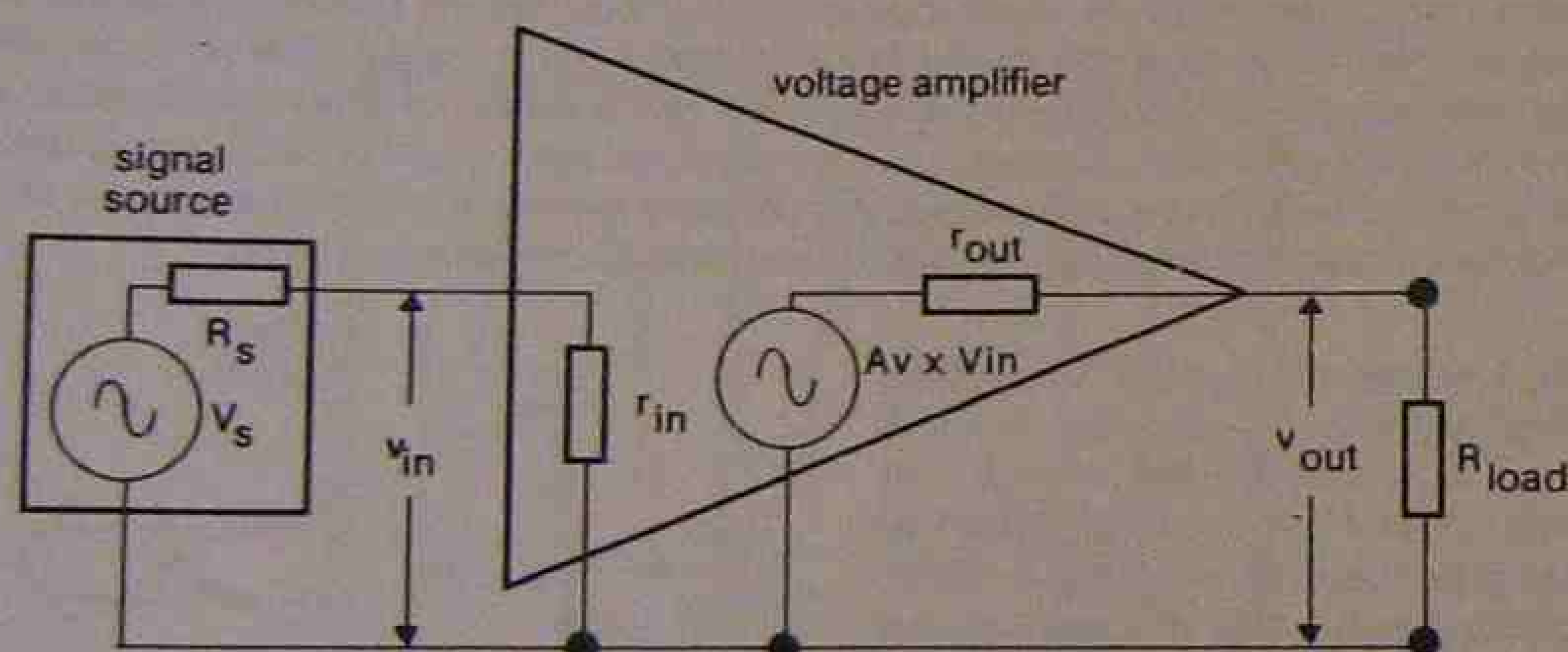


Fig.3: Equivalent circuit of a voltage amplifier

Fig.3 shows the equivalent circuit of a voltage amplifier connected to a signal source and a load. Note that this diagram can be modified to describe a power amplifier or a current amplifier.

The most important characteristic of an amplifier is its gain which is defined as the **ratio of its output to its input**. Thus:

- Voltage gain (Av) is the ratio of the output voltage of an amplifier to its input voltage. In Fig.3, the voltage gain is shown by the internal voltage generator labelled Av x vin. This generator represents that part of the amplifier circuit causing the input signal to be amplified.
- Current gain (Ai) is the ratio of the output current and the input signal current to the amplifier.
- Power gain (Ap) is the ratio of the output power of the amplifier to the power applied to its input.

☐ $Av = \frac{Vo}{Vin}$ $Ai = \frac{Io}{Iin}$ $Ap = \frac{Po}{Pin}$

The next two characteristics are the input impedance and the output impedance of the amplifier. The term impedance is used to describe any capacitance or inductance present with the resistance value. In these notes, the only electrical quantity that will be considered is the resistance, and the terms input resistance and output resistance will be used. However, a practical amplifier will have capacitance and inductance which can affect its performance at particular frequencies.

- Input resistance (rin) is the resistance present between the input terminals of an amplifier. It cannot be measured with an ohmmeter, as the resistance comprises the effects of all the components within the amplifier. Instead it has to be determined by measuring the input signal voltage and the input signal current, then calculated with Ohm's law. That is:

☐ $r_{in} = \frac{Vin}{Iin}$

The input resistance of an ideal amplifier is infinity, or an open circuit. Practical amplifiers have input resistance values ranging from a few ohms to hundreds of megohms. A typical transistor amplifier has an input resistance of several thousand ohms.

Input resistance is important as it determines the type of signal source that can be connected to the amplifier. If the amplifier has a low input resistance, the signal source connected to it needs to have a correspondingly low source resistance. An amplifier with a high input resistance can be connected to any type of signal source, as the amplifier will not load the source.

- Output resistance (ro) is the resistance value between the internal voltage source of the amplifier and its output terminal. The ideal value of output resistance is zero, and a practical amplifier will have an output resistance anywhere from a few ohms to several thousand ohms.

In fact, an amplifier can be seen as a form of signal source itself, although it needs an electrical input signal to operate. The value of the output resistance determines the type of load that can be connected to the amplifier. If the output resistance is low, then a low resistance load can be connected, assuming the amplifier is able to produce the required load current. A voltage amplifier with a low output resistance cannot drive a low resistance load as it is generally unable to supply the high value of current required without overheating. An amplifier with a high output resistance can only drive high resistance loads, as a low resistance load will cause the output of the amplifier to be reduced.

Like input resistance, output resistance needs to be determined experimentally. This can be achieved in a number of ways, and a commonly used method is to first measure the unloaded output voltage of the amplifier. A variable resistor is then connected across the output terminals and its value adjusted until the output of the amplifier drops to half its previous value. The resistance of the potentiometer will now equal the output resistance of the amplifier.

Summary of input and output resistance

☐ Input resistance (rin):	ideal	=	infinity,	practical:	>10 x Rs
☐ Output resistance (ro):	ideal	=	zero,	practical:	$< \frac{ro}{10}$

5. THE TRANSISTOR

These notes assume students are already familiar with the basic operation of a transistor. For further details, refer to a recommended text. The following summarises some aspects of the operation of a transistor.

A transistor contains two PN junctions and has three terminals labelled base, collector and emitter. This type of transistor is referred to as a bipolar junction transistor (BJT) and other transistors include the field effect type (FET). For the purposes of these notes, the term transistor refers to the BJT.

Fig.4 shows the basic construction, the diode equivalent circuit and the schematic symbol for the NPN and the PNP transistor. Note that the arrow on the emitter terminal points in the direction of the current flow.

A transistor is a current amplifier, in which the base-emitter current controls the value of the collector-emitter current. To produce the base-emitter current, a voltage equal to the barrier potential of the PN junction (0.6V for silicon, 0.2V for germanium) is required.

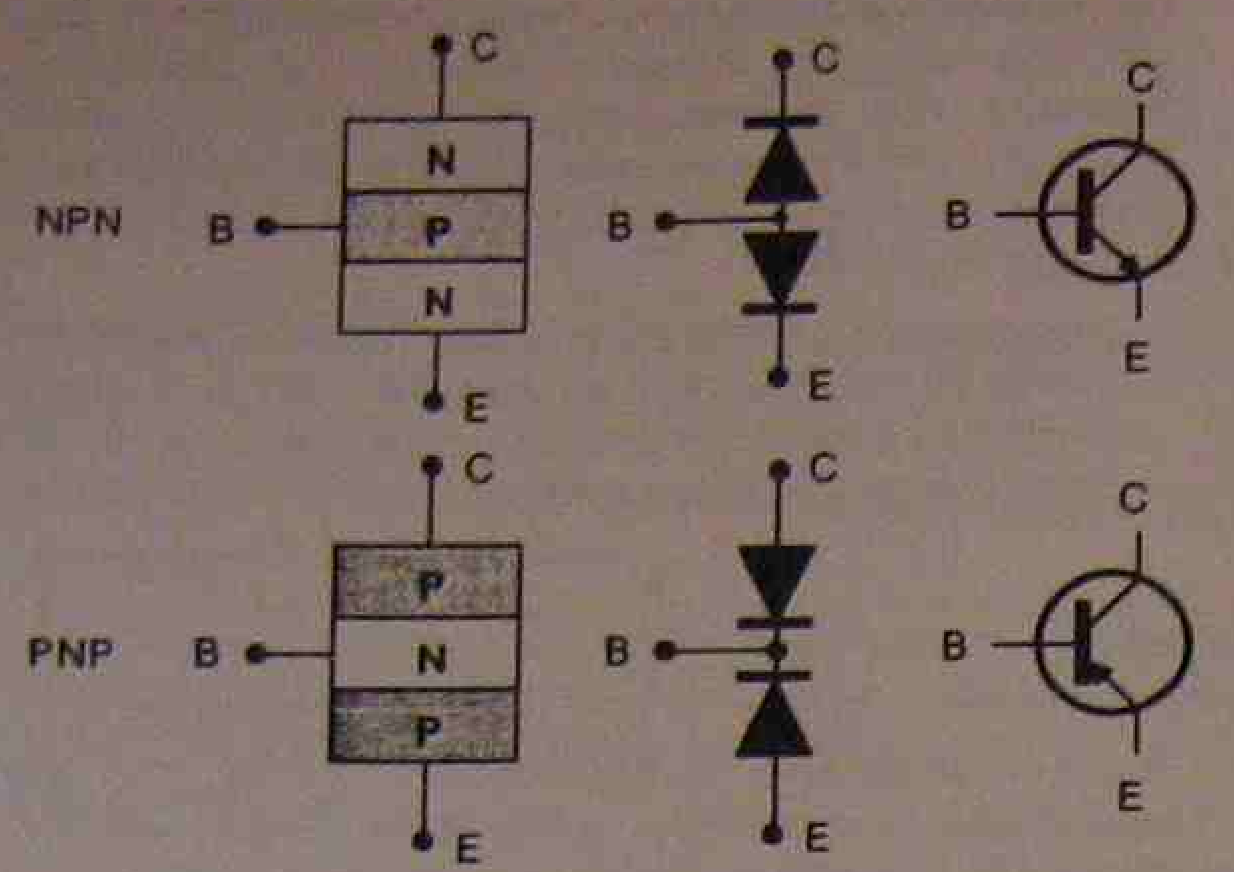


Fig.4 (a) construction (b) diode equivalent (c) schematic symbol

Temp Coff. -2-5mV/°C

The voltage required to overcome the barrier potential is called Vbe and its value depends on the amount of base current required. Like all PN junctions, the barrier voltage of the base-emitter junction (Vbe) for a transistor drops by approximately 2.5mV/°C as the temperature rises. Refer to Fig.5 which shows this as a graph. Thus if the value of Vbe is held at 0.6V and the temperature rises, the base current will increase, as the internal barrier voltage has now dropped. An increase in base current will cause a corresponding increase in the collector current.

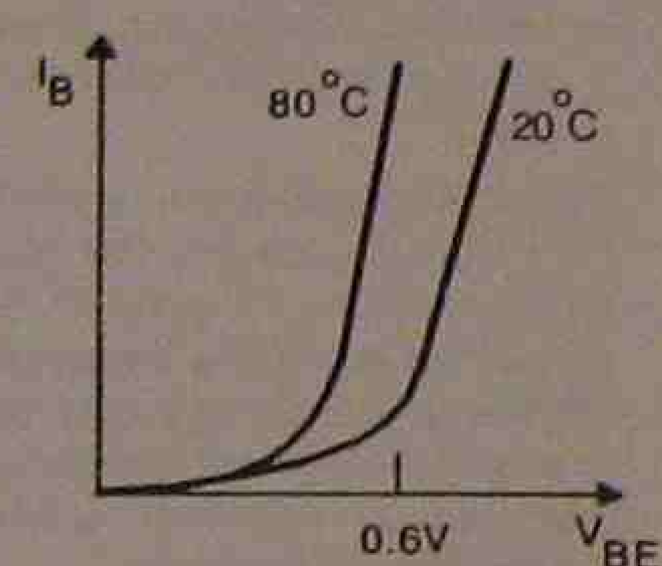


Fig.5: Effects of temperature on I_B.

6. TRANSISTOR CHARACTERISTICS

There are thousands of transistor types in use and they all have particular characteristics. The following summarises some of the more important of these.

- Current gain (hfe) or ß: This is the ratio of the collector current and the base current. That is:

$$\beta = \frac{I_C}{I_B}$$

Current gain of a transistor can either refer to its DC operating conditions or to the AC conditions. Most manufacturers give a value of current gain that refers to the DC conditions, and denote it with the term Hfe (upper case H refers to DC conditions). DC current gain is also often called the beta (ß) of the transistor. The AC current gain is denoted by the term hfe. For the purposes of this subject, the AC current gain and the DC current gain are assumed to be equal.

DC current gain of a transistor is generally given for a specified value of collector current. Because of manufacturing limitations, it is usual for manufacturers to quote a range of current gain values for each type of transistor. For example, the BC547 type has a guaranteed current gain of from 110 to 800 at a collector current of 2mA. Current gain will also vary with temperature in which the current gain increases with temperature.

- Maximum value of collector current (I_{Cmax}) is the highest value of collector current a transistor can pass without damage, and ranges from 100mA for a typical small signal transistor to 20A or more for the larger types.

- **Voltage ratings**
  - Collector emitter voltage, V_{CE}: This is the most important voltage rating for a transistor and is the maximum value of collector-emitter voltage the transistor can withstand before breakdown occurs. This value is generally specified for the condition of base open circuit, denoted by the term V_{CEO}. For an NPN transistor, the polarity of the maximum voltage will be positive at the collector, negative at the emitter. For a PNP transistor, the polarities are opposite. Note that the maximum reverse voltage a small signal transistor can withstand before conduction occurs is much lower than V_{CE}, generally around 5 to 7V. Transistors with a V_{CE} rating of several thousand volts are available, but most transistors are rated for voltages between 40V to 100V.
  - Collector-emitter saturation voltage V_{CES}: Saturation occurs when the collector-emitter voltage has fallen to its lowest value as a result of an increase in the collector current. It is generally around 0.2V to 0.7V.
  - Base-emitter voltage Vbe: As already described, Vbe has a nominal value of 0.6V for a silicon transistor and 0.2V for a germanium type. For an NPN transistor, the base is positive with respect to the emitter, and a PNP transistor has its base negative compared to the emitter. The maximum reverse voltage a small signal transistor can withstand is generally around 5V.

- **Case outline and power rating**
  - The case outline refers to the type of package used to contain the transistor element. Many of these are standardised and are used by all transistor manufacturers, while some are peculiar to a particular manufacturer. The standard outlines are give a TO (transistor outline) number and the type of package used depends on the amount of power dissipation the transistor is rated for. Small signal transistors are usually packaged in either the TO-18 or TO-92 styles. The TO-92 outline has several variations in which the pin connections differ depending on which variation of the outline is used. There are numerous package styles for power transistors including the TO-3 type (used by the 2N3055 transistor), the TO-220 and others. These package styles allow a heatsink to be added to help dissipate the heat generated by the transistor.
  - The power rating of a transistor determines the maximum collector current that can be used for that transistor for a given collector-emitter voltage. As power equals the product of voltage and current, a transistor can never be operated at both its maximum current and voltage ratings. As well, the power dissipation rating is generally stated for a specified case temperature, and a heat sink is usually needed to realise the maximum power rating. Power ratings vary from 300mW for a small signal transistor to several hundred watts for the larger types.

****

BIPOLAR TRANSISTORS

TYPE	CASE	POL MAT	Vce	Vcb	IC mA	Vcesat @	IC	Hfe @	IC mA	Ft @ MHz	IC mA	Ptot mW	USE	COMPARABLE TYPES
AC126	TO-1	PG	12	32	100			140	2	1.7	10	500	Audio driver	2N406
AC127	TO-1	NG	18	32	500			105	50	1.5	10	340	Audio O/P	AC187
AC128	TO-1	PG	18	32	1000	[illegible]	1A	80-175	300	1	10	1W	Audio O/P	AC186
AC132	TO-1	PG	12	32	200	0.35	200	115	50	1.3	10	215	Audio O/P	AC188
AC187	TO-1	NG	15	25	1000	0.8	1A	100-500	300	1	10	1W	Audio O/P	AC127
AC188	TO-1	PG	15	25	1000	0.8	1A	100-500	300	1	10	220	Audio O/P	AC128
AD161	PT1	NG	20	32	3000	0.8	1A	80-320	500	0.02	300	4W	Audio Amp.	AD165, 2N1218, 2N1292
AD162	PT1	PG	20	32	3000	0.4	1A	80-320	500	0.015	300	6W	Audio Amp.	AD143, AD152, AD147
AF118	TO-7	PG	15	32	10			150	1	75	1	50	H.F. Amp.	AF135, AF136, 2N3127
AS215	TO-3	PG	80	100	10A	0.4	10A	20-55	1A	0.2	1A	30W	H.C. Sw	OC28
AS216	TO-3	PG	32	100	10A	0.4	10A	30-110	1A	0.22	1A	30W	H.C.Sw.	O.C.36
BC107	TO-18	NS	45	50	100	0.2	10	110-450	2	300	10	300	S.S. Amp.	BC207, BC147, BC182
BC108	TO-18	NS	20	30	100	0.2	10	110-800	2	300	10	300	S.S. Amp.	BC208, BC148, BC183
BC109	TO-18	NS	20	30	100	0.2	10	200-800	2	300	10	300	Low Noise S.S. Amp.	BC209, BC149, BC184
BC109C	TO-18	NS	20	30	100	0.2	10	200-800	2	300	10	300	Low Noise S.S. Amp.	BC209C, BC184C, BC149C
BC157	SOT-25	PS	45	50	100	0.25	10	75-260	2	150	10	300	S.S. Amp.	BC177, BC307, BC212
BC158	SOT-25	PS	25	30	100	0.25	10	75-500	2	150	10	300	S.S.Amp.	BC178, BC308, BC213
BC159	SOT-25	PS	20	25	100	0.25	10	125-500	2	150	10	300	S.S. Amp.	BC179, BC309, BC214
BC177	TO-18	PS	45	50	100	0.25	10	75-260	2	150	10	300	S.S. Amp.	BC157, BC307, BC212
BC178	TO-18	PS	25	30	100	0.25	10	75-500	2	150	10	300	S.S. Amp.	BC158, BC308, BC213
BC179	TO-18	PS	20	25	100	0.25	10	125-500	2	150	10	300	S.S. Amp.	BC159, BC309, BC124
BC182(L)	SOT-30 (TO-92/74)	NS	50	60	200	0.25	10	100-480	2	150	10	300	S.S. Amp.	BC107, BC207, BC147
BC183(L)	SOT-30 TO-92/74)	NS	30	45	200	0.25	10	100-850	2	150	10	300	S.S. Amp.	BC108, BC208, BC148
BC184(L)	SOT-30	NS	30	45	200	0.25	10	250-850	2	150	10	300	Low Noise High Gain	BC109, BC209, BC149
BC207	TO-106	NS	45	50	100	0.25	10	110-220	2	150	10	300	S.S. Amp.	BC107, BC182, BC147
BC208	TO-106	NS	20	25	100	0.25	10	110-800	2	150	10	200	S.S. Amp.	BC108, BC183, BC148
BC209	TO-106	NS	20	25	110	0.25	10	200-800	2	150	10	200	Low Noise High Gain	BC109, BC184, BC149
BC213(L)	SOT-30 (TO-92/74)	PS	30	45	200	25	10	80-400	2	200	10	300	S.S. Amp	BC308, BC158, BC178
BC214(L)	SOT-30 (TO-92/74)	PS	30	45	200	0.25	10	80-400	2	200	10	300	S.S. Amp	
BC327	TO-92 VAR 1	PS	45	50	800	0.7	500	100-600	100	100	10	800	O/P	2N3638
BC328	TO-92 VAR 1	PS	25	30	800	0.7	500	100-600	100	100	10	800	O/P	BC327, 2N3643
BC337	TO-92 VAR 1	NS	45	50	800	0.7	500	100-600	100	200	10	800	O/P	2N3642
BC338	TO-92 VAR 1	NS	25	30	1000	0.7	500	100-600	100	200	10	800	O/P	BC337
BC546	TO-92 VAR 1	NS	80	80	100	0.8	100	125-500	2	300	10	500	S.S. Amp.	
BC547	TO-92 VAR 1	NS	45	50	100	0.6	100	110-800	2	300	10	500	S.S. Amp.	BC107, BC207, BC147
BC548	TO-92 VAR 1	NS	30	30	100	0.6	100	110-800	2	300	10	500	S.S. Amp.	BC108, BC208, BC148
BC549	TO-92 VAR 1	NS	30	30	100	0.6	100	200-800	2	300	10	500	Low Noise Small Sig.	BC109, BC209, BC149
BC549C	TO-92 VAR 1	NS	30	30	100	0.6	100	420-800	2	300	10	500	Low Noise High Gain	SC109C, BC149C
BC556	TO-92 VAR 1	PS	80	80	100	0.65	100	75-500	2	150	10	500	S.S. Amp.	
BC547	TO-92 VAR 1	NS	45	50	100	0.6	100	110-800	2	300	10	500	S.S. Amp	BC107, BC207, BC147
BC548	TO-92 VAR 1	NS	30	30	100	0.6	100	110-800	2	300	10	500	S.S. Amp.	BC108, BC208, BC148
BC549	TO-92 VAR 1	NS	30	30	100	0.6	100	200-800	2	300	10	500	Low Noise Small Sig.	BC109, BC209, BC149
BC549C	TO-92 VAR 1	NS	30	30	100	0.6	100	420-800	2	300	10	500	Low Noise High Gain	BC109C, BC149C
BC556	TO-92 VAR 1	PS	80	80	100	0.65	100	75-500	2	150	10	500	S.S. Amp.	
BC557	TO-92 VAR 1	PS	45	50	100			110-330	2	150		500	G.P. Small Sig.	BC157, DS557
BC558	TO-92 VAR 1	PS	30	30	100			75	2	150		500	G.P. Small Sig.	BC158, DS558
BC559	TO-92 VAR 1	PS	30	30	100			125-600	2	150		500	G.P. Small Sig.	BC159
BC635	TO-92(74)	NS	45	45	1A	0.5	500	40-250	150	130	500	1W	Audio O/P	BC639
BC636	TO-92(74)	PS	45	45	1A	0.5	500	40-250	150	130	500	1W	Audio O/P	BC640
BC639	TO-92(74)	NS	80	100	1A	0.5	500	40-160	150	130		1W	Audio O/P	MU9610, TT801
BC640	TO-92(74)	PS	80	100	1A	0.5	500	40-160	150	50		1W	Audio O/P	MU9660, TT800
BCY70	TO-18	PS	40	50	200	0.5	50	50	10	250	50	350	G.P.	BC212
BCY71	TO-18	PS	45	45	200	0.5	50	100-600	10	200	50	350	G.P.	BC212
BCY72	TO-18	PS	25	25	200	0.5	50	50	10	200	50	350	G.P.	BC213
BD137	TO-126	NS	60	60	1A	0.5	500	40-160	150	250	500	8W	G.P. O/P	BD139
BD138	TO-126	PS	60	60	1A	0.5	500	40-160	150	75	500	8W	G.P. O/P	BD140
BD139	TO-126	NS	80	100	1A	0.5	500	40-160	150	250	500	8W	G.P. O/P	40409
BD140	TO-126	PS	80	100	1A	0.5	500	40-160	150	75	500	8W	G.P. O/P	40410
BD262	TO-126	PS	60	60	4A	2.5	1.5A	750	1.5A	7	1.5A	36W	High Gain Darl. O/P	BD266
BD263	TO-126	NS	60	60	4A	2.5	1.5A	750	1.5A	7	1.5A	36W	High Gain Darl. O/P	BD267
BD266A	TO-220	PS	80	80	8A	2	3A	750	3A	7		60W	High Gain Darl. O/P	
BD267A	TO-220	NS	80	100		2	3A	750	3A	7		60W	High Gain Darl. O/P	
BD681	TO-126	NS	100	100	4A			500	4A	1		40W	Darlington O/P	BD263
BD682	TO-126	PS	100	100	4A			750	1.5A	1		40W	Darlington O/P	BD263
BDY20	TO-3	NS	60	100	15A	1.1	4A	20-70	4A	1	4A	115	Power O/P	2N3055
BF115	TO-72(28)	NS	30	50	30			45-165	1	230	1	145	V.H.F. Amp	
BF167	TO-72(28)	NS	40	25				26	4	350	4	130	T.V. I.F. Amp	
BF173	TO-72(28)	NS	25	40	25			92	7	550	5	230	T.V. I.F. Amp	
BF178	TO-39	NS	115	185	50			20	30	120	10	600	T.V. Video Amp	BF336
BF179	TO-39	NS	115	250	50			20	20	120	10	600	T.V. Video Amp	BF338
BF180	TO-72(25)	NS	20	30	20			13	2	675	2	150	U.H.F. Amp	BF200
BF184	TO-72(28)	NS	20	30	30			75-750	1	300	1	145	H.F. Amp	
BF185	TO-72(28)	NS	20	30	30			34-140	1	220	1	145	H.F. Amp	BF195
BF194	SOT-25/1	NS	20	30	30			65-220	1	260	1	250	H.F. Amp	
BF195	SOT-25/1	NS	20	30	30			35-125	1	200	1	250	H.F. Amp	BF185
BF199	TO-92 (VAR)2	NS	25		25			37	7	550		500	H.F. Amp	BF180
BF200	TO-72(25)	NS	20	30	20			15	3	650	3	150	V.H.F. Amp	BF180
BF336	TO-39	NS	180	185	100			20-60	30	130		3W	Video Amp	
BF337	TO-39	NS	200	300	100			20-60	30	130		3W	Video Amp	
BF338	TO-39	NS	225	250	100			25-60	30	130		3W	Video Amp	
BF463	TO-202	PS	25	50	500	-	-	40-180	30	-	2W	20	H.V. Med Pwr	
BF469	TO-126	NS	250	250	50			50	25	60		1.8W	G.P. High V. Amp	
BF470	TO-126	PS	250	250	50			50	25	60		1.8W	G.P. High V. Amp	
BFR90	SOT-37(2)	NS	15	20	25			25-250	14	5GHz	14	180	Wide Band Amp	
BFR91	SOT-37(2)	NS	12	15	35	0.3	30	25-250	30	5GHz	30	180	Wide Band Amp	
BFY50	TO-39	NS	35	80	1A	0.2	150	30	150	60	50	2.85W	G.P.	
BFY51	TO-39	NS	30	60	1A	0.35	150	40	150	50	60	2.85W	G.P.	
BFY52	TO-39	NS	20	40	1A	0.35	150	60	150	50	50	2.85W	G.P.	
BFY90	TO-72(25)	NS	30	30	25			25-125	2	1.3GHz	25	200	U.H.F. Amp	

THEORY ASSIGNMENT 3

1. Draw a block diagram showing how a dual trace CRO can be connected to measure the voltage gain of an amplifier.

2. Calculate the voltage gain of an amplifier if a 10kHz input signal of 200mVp-p produces an output of 2.4Vp-p.

3. List the effect an increase in temperature will have on the base current and the collector current of a silicon transistor.

4. An amplifier with an output resistance of 1k produces an output signal of 20Vp-p when it is not connected to a load. Calculate the output voltage if the amplifier is connected to a load of:

   (a) 1k,

   (b) 100 ohms

5. Using the data sheet on the previous page, determine the most suitable replacement for a BC107 transistor from a choice of transistor types BC108, BC109, BC157, BC177, BC327, BC337 and BC547.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 4

References: Electronic Devices, 2nd Ed. Floyd. Chapters 5 & 6.

TRANSISTOR BIASING

OBJECTIVES At the end of this lesson you will be able to:

- Calculate the quiescent DC conditions of a transistor amplifier with single resistor biasing.
- Calculate the quiescent DC conditions of a transistor amplifier with potential divider biasing.
- List the advantages of potential divider biasing compared to single resistor biasing.
- Describe the purpose of the emitter resistor in a transistor amplifier.

1. INTRODUCTION

When used as an amplifier, a transistor needs to be forward biased. That is, its base-emitter junction must have a voltage across it of 0.6V. This voltage needs to be supplied from the DC power supply and various methods are used to achieve this. To act as a voltage amplifier, a collector resistor is also needed to convert the current changes to a voltage change. These notes describe two methods of biasing a transistor and how the DC conditions are calculated.

2. INTRODUCTION TO BIASING

As described previously, a transistor needs to be forward biased to function. For an NPN transistor, this means the base terminal must have a positive DC voltage 0.6V higher than the emitter. For a PNP transistor the base must be 0.6V lower than the emitter. The term biasing refers to the presence of the DC voltage across the base-emitter junction, and the circuit to derive it is known as the biasing circuit. There are various methods of producing the required bias of 0.6V and they usually involve one or more resistors connected to the DC power supply for the circuit.

When the base-emitter junction is forward biased, base current flows. When base current flows, collector current also flows, and the value of the collector current will equal ßIB. A voltage amplifier needs to produce an output voltage that is an amplified version of the input voltage and a resistor has to be connected between the collector and the DC power supply to convert the variations in the collector current to a voltage change.

The simplest possible transistor amplifier circuit is shown in Fig.1, in which RB supplies the bias voltage for the base-emitter and RC converts the collector current variations to a voltage variation.

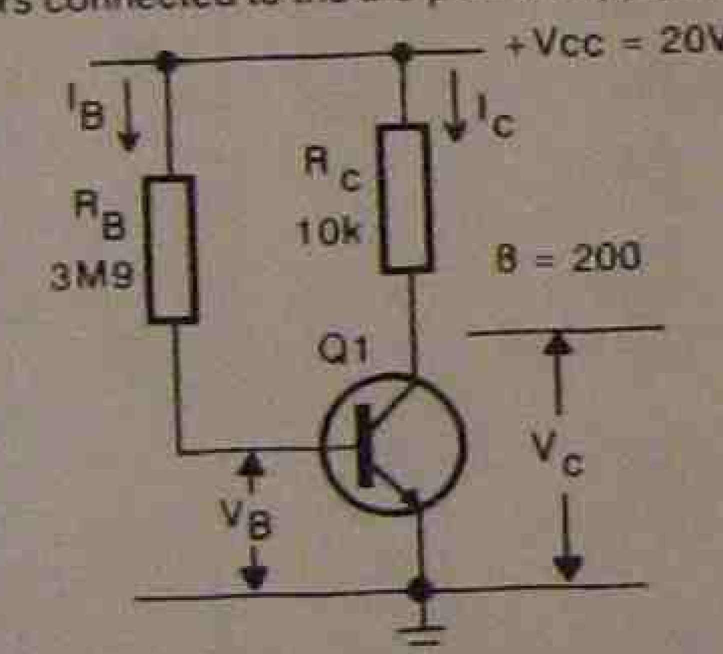


Fig.1: Basic transistor amplifier circuit

3. DC CONDITIONS

When an amplifier has no input signal, the DC voltages around the circuit are referred to as the DC **quiescent** voltages. That is, it is ready to go, in the same way a car engine idles at standstill. The DC values that need to be considered are shown in Fig.1 and are defined as follows:

- IB = base current in Q1
- Ic = collector current (equals ßIB)
- VB = base voltage (measured from base terminal to ground)
- Vc = collector voltage (measured from collector to ground)
- Vcc = value of the supply voltage

Note that in most cases, the collector voltage should equal half the supply voltage so that it can vary equally in either direction.

4. CALCULATING DC QUIESCENT CONDITIONS FOR FIG.1

For the single resistor base bias circuit of Fig.1 the DC conditions are calculated as follows:

(a) $I_B = \frac{V_{cc} - V_{be}}{R_B} = \frac{20 - 0.6}{3M9} = \underline{5\mu A}$

(b) $I_c = \beta I_B = 200 \times 5\mu A = \underline{1mA}$

(c) $V_c = V_{cc} - I_c R_c = 20 - (10k \times 1mA) = \underline{10V}$

5. CIRCUIT ANALYSIS

In Fig.1, the base current is supplied through RB, and the collector current that results is simply the base current multiplied by the DC current gain (ß) of the transistor. The collector voltage (Vc) is determined by subtracting the voltage drop across the collector resistor Rc from the supply voltage Vcc. If an AC signal is applied to the base terminal, via a DC blocking capacitor, the base current will vary with the AC signal, increasing when the AC input goes positive, and decreasing when it goes negative. The changes in the base current will produce corresponding changes in the collector current, and the voltage drop across the collector resistor will also change. Thus, an amplified version of the input voltage will appear at the collector terminal.

However, this circuit is too simple to be practical for the following reasons:

- Temperature change: When the temperature changes, the barrier voltage of the base-emitter PN junction changes. Thus, if the temperature rises, the barrier voltage drops, allowing more base current to flow. When more base current flows, the collector current increases, increasing the voltage drop across the collector resistor. This causes the collector voltage Vc to drop, and if the temperature rise is substantial the collector voltage will drop by a large amount.
- Change in ß: A change in the current gain (ß) occurs to some extent with temperature, but large changes can occur if the transistor is replaced. As described in previous notes, a transistor of a given type is usually specified as having a current gain between a range of figures. In the circuit of Fig.1, the current gain of the transistor is assumed to be 200, and the resistor values have been determined on this basis. However, if the transistor is replaced with one having a current gain of say 300, then the DC conditions for the circuit will alter as IB will change, in turn changing Ic which then alters the value of Vc. The proof is shown below.

because:	$V_c = V_{cc} - I_c R_c$
and:	$I_c = \beta I_B$
therefore:	$V_c = V_{cc} - \beta I_B R_c$
Thus:	**as ß is in the above equation, a change in its value affects Vc**

Example: Calculate the value of Vc for the circuit of Fig.1 if the value of ß increases from 200 to 300.

$I_B = 5\mu A$ (as already calculated)

$I_c = \beta I_B = 300 \times 5\mu A = 1.5mA$

$V_c = V_{cc} - I_c R_c = 20 - (1.5mA \times 10k) = \underline{5V}$

Note: **A 50% increase in ß has caused a 50% decrease in Vc**

6. POTENTIAL DIVIDER BIASING

To overcome the problems of the circuit of Fig.1, the base bias voltage is produced with a potential divider circuit. As well, a resistor is connected between the emitter terminal and ground. The combined effect is a circuit that is very stable against changes in current gain and changes in temperature. This circuit is shown in Fig.2 where:

- I_C = collector current (also equals I_E)
- V_B = base voltage (from base to ground)
- V_{be} = voltage between base & emitter (0.6V)
- V_C = collector voltage (collector to ground)
- V_E = emitter voltage (emitter to ground)
- V_{CC} = value of the supply voltage

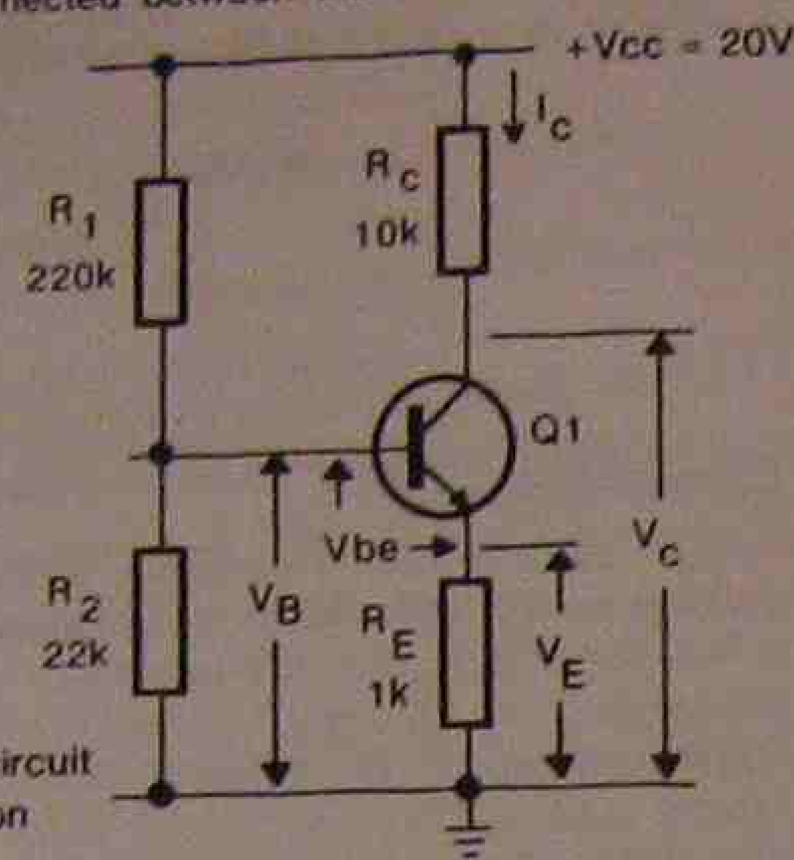


Fig.2: Potential divider bias circuit with emitter stabilisation

7. ANALYSIS OF FIG.2

The circuit of Fig.2 is typical and its analysis relies on three approximations:

- Base current is small enough to be ignored.
- The emitter current equals the collector current.
- The base-emitter voltage (Vbe) equals 0.6V.

The analysis is as follows:

(a) The base voltage (V_B) is determined by the potential divider formed by R1 and R2.

$$V_B = \frac{V_{CC} \times R2}{R1 + R2}$$

(b) The emitter voltage (V_E) equals V_B - Vbe

$$V_E = V_B - V_{be} = V_B - 0.6V$$

(c) The emitter current equals the collector current which equals the emitter voltage (V_E) divided by R_E:

$$I_E = I_C = \frac{V_E}{R_E}$$

(d) The collector voltage (Vc) equals the supply voltage (Vcc) minus the voltage drop across the collector resistor Rc.

$$V_C = V_{CC} - I_C R_C$$

Note that the DC current gain β is not used in any of the above equations. Thus, the circuit is not dependent on the β of the transistor, and the resistor values alone set the DC conditions. As well, a change in Vbe will only cause a small change in the emitter current, as the emitter current is determined by the voltage across the emitter resistor and not by the base current. The stability of the circuit relies on the potential divider of R1 and R2 providing a fixed value of voltage for V_B and on the feedback supplied by R_E.

8. EFFECT OF EMITTER RESISTOR

In Fig.2, resistor R_E is connected from the emitter terminal to ground. Its purpose is to help stabilise the DC conditions against changes in temperature and changes in β. To explain, consider the effect of an increase in temperature.

(a) The barrier voltage (Vbe) in the base-emitter junction drops from 0.6V to some lower value (by -2.5mV/°C)

(b) The base current increases as V_B is fixed by R1 and R2 and the barrier voltage (Vbe) has now fallen.

(c) The emitter current increases as the base current has increased.

(d) The voltage drop across R_E rises because the emitter current has increased.

(e) As V_B is fixed by R1 and R2, and V_E has now increased, the voltage difference across the base-emitter junction drops. Thus, I_B drops and the circuit voltages returns to normal.

That is, R_E has provided feedback and has corrected against a change in temperature. The correction process will be immediate and the stability of the circuit depends on the value of R_E. The higher the value of R_E the greater the stability, but the greater the circuit losses. The lower the value of R_E, the less the feedback and the lower the stability.

as a general rule, for the circuit of Fig.2:

R1 = 10R2, Rc = 10RE and $V_C = \frac{V_{CC}}{2}$

9. WORKED EXAMPLE

The DC conditions for a potential divider, emitter stabilised amplifier circuit are calculated using the equations listed above. The following shows the full working to calculate the DC conditions for Fig.2. Note that in this example, Vc does not equal half Vcc although it is close enough for the circuit to function correctly. This illustrates that it is incorrect to assume that Vc always equals Vcc/2.

(a) $V_B = \frac{V_{CC}\,R2}{R1 + R2} = \frac{20 \times 22k}{220k + 22k} = \underline{1.8V}$

(b) $V_E = V_B - V_{be} = 1.8 - 0.6 = \underline{1.2V}$

(c) $I_E = I_C = \frac{V_E}{R_E} = \frac{1.2}{1k} = \underline{1.2mA}$

(d) $V_C = V_{CC} - I_C R_C = 20 - (10k \times 1.2mA) = \underline{8V}$

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 4

1. For the circuit of Fig.1, calculate the following, given that the current gain of the transistor is 200

   (a) I_B

   (b) I_C

   (c) V_C

   (d) Briefly describe the effect on the circuit if the current gain of the transistor decreases.

   (e) Describe the effect an increase in temperature has on I_B and I_C

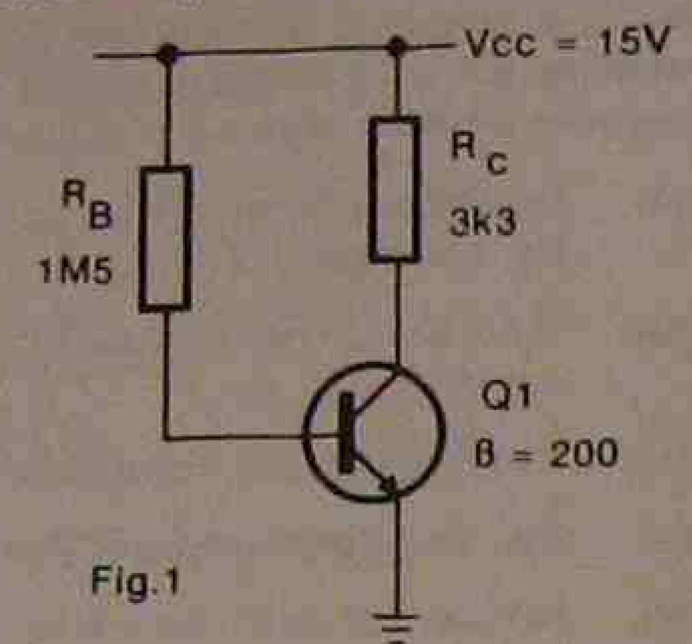


Fig.1

2. For the circuit of Fig.2, determine the value for R_B required to give a value of collector voltage equal to half the supply voltage.

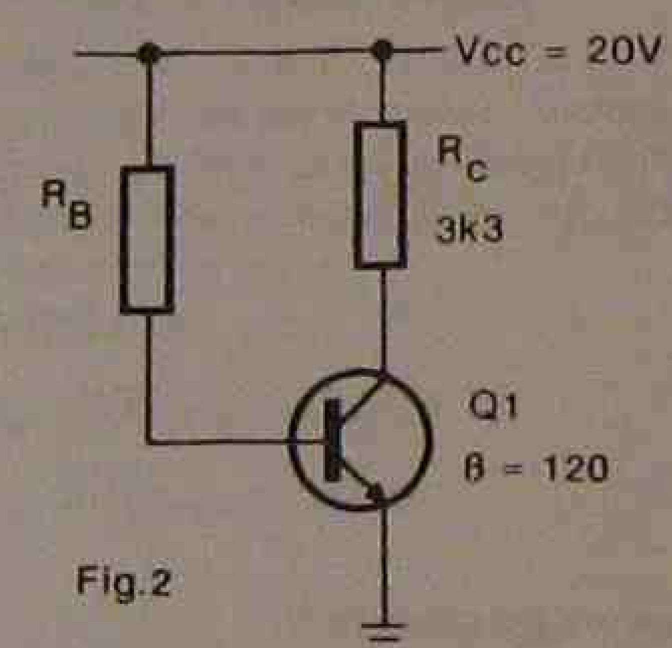


Fig.2

3. Calculate the base voltage (V_B), emitter voltage (V_E) and the collector voltage (V_C) for the circuit of Fig.3.

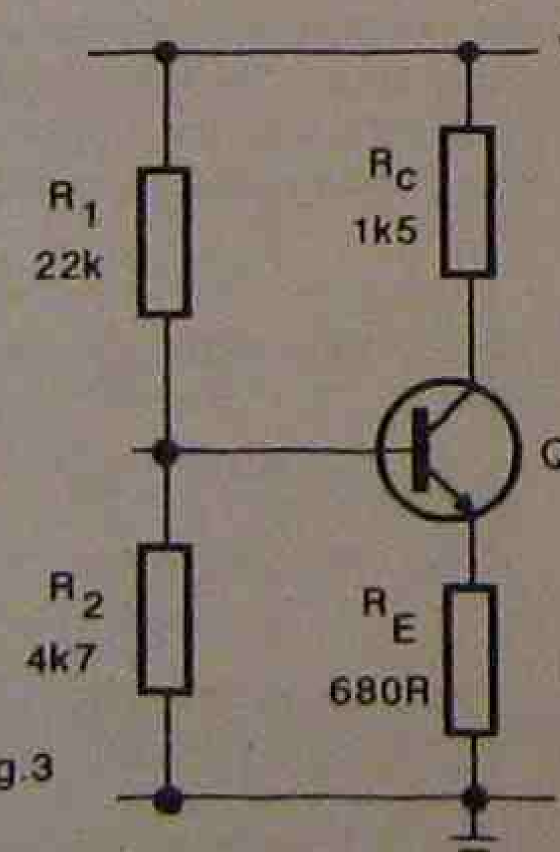


Fig.3

4. Give two reasons why the circuit of Fig.3 is more stable against changes in temperature and β than Figs.1 or 2.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 5

References: Electronic Devices, 2nd Ed. Floyd. Chapters 6 & 8.

TRANSISTOR AMPLIFIERS - PART 1

OBJECTIVES At the end of this lesson you will be able to:

- Calculate the voltage gain, the input and the output resistance of a common emitter transistor amplifier.
- Draw the waveforms present at various points around a common emitter amplifier.

1. INTRODUCTION

These notes examine the AC conditions of a transistor amplifier. So far, circuit analysis has concentrated on the DC conditions, as the DC conditions need to be correct before the circuit can function. The equations to calculate voltage gain, input and output resistance are described and the term 'common emitter amplifier' is defined.

2. THE COMMON EMITTER AMPLIFIER

The circuit of Fig.1 shows a common emitter amplifier that uses potential divider biasing and emitter stabilisation. The term 'common emitter' is used to indicate that the emitter terminal of the transistor is neither an input or an output, but instead connects to the common (or ground) rail. There are two other possible configurations, called the common collector and the common base which will be described in the notes for Week 6.

As shown in Fig.1, the input voltage is applied via a coupling capacitor (C1) to the base of Q1, and the output is taken from the collector, again through a coupling capacitor (C2). The capacitors are used to prevent the DC voltages present at the base and collector terminals from being affected by the signal source or the load. If the capacitor C1 was not present, the signal source would act as a resistor in parallel with R2 and alter the DC bias voltage to the base of Q1, affecting the rest of the DC conditions. If the load resistor RL was connected directly to the collector of Q1, the DC collector voltage would drop, as RL would now be in parallel with Q1, giving two paths for the current in Rc. Capacitors C1 and C2 are referred to as coupling capacitors, and because their value usually exceeds 1μF, they need to be an electrolytic type, requiring their polarity to be correct.

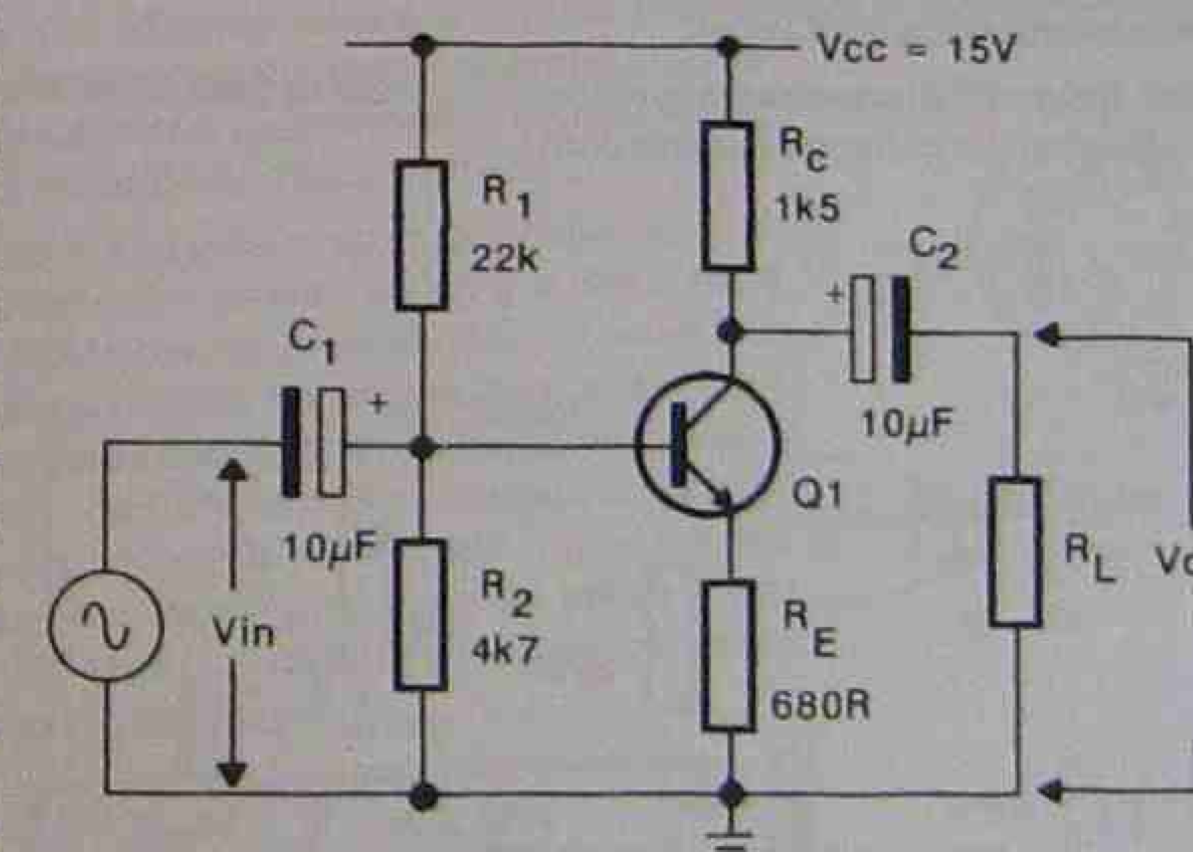


Fig.1: The common emitter amplifier

The method to derive the DC conditions for Fig.1 was presented in the notes for Week 4, and it is essential to be able to calculate the DC conditions of the circuit before attempting to analyse its AC conditions.

3. AC CONDITIONS

The AC characteristics of a CE transistor amplifier that will be described in these notes are:

- voltage gain
- input resistance
- output resistance

These three characteristics were described in the notes for Week 3, and are listed as AC conditions, as they are values applicable to an AC input/output only. When considering the AC conditions, it is useful to remember that all coupling capacitors are a short circuit to an AC signal, and an open circuit for DC conditions. The only time this isn't the case is when the frequency of the signal is low enough to make the capacitive reactance of the capacitors equal to, or higher than the resistance values around the circuit. For the purposes of these notes, it can be assumed that the capacitors are a short circuit to an AC signal, unless otherwise stated.

The circuit of Fig.2 shows the CE amplifier described in the notes for Week 4. The component values are the same, and all the DC conditions were calculated and presented as an example in these notes. However, now the AC conditions are being included, as depicted by the various waveforms around the circuit. Note the following about these waveforms:

- The voltage at the base terminal (1.8V) is isolated from the signal source. Thus, the input signal has no DC component, (AC only) while the voltage at the base has both a DC and an AC component.
- The voltage at the output terminal again has no DC component, (isolated by C2), and the voltage at the collector is an AC signal superimposed around a DC voltage of 8V.
- The output signal is 180° out of phase with the input signal. Also, it is larger than the input signal.
- The waveform at the emitter is identical to that at the base, except the DC component has been reduced by 0.6V (Vbe).
- The height of all waveforms is shown as a peak to peak value and the values shown are with respect to the common line (ground).

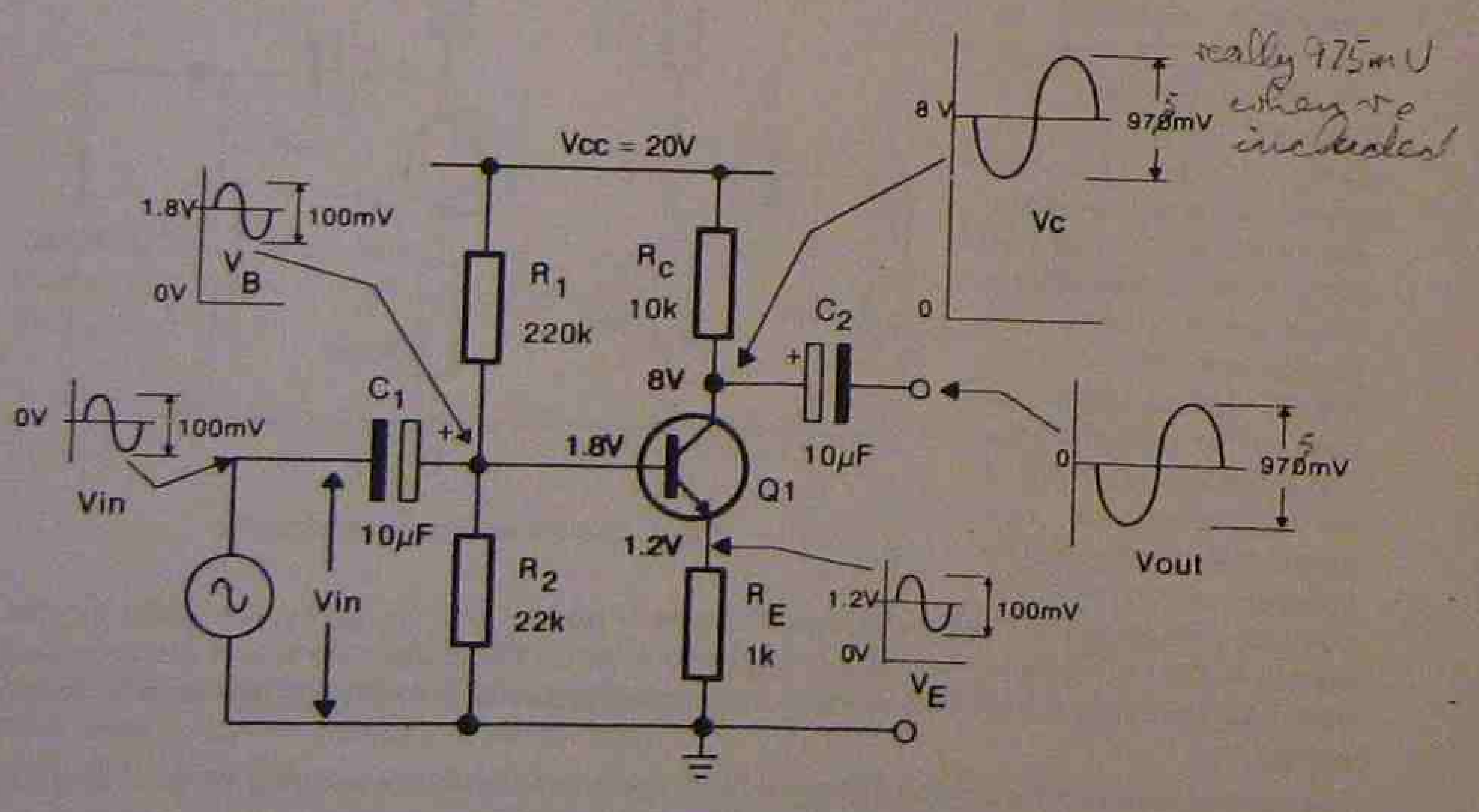


Fig.2: *Waveforms for the common emitter (CE) amplifier*

4. VOLTAGE GAIN

The circuit of Fig.2 produces a larger output signal than the input signal as a result of the current gain of the transistor. To convert the current change in the collector to a voltage change, the collector resistor (Rc) is required. When the input signal becomes more positive, the base current in the transistor will increase. This will increase the value of the collector current, and the voltage drop across Rc will also increase. The voltage at the collector (Vc) will therefore drop, as more of the supply voltage is now lost across Rc. This explains why the output signal is 180° out of phase with the input. Because the collector current rises, the emitter current (which virtually equals the collector current) also rises, increasing the voltage drop across RE. Thus the signal at the emitter terminal is in phase with the input signal, but displaced by 0.6V, due to Vbe.

As previously described, voltage gain equals output/input. For voltage gain (Av):

$$Av = \frac{Vo}{Vin}$$

This equation can only be used if the input and output voltages are measured, and it is usual to calculate the gain using an equation based on component values within the circuit. As already explained, the AC signal across the emitter resistor (RE) has the same amplitude as the input signal Vin. As an approximation, because Vin appears across RE, and Vout is effectively across Rc, the ratio of Rc and RE represent the gain of the circuit. That is,

$$Av = \frac{Rc}{R_E} \quad \text{.......... (1)}$$

However, equation (1) ignores two important aspects. The first is the effect of adding a load to the circuit and the second is the presence of an additional, though hidden value of resistance in the emitter circuit. As Ohm's law states, when a current and a voltage are present across a component, there must be resistance. In the base-emitter junction of a transistor, there is a current in the emitter (equals Ic + IB) and a voltage across the base-emitter junction; Vbe (0.6V). Although most of the 0.6V is the result of the barrier potential formed at the PN junction, there is an additional resistance that causes Vbe to rise slightly as the current (Ic) through the transistor rises.

This resistance is shown in Fig.3, and is referred to as re (little r e). The value of re changes with the collector current, dropping as the current increases, explaining why Vbe rises by a small amount when the collector current increases substantially. For this reason, re is known as a **dynamic** resistance. Its value therefore needs to be determined at a specified value of collector current, usually the quiescent current. The equation used to find re is an approximation, and it only applies for currents less than 20mA or so. The equation is shown below and as Ic must be a current in the order of milliamps, re can be easily calculated by dividing the current (in mA) into 30.

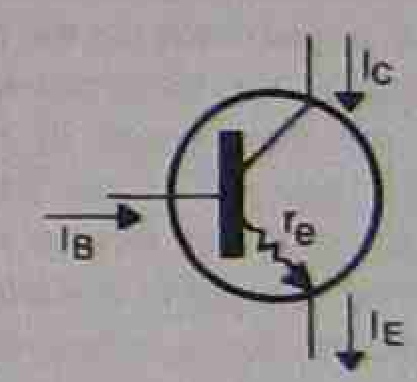


Fig.3: dynamic resistance re

$$r_e = \frac{30mV}{Ic\ mA}$$

For the circuit of Fig.2, because Ic equals 1.2mA (see week 4 notes for working), re equals 30mV/1.2mA, or simply 30/1.2, giving a value of 25 ohms.

Because re is in series with RE, it needs to be included in the gain equation, although in some cases its value is low enough to ignore. (That is, ignore if re is less than RE/10). Thus, the gain equation becomes:

$$Av = \frac{Rc}{R_E + r_e} \quad \text{.......... (2)}$$

To allow for the drop in gain when a load is connected to the amplifier, the value of the load resistor must be included in the equation. In effect, the load resistor RL is in parallel with the collector resistor (Rc) so the gain equation that takes into account re and RL is:

$$Av = \frac{Rc//RL}{RE + re}$$

For the circuit of Fig.2, there is no load resistor connected, so equation (2) on the previous page can be used to calculate the gain. The gain is therefore 10k/(1k + 25) which equals 9.75. Thus and input signal of 100mV as shown will produce an output of 100mV x 9.75, giving 975mV at the output, or approximately 970mV as indicated on the waveforms. The equation shown above takes care of all possibilities, and RL will therefore appear as an open-circuit (infinite resistance), leaving Rc as the only component value in the top line if there is no load resistor connected.

5. INPUT RESISTANCE

As explained in the notes for week 3, input resistance is the effective resistance of the amplifier between its input terminals. To determine this value mathematically, all resistors in the circuit that offer a path to ground for the AC input signal need to be considered. Because a capacitor is a short circuit to AC, and because the power supply rail will have some form of filter capacitor, the supply rail is effectively an AC ground. This means that the bias resistors are in parallel, as far as an AC signal is concerned.

The other AC path to ground offered by the circuit of Fig.2 is from the base terminal, through the emitter to ground via any resistance in this path. However, the value of resistance represented by this path is affected by the current gain of the transistor, due to the emitter current flowing in the transistor. The equivalent circuit is shown in Fig.4 which shows the three possible paths the AC signal current can take. The path to the right of the dotted line is via the base terminal, and as a result of the transistor action, all resistance after the base needs to be multiplied by the current gain (β) of the transistor. That is, the series connected emitter resistor (RE) and the dynamic resistor re are effectively increased in resistance to a value equal to their sum multiplied by the β of the transistor. The equation to calculate input resistance (rin) is shown below:

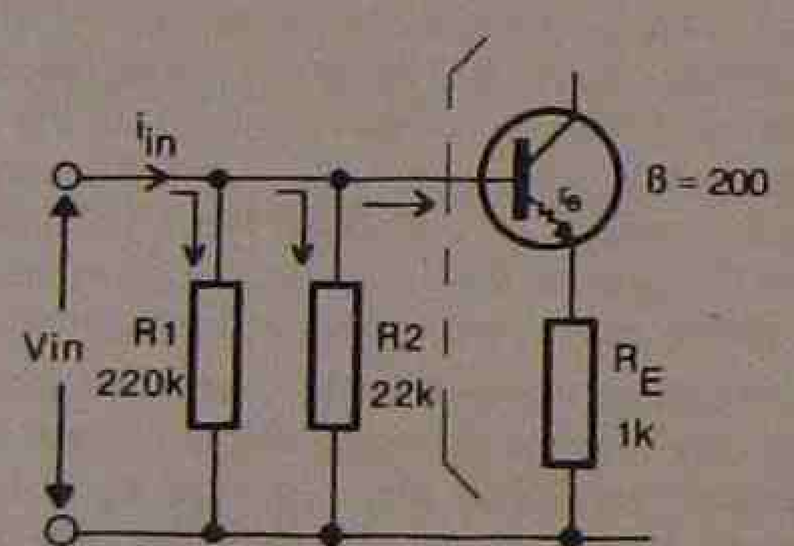


Fig.4: equivalent circuit for input resistance.

$$rin = R1//R2//[\beta(re + RE)]$$

For the equivalent circuit of Fig.4, rin = 220k//22k//[200(25 + 1k)], which gives 18.2k ohms. (Note, // means 'in parallel with')

OUTPUT RESISTANCE

It can be shown that the collector resistor is effectively in series with the output terminal of the amplifier and the internal voltage generator (Av x vin). (See notes for week 3 for the equivalent circuit.) Thus, the equation to find output resistance ro is:

$$ro = Rc$$

7. ADDING AN EMITTER BYPASS CAPACITOR

The equation for gain shows that the value of the emitter resistor (RE in Fig.2) affects the voltage gain of the circuit. For optimum DC stability, RE is usually greater than one tenth the value of the collector resistor, which means the gain of the circuit cannot exceed 10. To overcome this limitation, while retaining the best DC conditions, the emitter resistor can be made up of two resistors, with a bypass capacitor (CE) connected across one of these as shown in Fig.5. The bypass capacitor can also be connected across a single emitter resistor (RE in Fig.2) or across either emitter resistor (RE1 or RE2 in Fig.5). However, while the gain of the circuit will be increased, the input resistance of the amplifier will be reduced as illustrated by the following example.

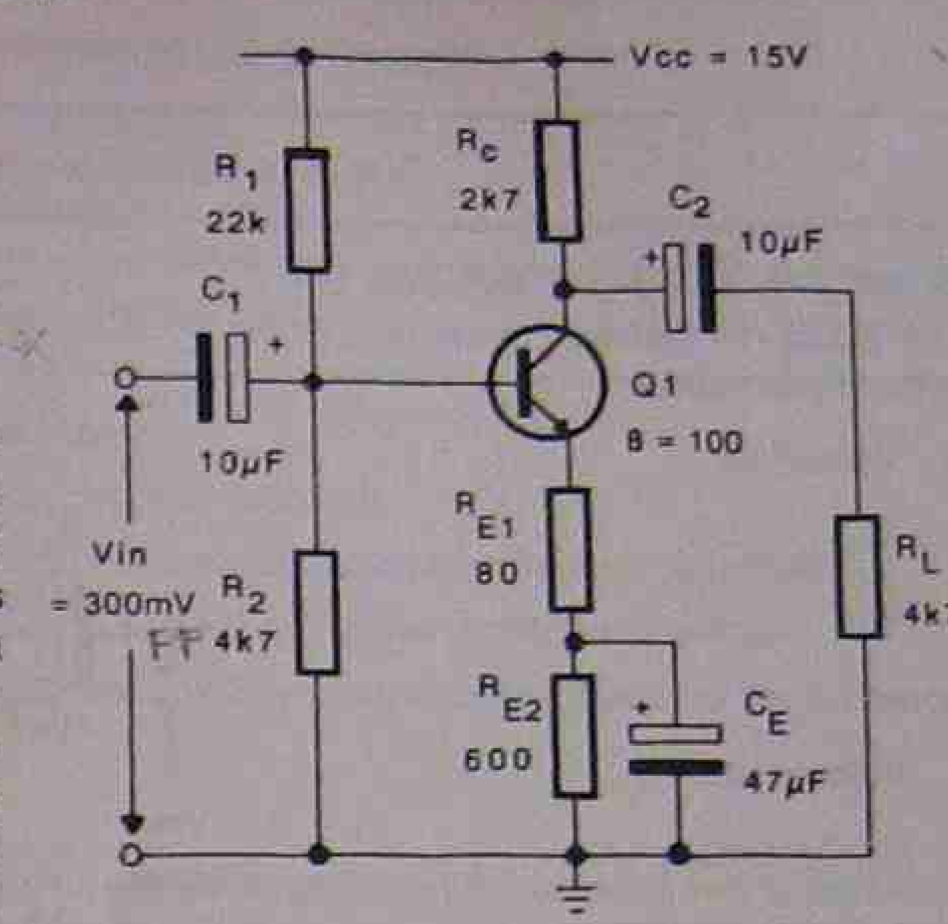


Fig.5: Adding a bypass capacitor

8. WORKED EXAMPLE

For the circuit of Fig.5, calculate:

(a) the DC voltages present at the base, emitter and collector of Q1.

(b) the gain of the circuit.

(c) the input resistance.

(d) the output resistance.

(e) the gain and input resistance if CE is disconnected.

(a) $VB = \frac{Vcc\,R2}{R1 + R2} = \frac{15 \times 4k7}{22k + 4k7} = \underline{2.6V}$

$VE = VB - Vbe = 2.6 - 0.6 = \underline{2V}$

$IE = Ic = \frac{VE}{RE} = \frac{2}{680} = 3mA$

$Vc = Vcc - IcRc = 15 - (2k7 \times 3mA) = \underline{6.9V}$

$re = \frac{30mV}{Ic} = \frac{30mV}{3mA} = 10 \text{ ohms}$

(b) $Av = \frac{Rc//RL}{re + RE1} = \frac{2k7//4k7}{10 + 80} = \frac{1715}{90} = \underline{19}$

(c) rin = R1//R2//[β(re + RE1)] = 22k//4k7//[100(10 + 80)] = 3k87//9k = **2k71 ohms**

(d) ro = Rc = **2k7 ohms**

(e) $Av = \frac{Rc//RL}{re + RE1 + RE2} = \frac{2k7//4k7}{10 + 80 + 600} = \frac{1715}{690} = \underline{2.5}$ (CE disconnected)

rin = R1//R2//[β(re + RE1 + RE2)] = 22k//4k7//[100(10 + 80 + 600)]
= 3k87//69k = **3k7ohms** (CE disconnected)

VBE = 0·6
β = 100

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 5

1. For the circuit of Fig.1, calculate the following, given that the current gain of the transistor is 200
   - (a) V_B and V_E
   - (b) I_C
   - (c) V_C
   - (d) r_e
   - (e) Voltage gain A_v
   - (f) input resistance r_{in}
   - (g) output resistance r_o
   - (h) output voltage v_o

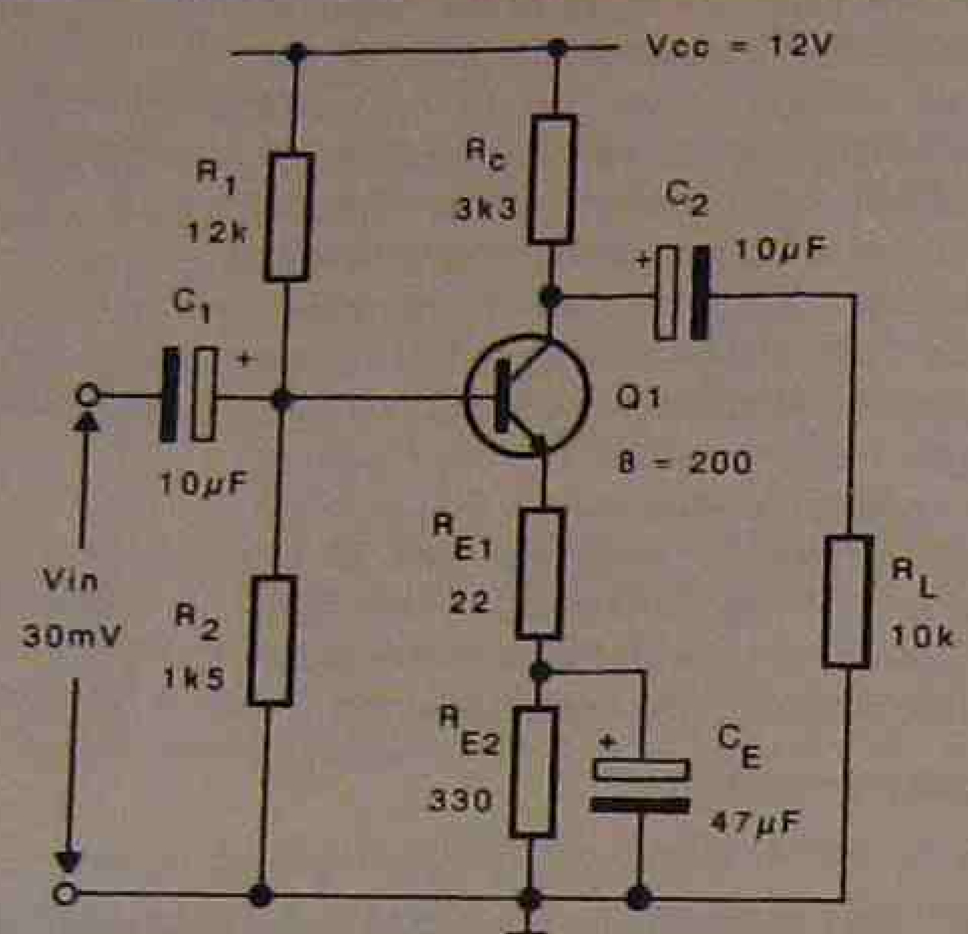
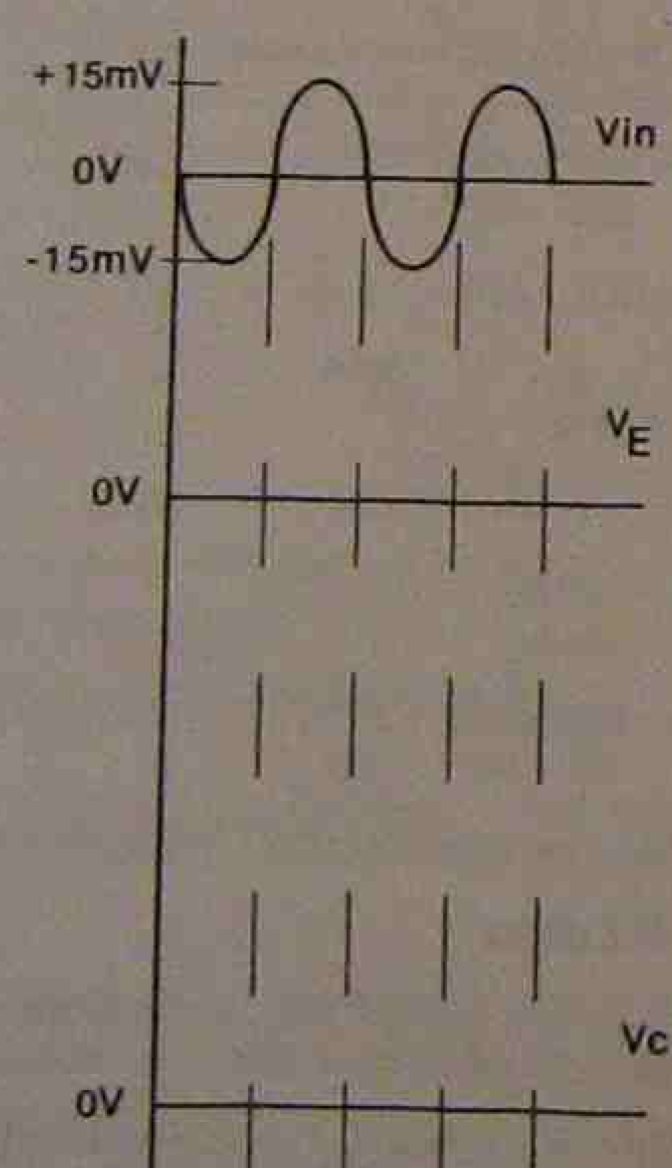


Fig.1

2. For the circuit of Fig.1 determine the voltage gain, output voltage and input resistance if capacitor C_E becomes an open circuit.

3. Using the values obtained in question 1, sketch the waveforms that should be present at the emitter and the collector of Q1. Include the DC component and show the correct phase relationship these waveforms will have to the input signal v_{in}. It is not necessary to draw the waveforms to scale, but all DC voltages and the peak to peak values of the waveforms should be shown.



****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 6

References: Electronic Devices, 2nd Ed. Floyd. Chapter 8.

TRANSISTOR AMPLIFIERS - PART 2

OBJECTIVES At the end of this lesson you will be able to:

- Calculate the DC conditions for a common collector amplifier
- Calculate the voltage gain, input resistance of a common collector transistor amplifier.
- Draw the waveforms present at various points around a common collector amplifier.
- List the characteristics of a common collector amplifier and compare these to the common emitter amplifier.

1. INTRODUCTION

The common emitter (CE) amplifier has been described in the notes for weeks 4 and 5, and these notes examine the common collector (CC) amplifier. This amplifier has a similar circuit to the CE amplifier, except the output is taken from the emitter terminal. Most of the calculations used with the CE amplifier apply to the CC amplifier, except the gain equation, which, as will be shown, equals unity. The purpose of the CC amplifier is also described, as this circuit has many applications in electronics.

2. THE COMMON COLLECTOR AMPLIFIER

The circuit of Fig.1 shows a common collector amplifier that uses potential divider biasing. The term 'common collector' is used to indicate that the collector terminal of the transistor is neither an input or an output. In effect, the collector terminal is connected to the common line, but through the DC supply. This refers to the AC conditions only, as the filter capacitors associated with the DC supply effectively connect all AC signals to ground.

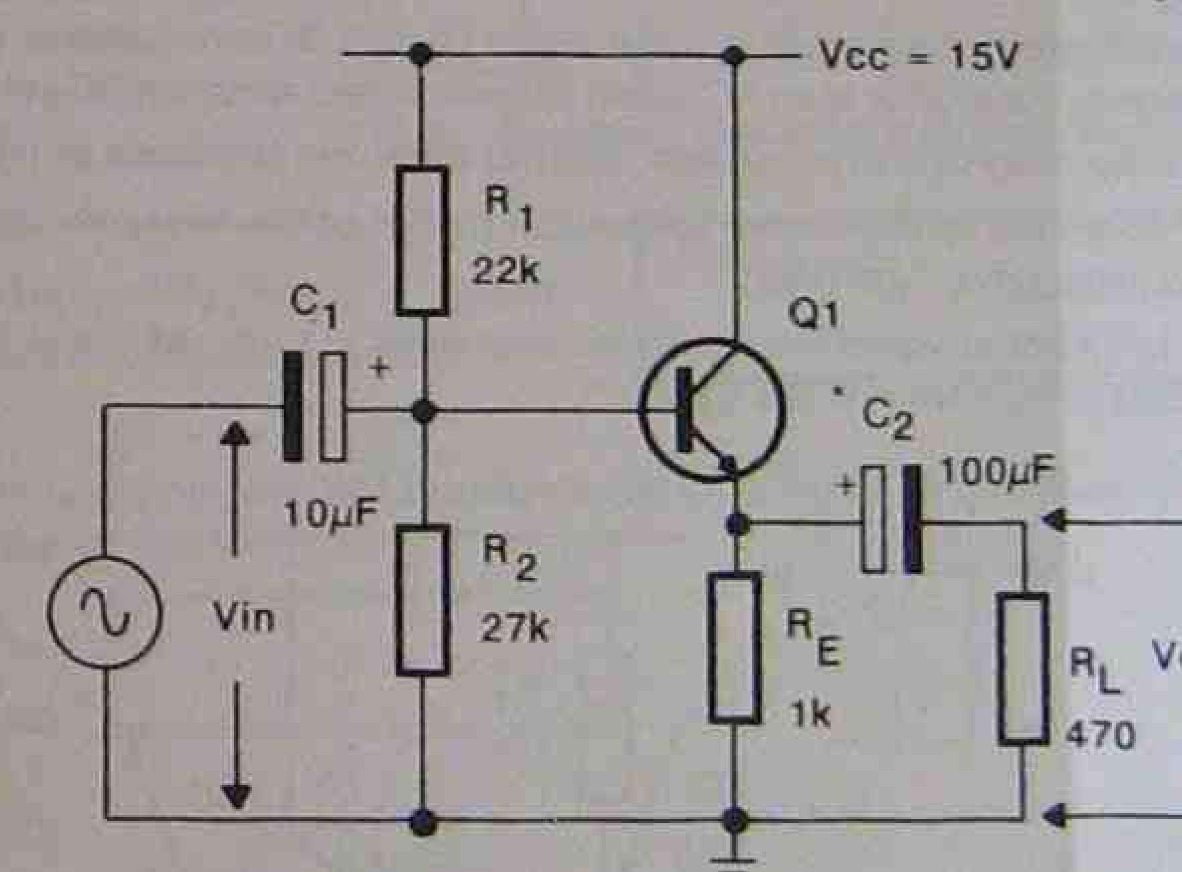


Fig.1: The common collector amplifier

As shown in Fig.1, the input voltage is applied via a coupling capacitor (C1) to the base of Q1, and the output is taken from the emitter, again through a coupling capacitor (C2). As for the CE amplifier, these capacitors are used to prevent the DC voltages present at the base and emitter terminals from being affected by the signal source or the load. If the capacitor C1 was not present, the signal source would act as a resistor in parallel with R2 and alter the DC bias voltage to the base of Q1, affecting the rest of the DC conditions. If the load resistor RL was connected directly to the emitter of Q1, the DC voltage at the emitter would appear across the load. If the load were a motor or a loudspeaker, the DC voltage would lock the motor on, or cause the speaker to move to one end of its travel. Note that the value of the coupling capacitor C2 is relatively large, often in excess of 1000µF. As for the CE amplifier, the coupling capacitors are electrolytic types, requiring their polarity to be correct.

3. DC CONDITIONS

The method to derive the DC conditions for a CC amplifier is virtually the same as for the CE amplifier. However, the collector voltage now equals the supply voltage, and the emitter voltage should, ideally, equal half the supply voltage. Thus, the base voltage is 0.6V higher than half the supply voltage (for NPN). The calculations for the DC voltages for Fig.1 are shown below:

(a) $V_B = \frac{V_{CC} R2}{R1 + R2} = \frac{15 \times 27k}{22k + 27k} = \underline{8.3V}$

(b) $V_E = V_B - V_{be} = 8.3 - 0.6 = \underline{7.7V}$

(c) $I_E = I_C = \frac{V_E}{R_E} = \frac{7.7}{1k} = \underline{7.7mA}$

(d) $V_C = V_{CC} = \underline{15V}$

4. AC CONDITIONS

The AC characteristics of a CC transistor amplifier that will be described are:

- voltage gain
- input resistance
- output resistance (simplified equation only).

The circuit of Fig.2 shows the CC amplifier of Fig.1, with the relevant DC voltages and waveforms included for an input voltage (v_{in}) of 2Vp-p sinewave. Note the following about the waveforms:

- The voltage at the base terminal (8.3V) is isolated from the signal source. The input signal has no DC component, (AC only) and the voltage at the base (V_B) has both a DC and an AC component.
- The voltage at the output terminal again has no DC component, (isolated by C2), and the voltage at the emitter is an AC signal superimposed around a DC voltage of 7.7V.
- The output signal is in phase with, and has the same amplitude as the input signal.
- The waveform at the emitter is identical to that at the base, except the DC component has been reduced by 0.6V (Vbe).
- The height of all waveforms is shown as a peak to peak value and the values shown are with respect to the common line (ground).

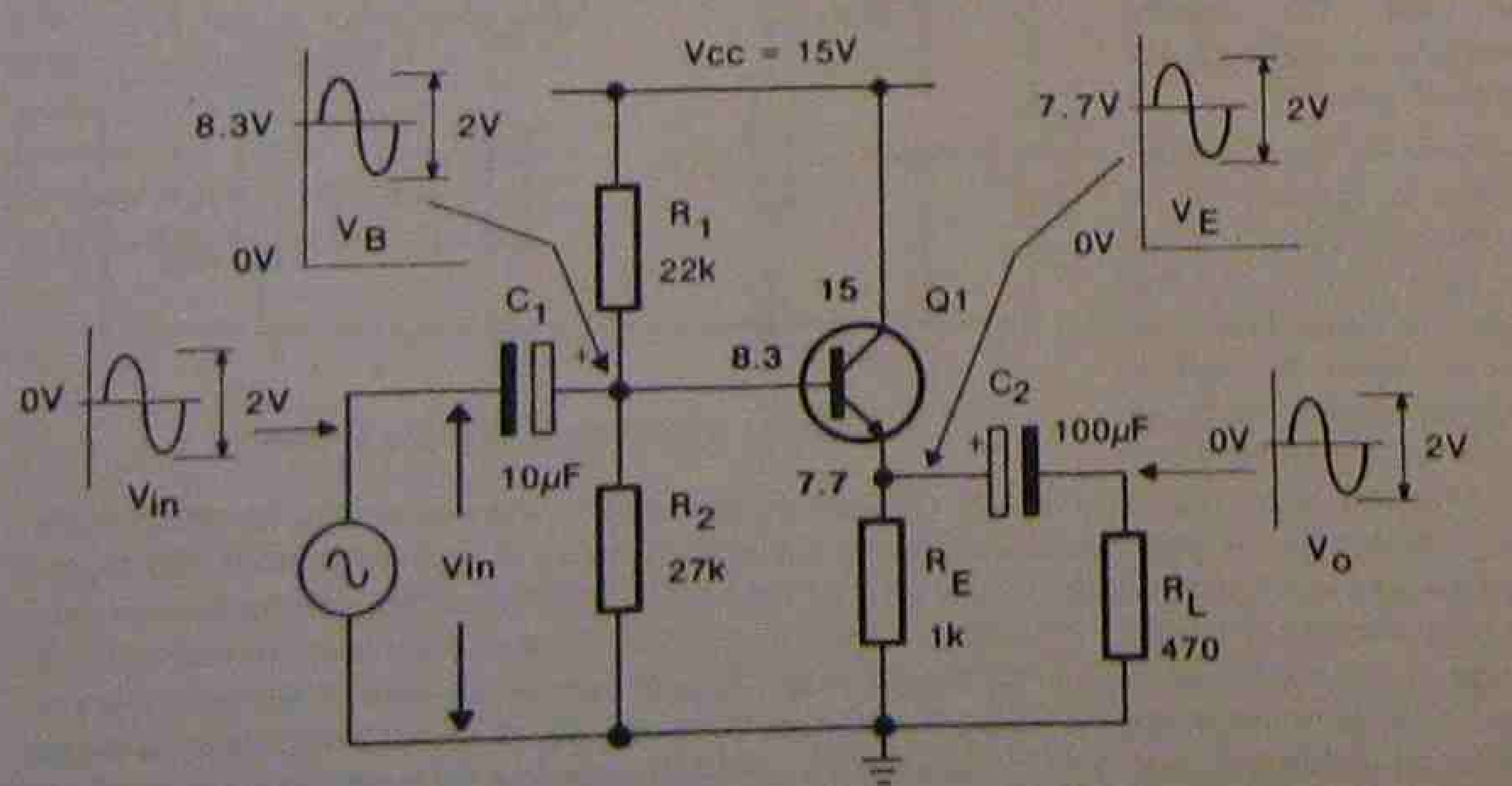


Fig.2: Waveforms for the common collector (CC) amplifier

5. VOLTAGE GAIN

As the waveforms of Fig.2 show, the output amplitude of a common collector amplifier is the same as the input. The principle of operation relies on the emitter terminal always being 0.6V lower than the base terminal (for an NPN transistor). If the voltage at the base increases by 1V as a result of the input signal, the emitter terminal will do the same. The common collector amplifier is therefore also known as an emitter follower. Thus, for an emitter follower (or common collector) amplifier gain Av:

□ A_v = unity

In fact, as a result of losses, the gain will be slightly less than unity, but for the purposes of these notes, the losses are ignored.

6. INPUT RESISTANCE

Input resistance for the CC amplifier is calculated in the same way as for the CE amplifier, except the load resistor needs to be included. The diagram of Fig.3 shows the equivalent circuit for calculating the input resistance. Note that the load is in parallel with the emitter resistor R_E.

As this diagram shows, the bias resistors R1 and R2 are in parallel, and the third path is through the base emitter junction, as for the CE amplifier. As before, any resistance after the base terminal is effectively multiplied by the current gain, (ß) of the transistor. For the CC amplifier, this resistance is r_e in series with the parallel combination of the emitter resistor R_E and the load R_L. The equation for calculating input resistance for the common collector amplifier is:

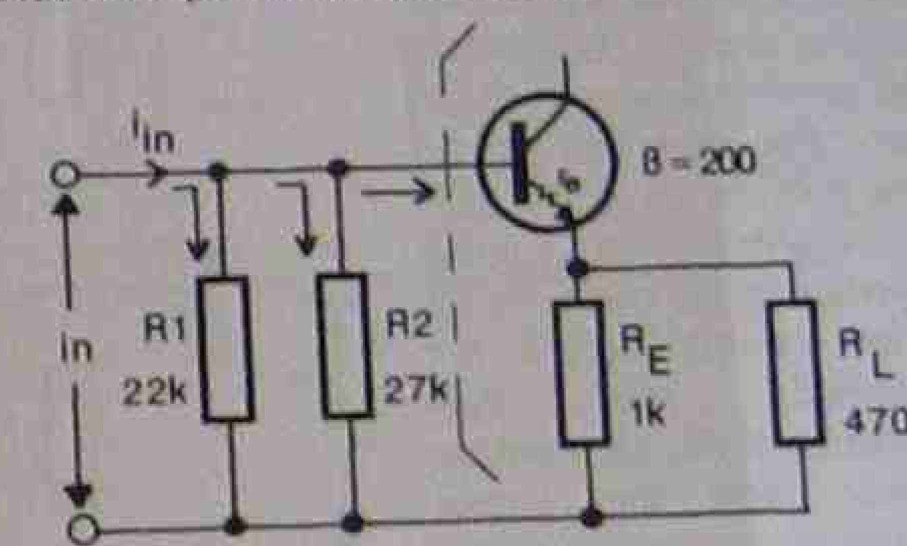


Fig.3: equivalent circuit for input resistance

□ $r_{in} = R1//R2//[ß(r_e + R_E//R_L)]$

where: □ $r_e = \frac{30mV}{I_C}$

7. OUTPUT RESISTANCE

The output resistance of a the common collector amplifier is rather complex to calculate, and as an approximation, it can be shown that this equals r_e. That is:

□ $r_o = r_e$

8. WORKED EXAMPLE

For the circuit of Fig.2, calculate:

(a) r_e

(b) the gain of the circuit.

(c) the input resistance.

(d) the output resistance.

Solution:

(a) $r_e = \frac{30mV}{I_C} = \frac{30mV}{7.7mA} = \underline{3.9\ ohms}$

(b) $A_v = \underline{1}$

(c) $r_{in} = R1//R2//[ß(r_e + R_E//R_L)]$
$= 22k//27k//[200(3.9 + 1k//470)]$
$= 12k1//[64k7] = \underline{10k2\ ohms}$

(d) $r_o = r_e = \underline{3.9\ ohms}$

9. APPLICATIONS OF THE CC AMPLIFIER

As the worked example shows, the input resistance of a CC amplifier is largely determined by the values of the biasing resistors. If these resistors are 100k or more, the input resistance can be relatively high. As the example also shows, the output resistance of the CC amplifier is very low, and it is because of its high input resistance and low output resistance that the CC amplifier finds application. Its main use is as a **buffer**. Typical applications are to use a CC amplifier before a CE amplifier to obtain a high input resistance, and then to connect a CC amplifier to the output of the CE amplifier to get a low output resistance. This way, the whole circuit has gain, a high input resistance and a low output resistance, which approaches the ideal characteristics of an amplifier.

10. THE COMMON BASE AMPLIFIER

The third configuration for a transistor amplifier is the common base (CB) connection, as shown in Fig.4. This amplifier has the input signal applied to the emitter and the output is taken from the collector. The base terminal is connected to bias resistors, and a capacitor is often connected across R2 to ensure the base has no AC signal present. Thus, the base is regarded as being connected to the common line as far as AC signals are concerned. The input impedance of the CB amplifier is low, and the output impedance equals the collector resistor Rc. Unlike the CC amplifier, the CB configuration has voltage gain, but no current gain.

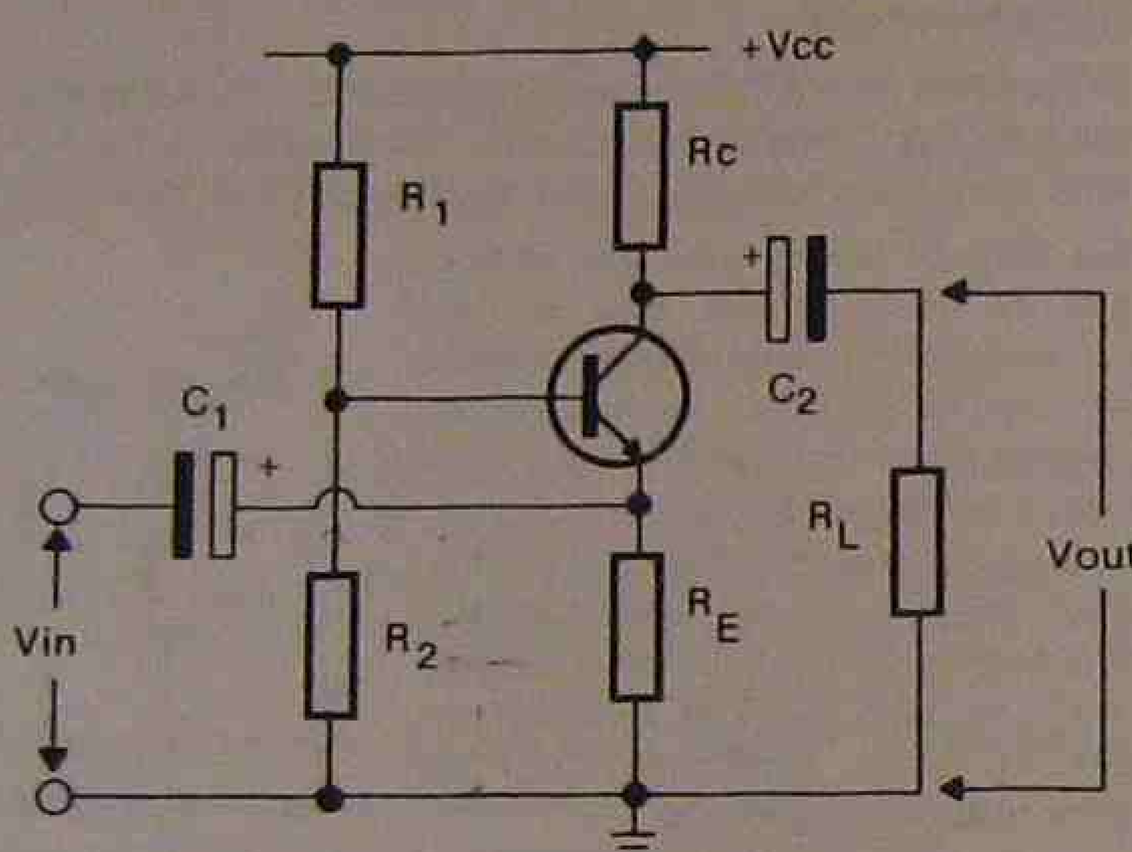


Fig.4: The common base amplifier

11. COMPARISON OF THE CE, CC AND CB AMPLIFIERS

The following table lists the various characteristics fo the CE, CC and CB amplifier. Because the CB connection has limited application, it is included for comparison purposes, but knowledge of its characteristics is not essential.

Configuration	input resistance (rin)	output resistance (ro)	phase shift I/P to O/P (degrees)	voltage gain (Av)	current gain (Ai)	power gain (Ap)
CE	med	med (= Rc)	180	high	med	high
CC	high	low (= re)	0	unity	high	high
CB	low	med (= Rc)	0	med	unity	med

The terms low, med and high are relative, and indicate a relationship between the various configurations. That is, a CC amplifier has a high input resistance compared to a CE amplifier.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 6

1. For the circuit of Fig.1, calculate the following:

(a) V_B and V_E

(b) I_C

(c) r_e

(d) Voltage gain Av

(e) input resistance r_{in}

(f) output resistance r_o

(g) output voltage v_o

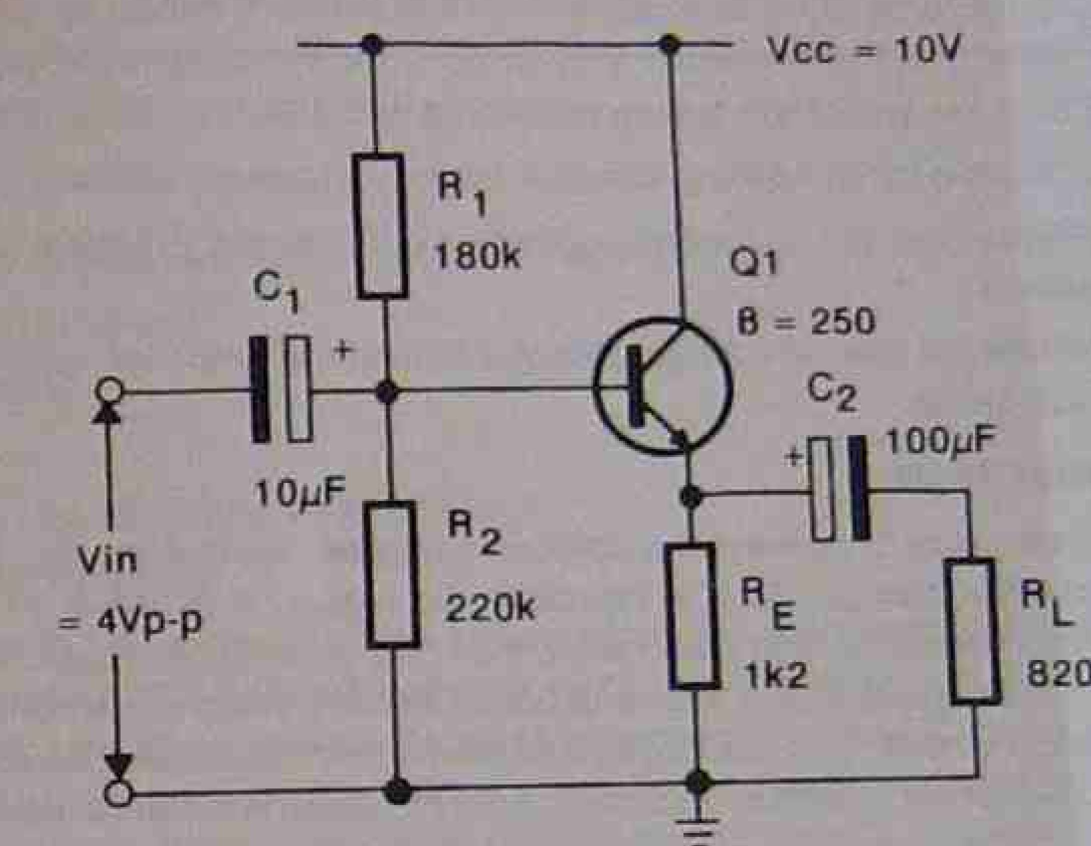
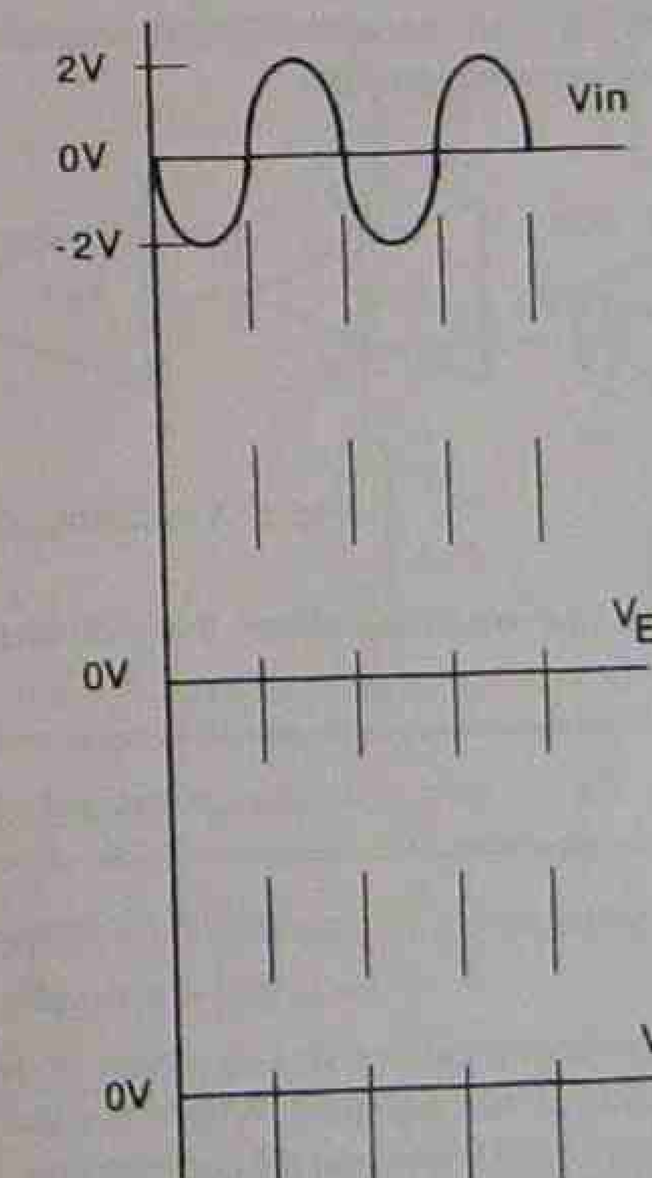


Fig.1

2. Using the values obtained in question 1, sketch the waveforms that should be present at the emitter and across the load resistor RL. Include any DC component and show the correct phase relationship these waveforms will have to the input signal v_{in}. It is not necessary to draw the waveforms to scale, but all DC voltages and the peak to peak voltage values of the waveforms should be shown.



****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 8

References: Electronic Devices, 2nd Ed. Floyd. Chapter 8.

MULTISTAGE AMPLIFIERS

OBJECTIVES At the end of this lesson you will be able to:

- List reasons for combining amplifier stages to form a multistage amplifier.
- Calculate the DC quiescent conditions for direct coupled and RC coupled multistage amplifiers.
- Calculate the overall voltage gain of a multistage amplifier, given the individual voltage gains of each stage.

1. INTRODUCTION

Most industrial or commercial amplifiers comprise several stages. Although a common emitter (CE) amplifier can produce a relatively high gain, it is usual to combine two or more CE amplifiers to obtain a high gain rather than rely on a single stage. While more components are used, a multistage amplifier will have better stability and a more controlled gain than a high gain single stage circuit. It is also typical to use a common collector (CC) amplifier in conjunction with a CE amplifier to obtain improved input or output resistance values.

These notes describe how individual amplifier stages are connected and how the DC conditions and the overall gain are calculated. Two methods of connecting individual stages are also described.

2. MULTISTAGE AMPLIFIERS

The block diagram of Fig.1 shows a multistage amplifier comprising n stages, in which each stage is 'cascaded' to a preceding stage.

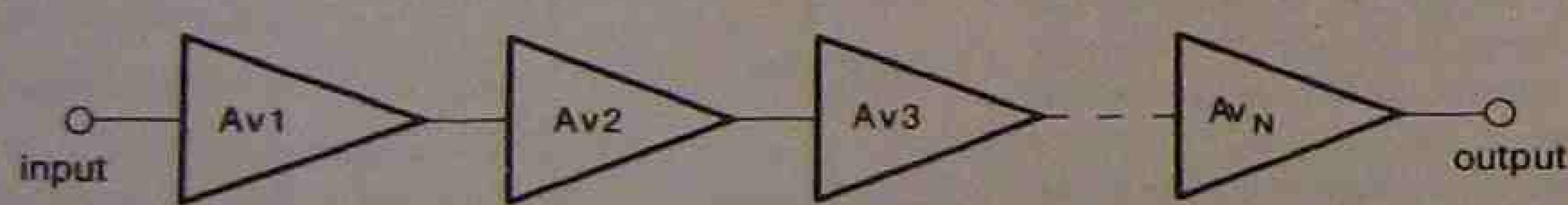


Fig.1: A multistage amplifier

To calculate the gain of the whole amplifier, the individual gains of each stage are multiplied. That is, overall gain:

$$A_{vT} = A_{v1} \times A_{v2} \times A_{v3} \times \ldots A_{vN}$$

It is common to express the gain of an amplifier in decibels (dB) where:

$$A_v \text{ in dB} = 20\log A_v$$

For example, an amplifier with a gain of 10 has a gain in decibels of 20log10, or 20dB. A gain of 100 gives 40dB, as the log of 100 equals 2. Where the gain is expressed in dBs, the overall gain of the amplifier is the sum of the individual dB gain figures.

$$A_{vT}\text{(in dB)} = \text{dB1} + \text{dB2} + \text{dB3} + \ldots \text{dBN}$$

3. REASONS FOR CASCADING AMPLIFIER STAGES

Amplifier stages are cascaded for various reasons, including:

- to give a higher gain than available from a single stage. This would involve cascading several CE amplifiers. If the individual gain of each stage in a three stage voltage amplifier is 5, the overall gain will equal 5 x 5 x 5, which equals 125.
- to obtain a high input resistance and a low output resistance. In this instance, a CC amplifier would precede a CE amplifier to obtain a high input resistance, and a CC amplifier would follow the CE stage to give a low output resistance. Because a CC amplifier has a gain of 1, the overall gain will equal the voltage gain of the CE amplifier.

4. RC COUPLED AMPLIFIERS

There are various methods used to couple amplifier stages including:

- Resistance-Capacitance (RC) coupling
- Direct coupling or DC coupling
- Transformer coupling
- Tuned transformer coupling (eg. radio and TV amplifiers)
- Inductance-Capacitance (LC) coupling

The simplest form of coupling is RC coupling, in which individual stages are joined together with capacitors. Each stage is therefore DC isolated from the next, making design of the circuit relatively simple. However, this type of coupling uses more components than DC coupling, as the bias voltages for each stage need to be derived with the usual potential divider network, and a capacitor is required between each stage. The circuit of Fig.2 shows two CE amplifier stages connected via capacitor C2.

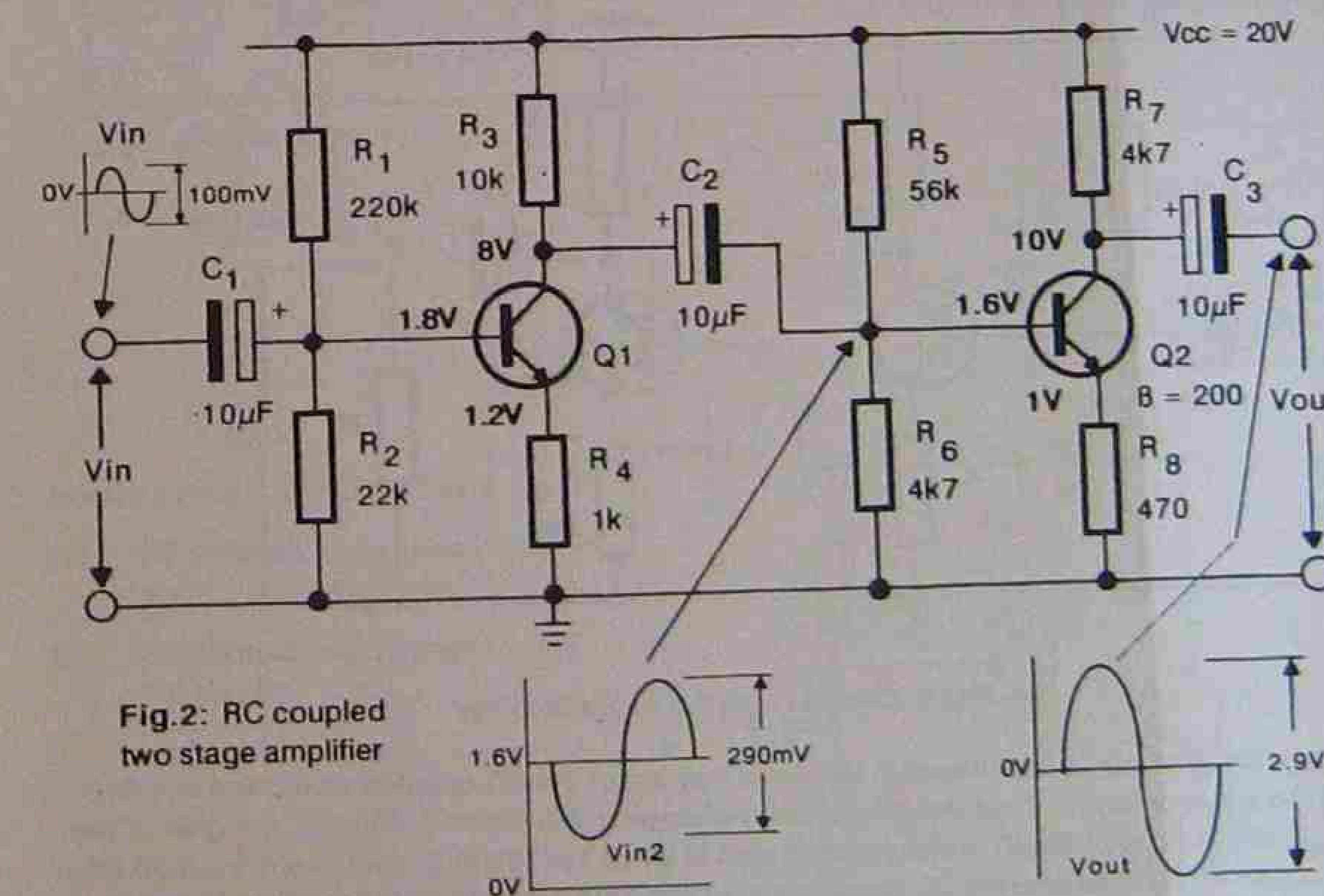


Fig.2: RC coupled two stage amplifier

The DC voltages for each stage are determined using the equations described in week 5 and are shown on the diagram.

The waveforms for the circuit are also shown, with the amplitudes calculated for an input signal of 100mV. Note the following about these waveforms:

- The gain of the first stage is 2.9, giving an output of 290mV. This amplifier stage was described in Week 5, in which the gain was shown to be 9.7. However, this gain was calculated with the amplifier not connected to a load. In this application, the input resistance of the second stage represents a load, and the gain of the first stage is therefore reduced.
- The gain of the second stage is 10, giving a total gain for the circuit of 29.
- The output waveform is in phase with the input, as the total phase inversion is 180° + 180°, or 360°.

If a bypass capacitor was connected across either, or both of the emitter resistors, the gain of the circuit would rise substantially. For example, if a bypass capacitor was connected across R8, the gain of the second stage would equal R7/re2, giving a gain of 4k7/15, or 313. However, the input resistance of this stage would fall, lowering the gain of the first stage. However, the overall gain would still be around 600.

<u>NOTE</u>: For the purposes of this course, the individual stage gains will not be calculated from the circuit values, but only from signal amplitudes. Thus, the equation Av = vo/vin is used to determine individual stage gains and the overall gain of the circuit. However, the DC values are calculated as previously described, as the coupling capacitor prevents any interaction between the stages.

5. THE DIRECT COUPLED AMPLIFIER

The circuit of Fig.3 shows a two stage amplifier consisting of two direct coupled CE amplifiers. This circuit is simpler than the RC coupled amplifier, in that less components are required. However, the DC conditions for the circuit are determined by R1 and R2, and a fault in the first stage will alter the DC voltages in the second, making fault finding more difficult. Because there are no biasing resistors required for the second stage, the input resistance of this stage is higher than if it were RC coupled. Thus, for given values of collector and emitter resistors, the direct coupled amplifier has a higher gain.

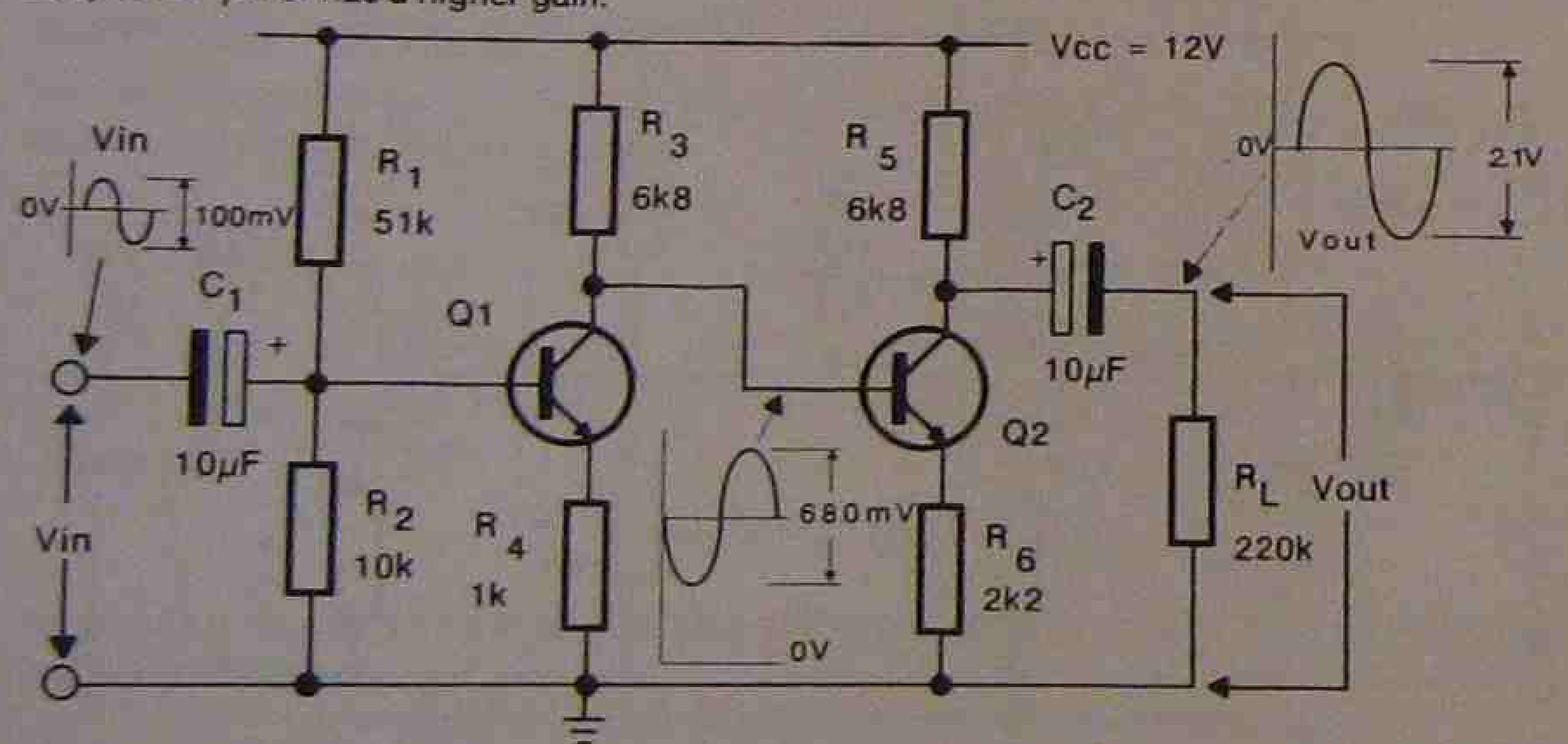


Fig.3: Direct coupled, 2 stage amplifier

The gain of the circuit is determined in the same way as for the RC coupled multistage amplifier using the equation vo/vin. Thus the gain of the first stage is 6.8 (680mV/100mV), the gain of the second is 3.08 (2.1V/680mV) and the overall gain is 21 (2.1V/100mV). Note that 6.8 x 3.08 also gives a gain of 21. Note also the DC component of the input signal to the second stage and that the phase relationship for each waveform is identical to that for the RC coupled amplifier

The DC conditions for the amplifier are calculated as for the RC coupled amplifier where:

V_{B1} = 1.96V, V_{E1} = 1.36V, I_{C1} = 1.36mA, V_{C1} = V_{B2} = 2.8V, V_{E2} = 2.2V, I_{C2} = 1mA, V_{C2} = 5.8V.

****

THEORY ASSIGNMENT 8

1. For the circuit of Fig.1, calculate the following:
   (a) DC voltages at the base, emitter and collector of Q1 and Q2.
   (b) voltage gain of first stage, from the waveforms shown.
   (c) voltage gain of second stage, from the waveforms.
   (d) overall gain of the circuit.

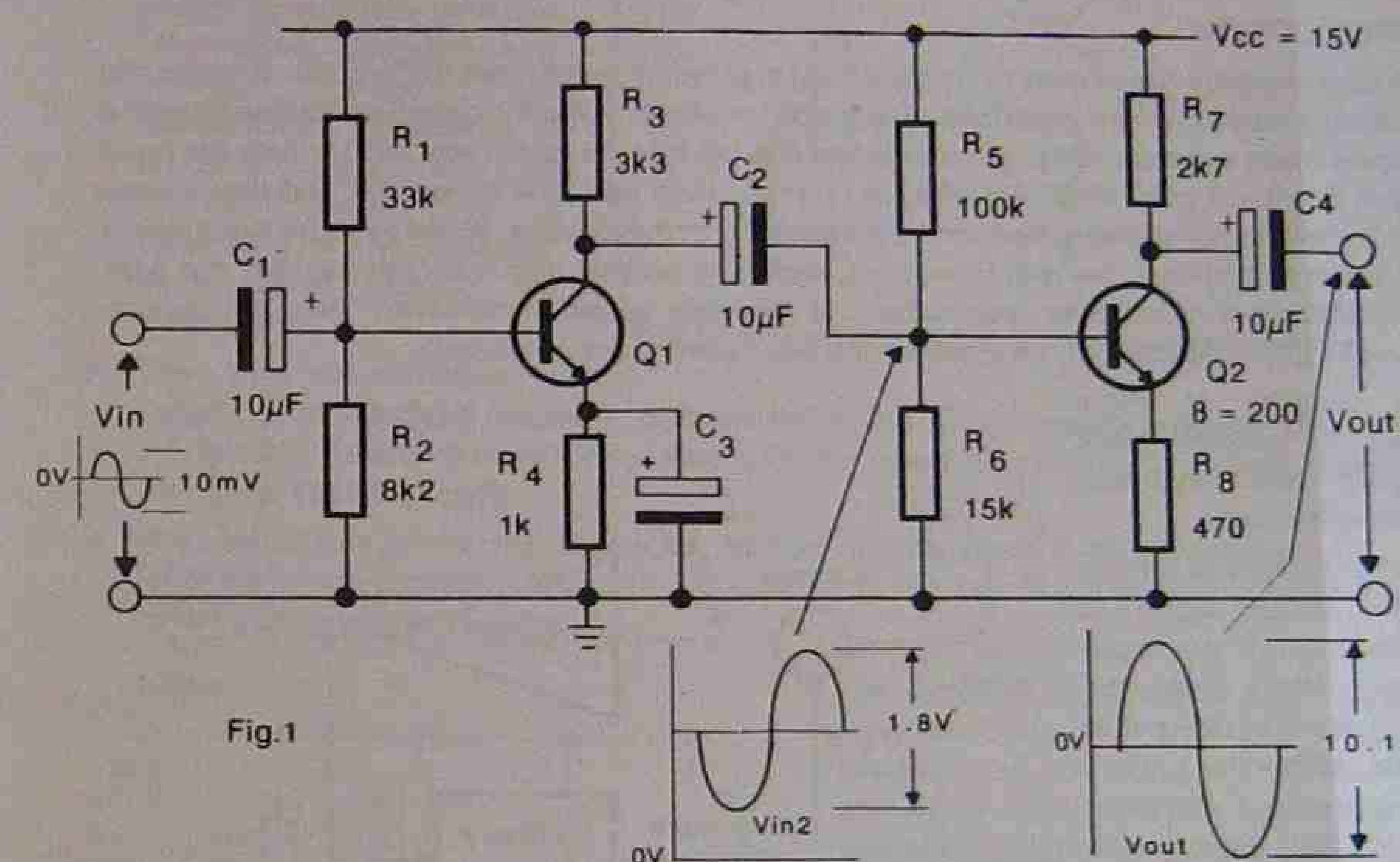


Fig.1

2. For the circuit of Fig.2, calculate:
   (a) DC voltages at the base of Q1 and Q2 and the emitter of Q2.
   (b) the approximate voltage gain of the circuit.

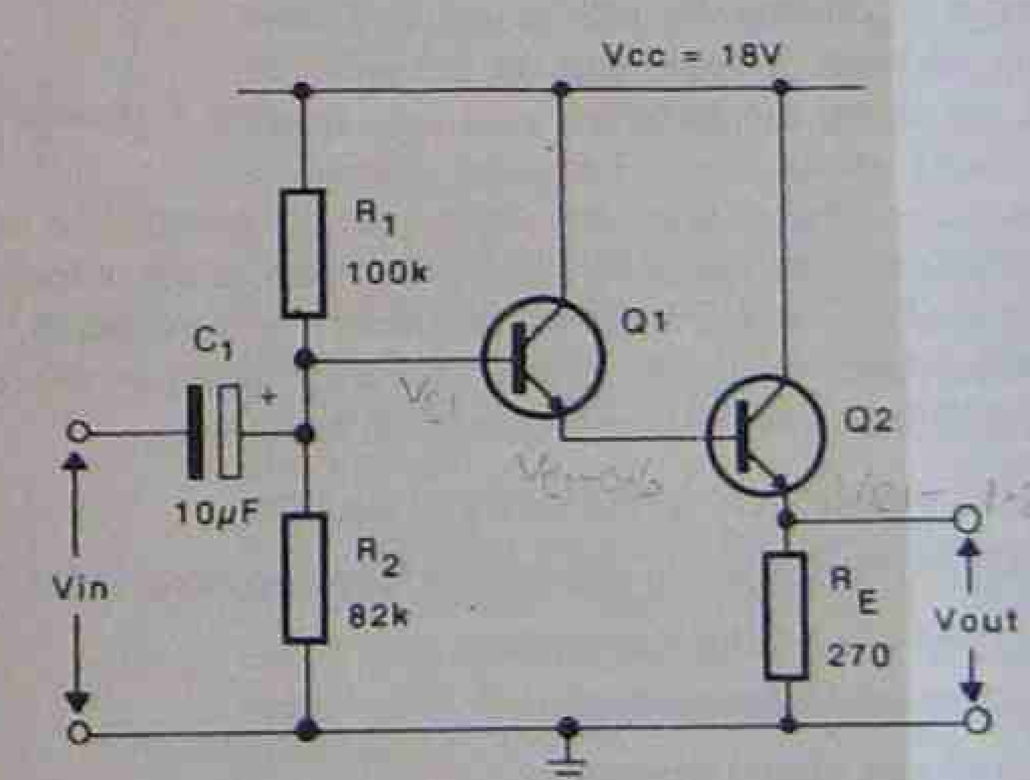


Fig.2

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 9

References: Electronic Devices, 2nd Ed. Floyd, Chapter 12 and 13.

FEEDBACK

OBJECTIVES At the end of this lesson you will be able to:

- List the benefits of using negative feedback in an amplifier.
- Identify the components providing AC and DC negative feedback in an amplifier circuit.
- Calculate the gain of an amplifier with voltage derived negative feedback.

1. INTRODUCTION

The term feedback has already been mentioned in previous notes when the benefits of an emitter stabilising resistor where described. Feedback, or more correctly, **negative** feedback is the process where a sample of the output is fedback to the input in such a way as to reduce the input signal. Feedback in an audio amplifier, or in any system has several benefits, and these along with the method of applying feedback are described in these notes. When negative feedback is applied to an amplifier, the gain is reduced, and if the original gain was high enough, the gain with feedback is much more predictable and generally easier to calculate. There are various ways of applying feedback in an amplifier and two methods are described.

The diagram of Fig.1 shows a general block diagram that illustrates negative feedback.

In this diagram, a sample of the output signal is combined with the input signal. Note that the output signal is 180° out of phase with the input, giving an input signal to the amplifier that is less than Vin. This signal is the result of adding Vin with the feedback voltage, and is the signal the amplifier accepts as its input. The feedback network is usually either a single resistor or a combination of several that produces a voltage that has the same shape and phase as the output voltage, but smaller in amplitude. The summer can be the base-emitter junction of a transistor, or some form of circuit that combines the feedback voltage and the input voltage giving an effective input signal to the amplifier that is smaller than it would be if there was no feedback.

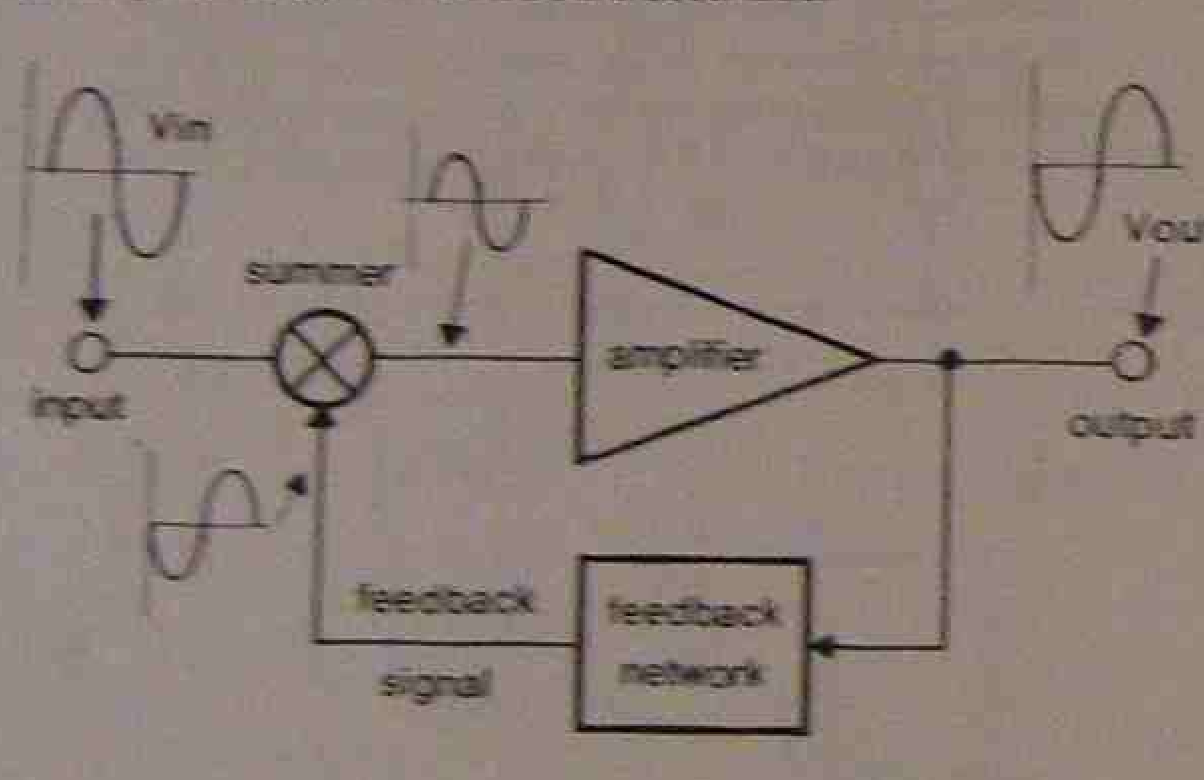


Fig.1: general form of negative feedback

2. ADVANTAGES OF NEGATIVE FEEDBACK

Negative feedback can provide the following advantages:

- reduced, but stabilised gain.
- increased input resistance
- reduced output resistance
- less noise in the output signal
- less distortion in the output signal
- wider frequency response

3. EMITTER RESISTOR AS FEEDBACK

As already described, if an emitter resistor is included in the circuit of a common emitter amplifier, as shown in Fig.2, the gain is reduced unless this resistor is bypassed with a suitable capacitor. This resistor produces negative feedback in the amplifier in which:

- the feedback voltage is developed by the emitter current flowing in RE and appears at the emitter as an in-phase signal with the input signal.
- the summer is the base-emitter junction.
- because the voltage at the emitter is in phase with the input signal, the difference voltage across the base-emitter junction is reduced. That is, the combined effect is a reduction of the effective input signal across the base-emitter.

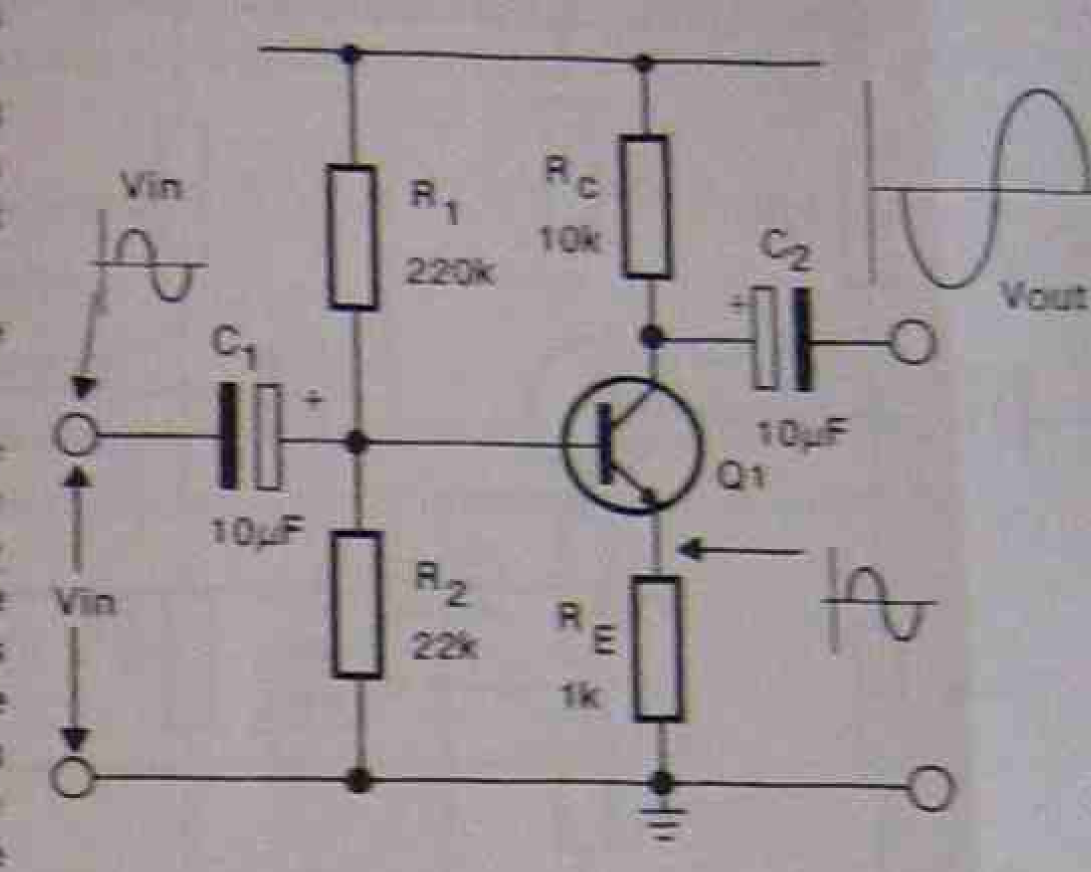


Fig.2: AC and DC feedback with an emitter resistor

- This type of feedback is called **current-derived** feedback, because the emitter (which equals collector) current has produced the feedback signal. Because the emitter resistor is relatively large compared to r_e, the gain of Fig.2 is $(R_C)/R_E$.
- If a capacitor is connected across the emitter resistor, there is no AC feedback, but DC feedback is still provided. Thus, the DC stability of the circuit is improved compared to a circuit with no emitter resistor.

The circuit of Fig.3 shows a DC coupled amplifier with AC and DC feedback provided by R5. Because the waveform across R5 equals Vin, the output voltage can be calculated using Ohm's law in which the AC (and DC voltage) across R4 will be twice that across R5 (as R4 = 2k and R5 = 1k. Thus the AC output equals $V_{R5} + V_{R4}$ which equals 3 x Vin. The gain of the circuit is therefore three.

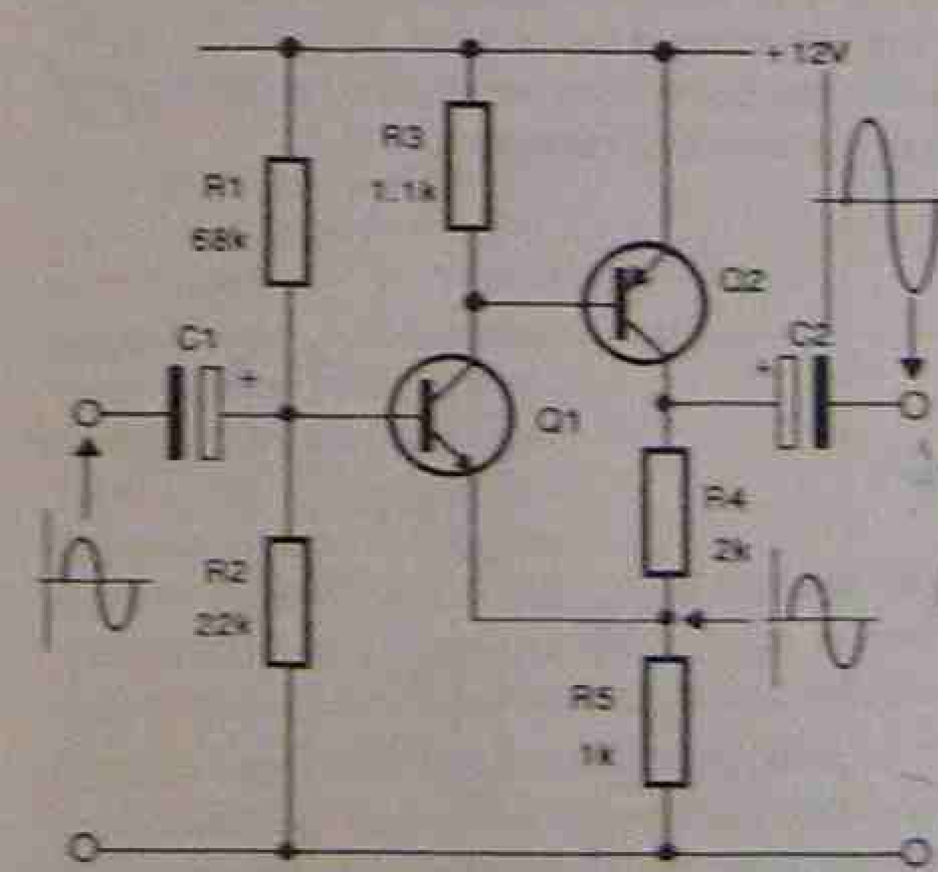


Fig.3: DC and AC feedback

Resistor R5 is part of the collector load for Q2, but is also the emitter resistor for Q1. It therefore also provides DC feedback for Q1, stabilising the DC conditions for the whole circuit.

Note that both stages of this amplifier are common emitter types, because in both cases the input signal is applied to the base, and the output taken from the collector.

As for Fig.2, the feedback is current derived and the summer is the base-emitter of Q1.

4. VOLTAGE DERIVED FEEDBACK

The circuit of Fig.4 shows feedback around a two stage RC coupled amplifier. Both stages of the amplifier are common emitter, and the feedback components are R1 and Rf.

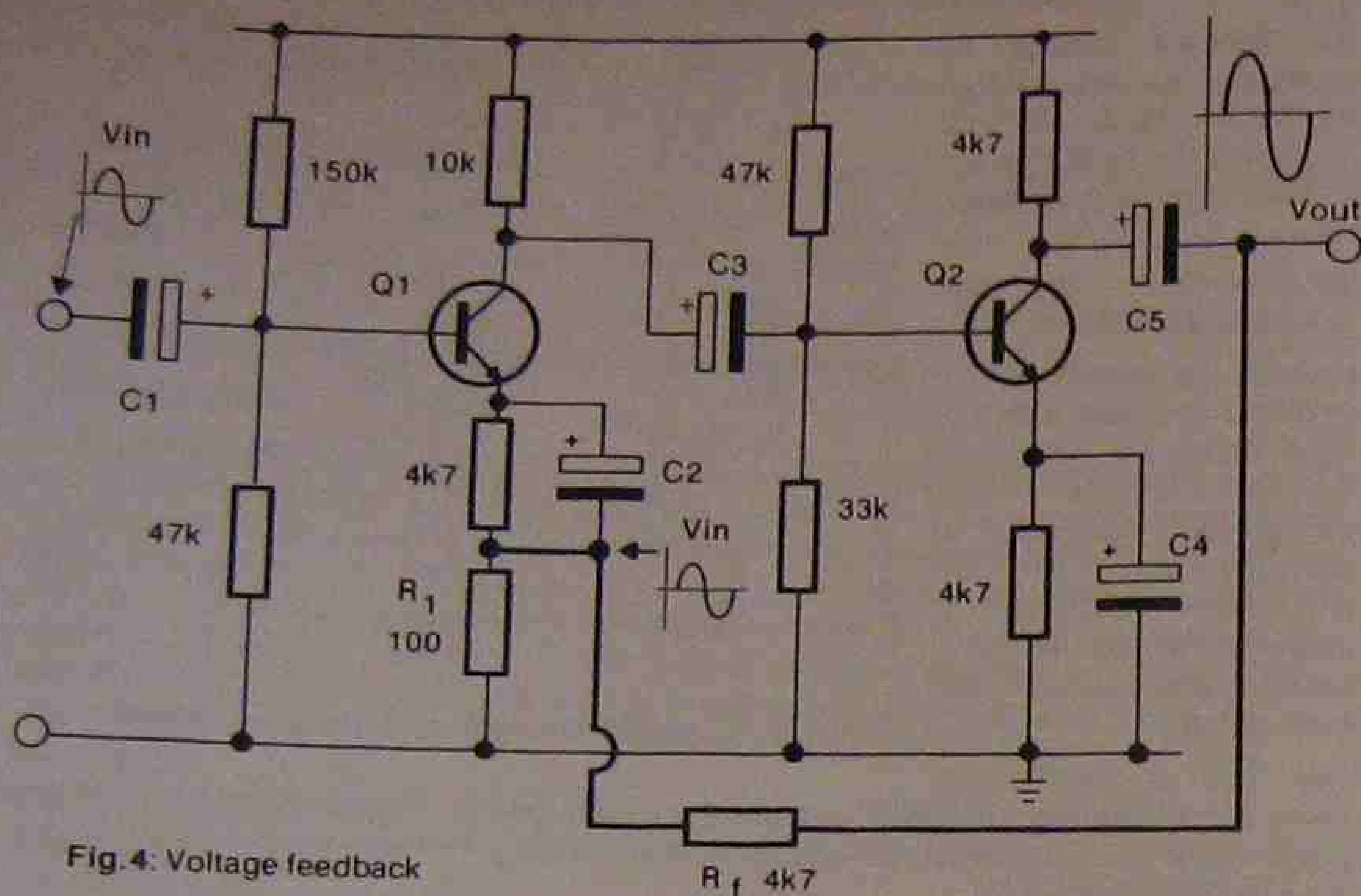


Fig.4: Voltage feedback

The feedback resistor Rf is connected to the output, DC isolated by C5. This resistor then connects to R1, which is part of the emitter circuit of Q1. Capacitor C2 bypasses the 4k7 resistor connected to the emitter of Q1, and R1 is not bypassed. As shown, by emitter follower action, the waveform across R1 is equal in amplitude to Vin. This is an approximation, as in fact it must be slightly less due to the presence of r_e for Q1. For the purposes of these notes, the effect of r_e is ignored. Because the AC signal across R1 equals Vin, the gain of the circuit can be determined with Ohm's law. Fig. 5 shows the equivalent circuit of the feedback network for Fig.4.

Solution:

$$I_{R1} = \frac{Vin}{R1} = I_{Rf}$$

$$V_{Rf} = I_{Rf} V_{Rf} = \frac{(Vin)\,Rf}{R1}$$

Vout = sum of voltages across Rf and R1:

$$Vout = \frac{(Vin)\,Rf}{R1} + Vin$$

dividing both sides by Vin:

$$\square \quad \frac{Vout}{Vin} = \frac{Rf}{R1} + 1 = \text{gain}$$

$$\text{Gain of Fig.4} = \frac{4k7}{100} + 1 = \underline{48}$$

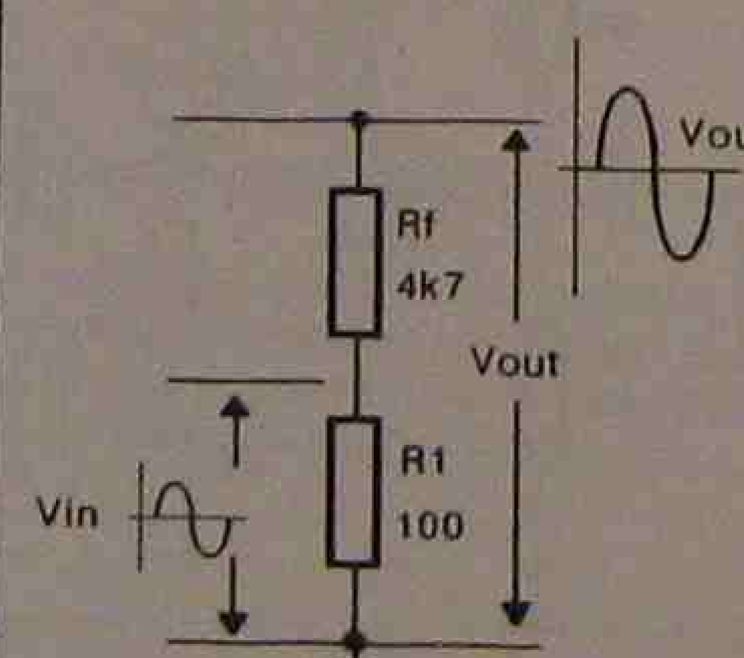


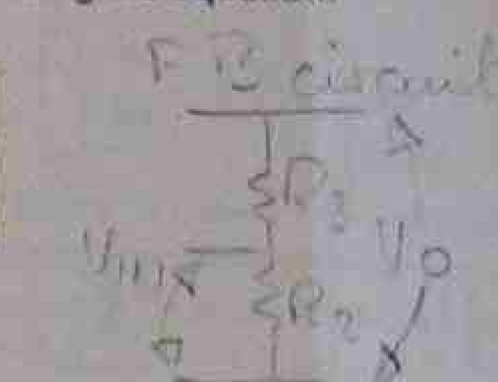
Fig.5: Equivalent circuit of feedback network of Fig.4.

Thus, the feedback components alone have determined the gain of the amplifier.

Note the following for the circuit of Fig.4:

- Resistors Rf and R1 form a potential divider network.
- The feedback provided by Rf and R1 is AC only, as the output is DC isolated by C5
- DC feedback is provided by the resistors in the emitter circuits of both Q1 and Q2. Because these resistors are bypassed (except R1), these resistors do not produce any AC feedback.
- This type of feedback will:
  - lower the circuit gain and stabilise it
  - increase the input resistance
  - decrease the output resistance
  - provide all the benefits of negative feedback listed at the start of these notes.
- for this type of circuit, the gain with feedback (A_{vf}) is calculated using the equation:

$$\square \quad A_{vf} = \frac{Rf}{R1} + 1$$



5. EXAMPLE OF FEEDBACK

The circuit of Fig.6 shows a commercial design of a direct coupled, two stage amplifier with AC and DC feedback. The original circuit includes several noise suppression capacitors that to simplify the circuit, have been excluded from the diagram of Fig.6.

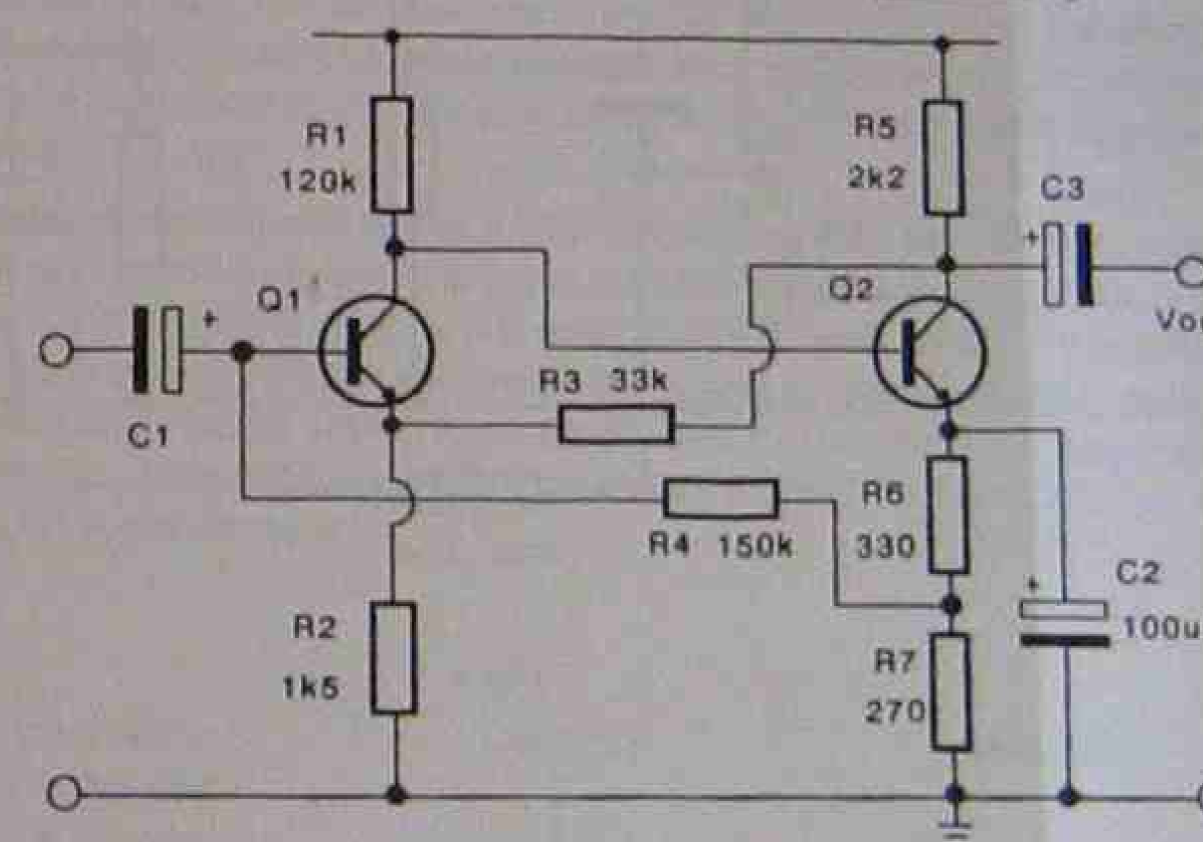


Fig.6: example of AC and DC feedback

An important feature of the circuit is how the DC conditions are established. The base bias voltage for Q1 is provided by the voltage across R7, supplied to the base of Q1 via R4. The emitter resistor of Q1, (R2) helps stabilise the DC voltages, but the whole circuit also forms a closed loop to ensure the DC voltage at the collector of Q2 is held to a fixed value. Bypass capacitor C2 eliminates any AC component from the DC voltage bias voltage to Q1.

Voltage derived AC feedback is applied to the emitter of Q1 with a network similar to that described for Fig.5. Thus, using the equation for closed loop gain, the gain equals (R3/R2) + 1, giving a voltage gain of 23. In this case, because R3 connects directly to the collector of Q2, DC feedback is also provided by R3.

****

THEORY ASSIGNMENT 9

For the circuit of Fig.1, calculate the following:

(a) DC voltages at the base, emitter and collector of Q1 and Q2.

(b) closed loop gain of the circuit.

(c) the output voltage if the input voltage is 200mVp-p.

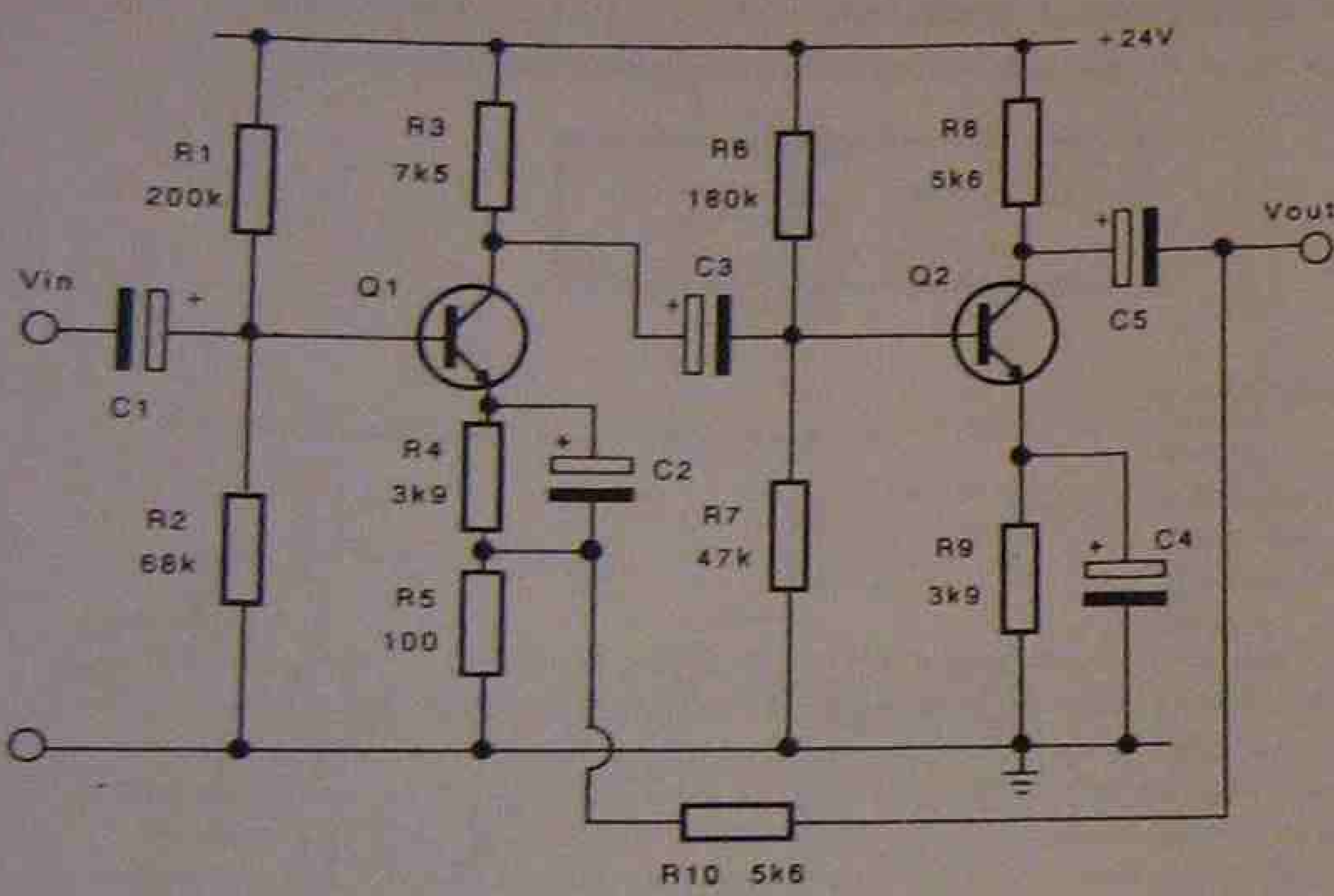


Fig.1

State the reason for using DC feedback in an amplifier circuit.

List the advantages ofAC negative feedback in an amplifier circuit.

****

THEORY LESSON 10

References: Electronic Devices, 2nd Ed. Floyd. Chapter 9.

FIELD EFFECT TRANSISTORS

OBJECTIVES At the end of this lesson you will be able to:

- List the DC operating characteristics of the Junction FET (JFET).
- List the DC operating characteristics of the Metal Oxide Silicon FET (MOSFET).
- Calculate the value of the bias resistor and drain resistor for an N channel JFET using the transfer curve for that JFET.
- Sketch the basic circuit configuration for a JFET amplifier.

1. INTRODUCTION

The field effect transistor is a relatively simple device that has, in effect, a single PN junction. However in most applications, this junction is operated in reverse bias. Thus unlike the BJT, the FET is a voltage operated device and requires a different circuit configuration to function. Like the transistor, the P and N type materials can be reversed, giving the so called P type and the N type FETS. As well, there are two basic FET families; the Junction FET (JFET) and the Metal Oxide Silicon FET (MOSFET). A MOSFET can be either a 'depletion' type or an 'enhancement' type. There are therefore six symbols used to describe all these FETs; three for the N channel types and three for the P channel.

To calculate the values of the resistors required within a FET amplifier circuit, the characteristic curves of the FET are used. It is possible to determine these resistor values with equations, but for the purposes of these notes, the graphical approach is used as it is simpler. There are certain fundamental characteristics applicable to FETs, and these are listed in manufacturers' data. These characteristics are described, along with the method used to calculate the required resistance values in a FET amplifier. These notes concentrate mainly on the N channel JFET, but a brief description is provided for the other types.

2. THE JFET

The basic structure of a JFET is shown in Fig.1, in which a PN junction is formed between the gate section and a channel made of doped silicon. In the N channel JFET, the channel is a piece of N doped silicon and the gate section is a piece of P doped silicon. The P channel JFET has a P type channel, and the gate section is N doped. The two terminals connected to either end of the channel are referred to as the drain and the source. The schematic symbols are shown in Fig.2.

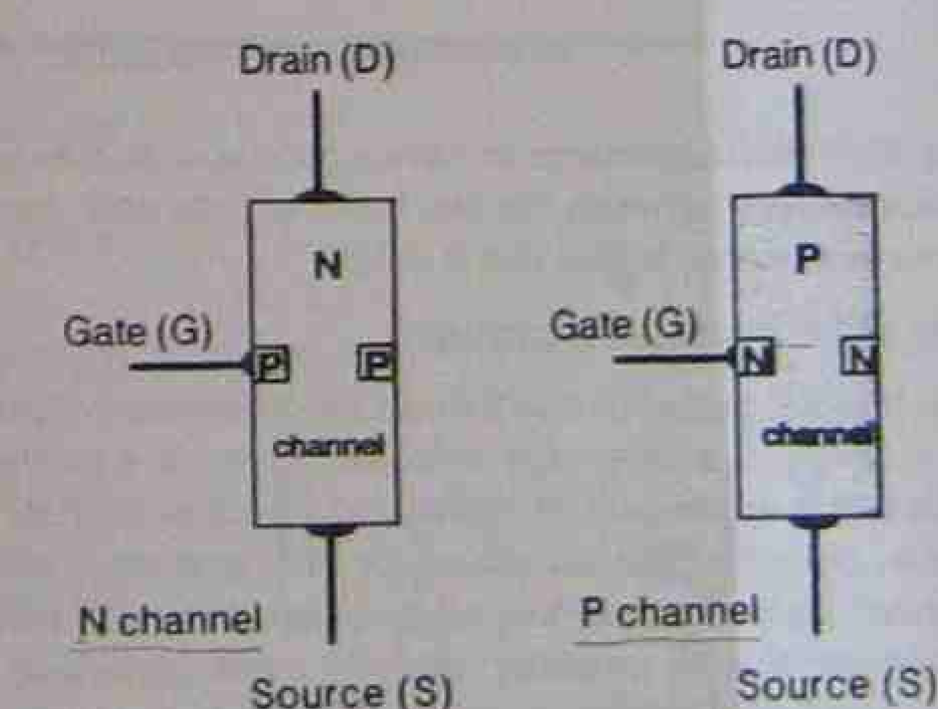


Fig.1: structure of the JFET

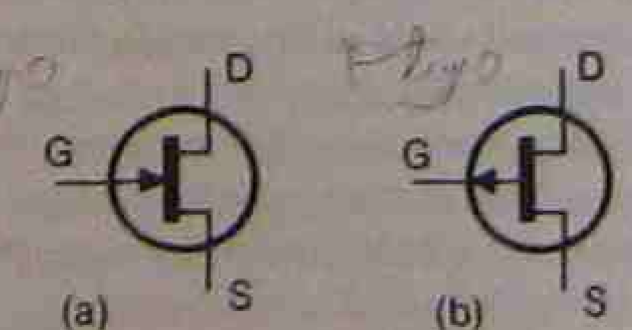


Fig.2 (a) N channel, (b) P channel

3. JFET OPERATION

A JFET is biased so that the voltage between the gate and the source reverse biases the gate-source junction. For an N channel JFET, the gate will be negative compared to the source, and the opposite polarities apply for the P channel. The required polarities for the N channel JFET are shown in Fig.3. Because the gate-source junction is reverse biased, a voltage field is formed around the junction. The higher the value of Vgs, the greater the depth of the field and the higher the resistance offered to current flowing from the drain to the source terminal. Thus, the voltage between the gate and source terminals of the JFET controls the current (I_D) flowing through the channel.

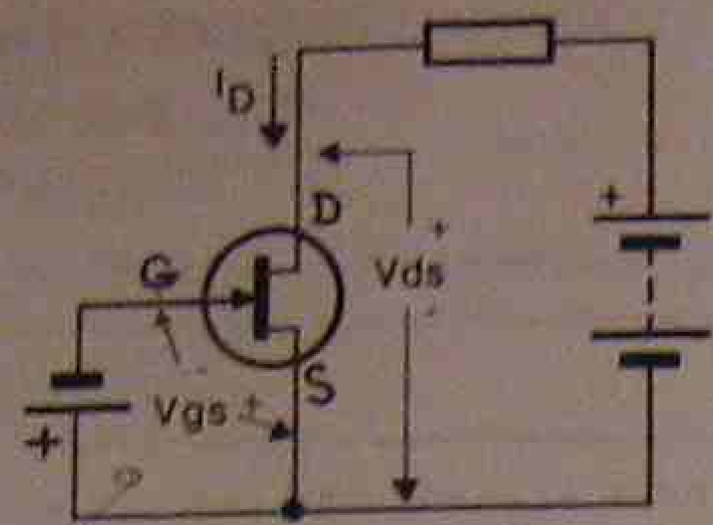


Fig.3: Voltage polarities for the N channel JFET

The effect of the field is shown in Fig.4. In (a), the field is small enough not to impede the flow of current significantly and (b) shows how the field has closed the available path thus restricting I_D. From this, there are two important characteristics inherent in JFETS:

- the current that flows when Vgs equals zero. This value of current is limited only by the FET and not by the gate voltage and is referred to as I_{DSS}.
- the value of Vgs that causes I_D to drop to zero. This voltage is usually referred to as **Vgs off**.

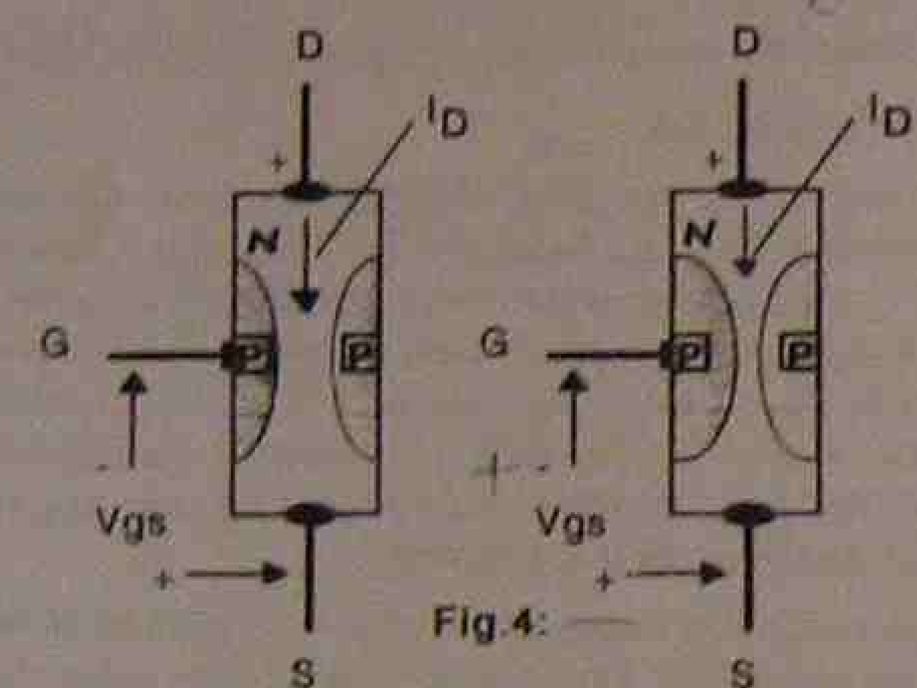


Fig.4: (a) small Vgs, large I_D (b) large Vgs, small I_D

> ☐ **I_{DSS} = drain current when V_{GS} equals 0**
>
> ☐ **$V_{GS\,OFF}$ = gate source voltage to cause I_D to drop to zero**

The JFET therefore needs to have a gate-source bias voltage that gives a quiesent drain current I_D somewhere between the two limits of I_{DSS} and zero. To determine the voltage, the transfer curve that relates I_D and V_{GS} is used.

THE JFET TRANSFER CURVE

The transfer curve for a JFET can be determined experimentally by measuring its drain current for a range of gate-source voltages. There is a mathematical relationship between these two values that applies to all FETs, and thus the transfer curve for a JFET always has the same shape, but with different values for I_{DSS} and $V_{GS\,OFF}$. An important feature to note about the transfer curve is that the relationship between the drain current and the gate-source is non-linear. For an amplifier, this will mean distortion unless the FET is operated only over a linear portion of the curve. This is achieved by selecting a quiescent value of Vgs that is central to the most linear part of the curve and ensuring that the input signal doesn't cause Vgs to operate the FET on the non-linear part of the curve. For the purposes of these notes, the quiescent point is chosen as being that value of Vgs that gives a drain current equal to half I_{DSS}. However, this is not necessarily the optimum operating point, but it makes calculations easier. The transfer curve for a JFET is shown in Fig.5.

For the transfer curve of Fig.5, note the following:

- I_{DSS} is 10mA in this example.
- the operating point has been chosen as $I_{DSS}/2$, = 5mA.
- from the transfer curve for this particular JFET, a gate-source voltage of -1V is required.
- the value of $V_{GS\,OFF}$ is around -4.7V in this example.
- the shaded portions show the range of operation to keep the JFET on the linear part of its transfer curve. Thus Vgs cannot vary outside the limits shown if distortion (or non-linear operation) is to be avoided.

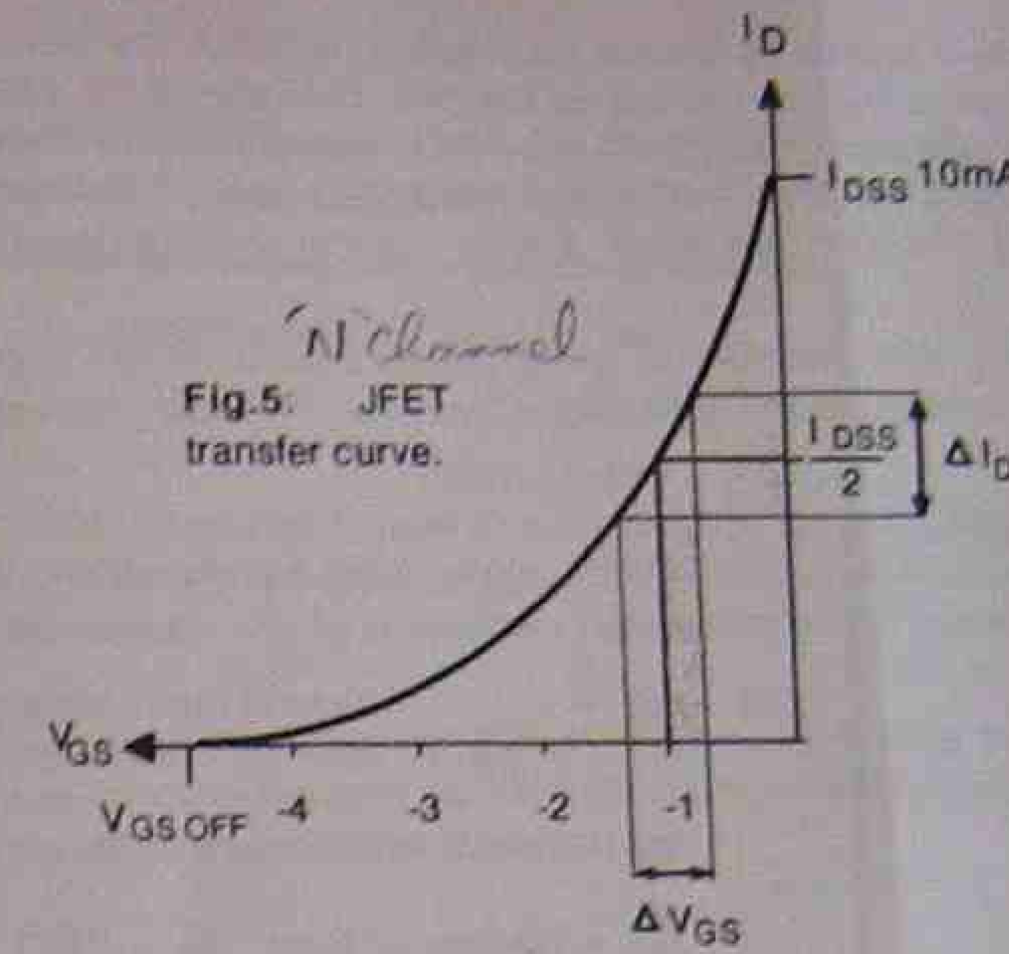


Fig.5: JFET transfer curve.

From the curve it can be seen that if Vgs is varied, the drain current will vary. The relationship between these values is an important FET characteristic, and identifies the effective gain of the device. The relationship is know as the mutual conductance (or forward transconductance) given the symbol **gm**. (g = conductance, m = mutual). Because the relationship is not linear, the value of gm needs to be determined for the chosen operating point. For the purposes of these notes, gm can be determined using the equation:

> ☐ $gm = \frac{I_D}{V_{GS}}$ siemens or amps/volt

This equation is an approximation, as gm should ideally be calculated by dividing the change in I_D by the change in Vgs. The transfer curve uses the delta symbol to depict the changes referred to. The value of gm is required to calculate the gain of an JFET amplifier.

5. DC CONDITIONS FOR A JFET

As for a transistor amplifier, a JFET needs to be biased using resistors that:

- give a drain voltage (Vd) equal to half the supply voltage.
- a gate-source reverse bias voltage to give a drain current of $I_{DSS}/2$.

To achieve this, the circuit configuration of Fig.6 can be used in which:

- Rg connects the gate to ground and Vg = 0V because no current flows in Rg.
- Vs is a positive voltage caused by the current I_D flowing in Rs.
- Because the source terminal is positive compared to ground, therefore ground can be regarded as being negative compared to the source terminal. As the gate is at ground potential, the gate is therefore negative compared to the source terminal.

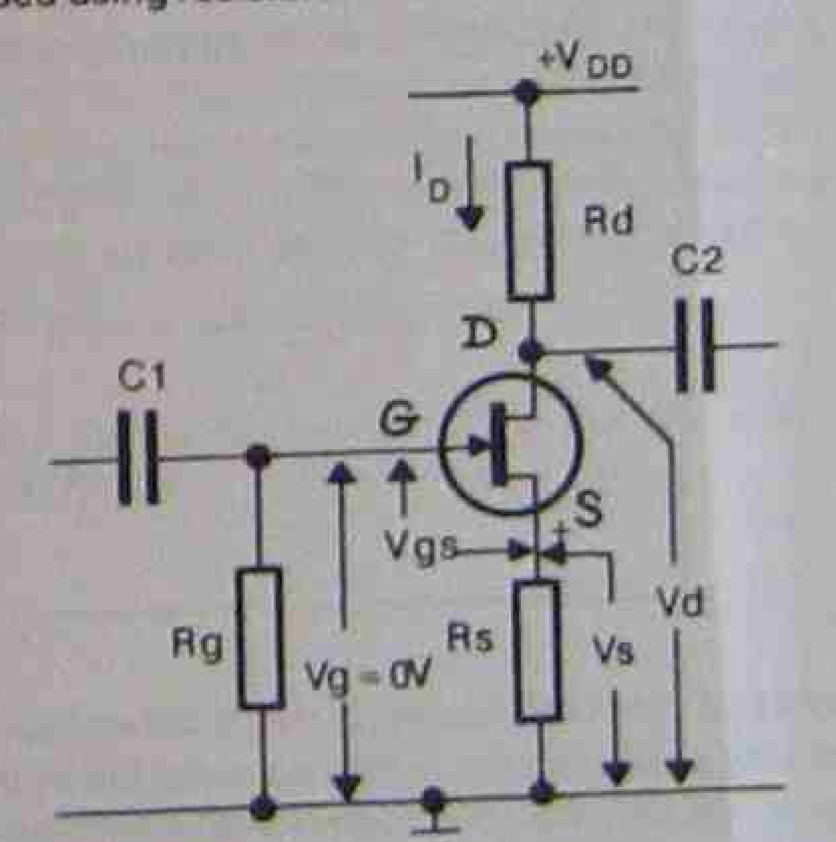


Fig.6

This is known as source self biasing, in which the voltage drop across the source resistor provides the reverse bias for the FET. The value of the source resistor therefore determines the DC operating conditions for the FET. The value of the drain resistor (R_D) is calculated to give a value of $V_{DD}/2$ for V_D. The equations required are listed below:

- ☐ $I_D = \frac{I_{DSS}}{2}$ (approximation)
- ☐ $V_{GS} = -I_D R_S$
- ☐ $R_S = \frac{V_{GS}}{I_D}$ (from equation above)
- *OR $R_S = \frac{1}{g_m}$ (if transfer curve not given)
- ☐ $V_D = V_{DD} - I_D R_d$ (choose R_D for $V_D = V_{DD}/2$)
- ☐ R_g = 1Mohm or less (for a JFET)

The following example shows how these resistor values are calculated.

EXAMPLE: For the circuit of Fig.6, given that the supply voltage (V_{DD}) is 10V and the FET has the transfer curve shown in Fig.5, calculate the following:

(a) value of R_S to correctly bias the circuit.

(b) value of R_d for optimum value of drain voltage.

(c) g_m for the FET.

Solution:

(a) $I_D = \frac{I_{DSS}}{2}$ = 5mA

$V_{GS} = -1V$ (from transfer curve, Fig.5)

$R_S = \frac{V_{GS}}{I_D}$ = 1/5mA = 200 ohm (chose NPV of 220 ohm)

(b) $V_D = \frac{V_{DD}}{2} = 10/2$ = 5V

$R_D = \frac{V_{DD} - V_D}{I_D}$ = (10 - 5)/5mA = 1k ohm

(c) $g_m = \frac{I_D}{V_{GS}}$ = 5mA/1V = 5mA/V

Note that g_m is expressed as mA/V rather than with the SI unit (Siemen). Some texts use mS to indicate a value for g_m in milli siemens, but as this can be confused with milliseconds (ms) these notes will use the more conventional term of milliamps per volt, (mA/V). Note also that the nearest preferred value (NPV) for the resistors has been chosen.

6. THE MOSFET

The MOSFET has a similar construction to the JFET except a very thin layer of silicon oxide is placed between the gate and the channel. This prevents the junction from conducting if the voltage between the gate and the channel becomes forward biased. There are two types of MOSFETS:

- DE-MOSFET: depletion-enhancement mode MOSFET. Designed to operate with the gate-source voltage either reverse biased (depletion mode) or forward biased (enhancement mode).
- E-MOSFET: enhancement mode MOSFET, designed to operate with the gate-source voltage forward biased. This device will not operate with the gate-source voltage reverse biasing the device (as in the JFET) as it needs the forward bias to create the channel that allows current to flow.

The symbols for these devices are shown in Fig.7 for both the N and P type devices. Note that the E-MOSFET shows the channel as a broken line indicating a forward bias must be applied to complete the channel. The symbols of the JFET are included for comparison.

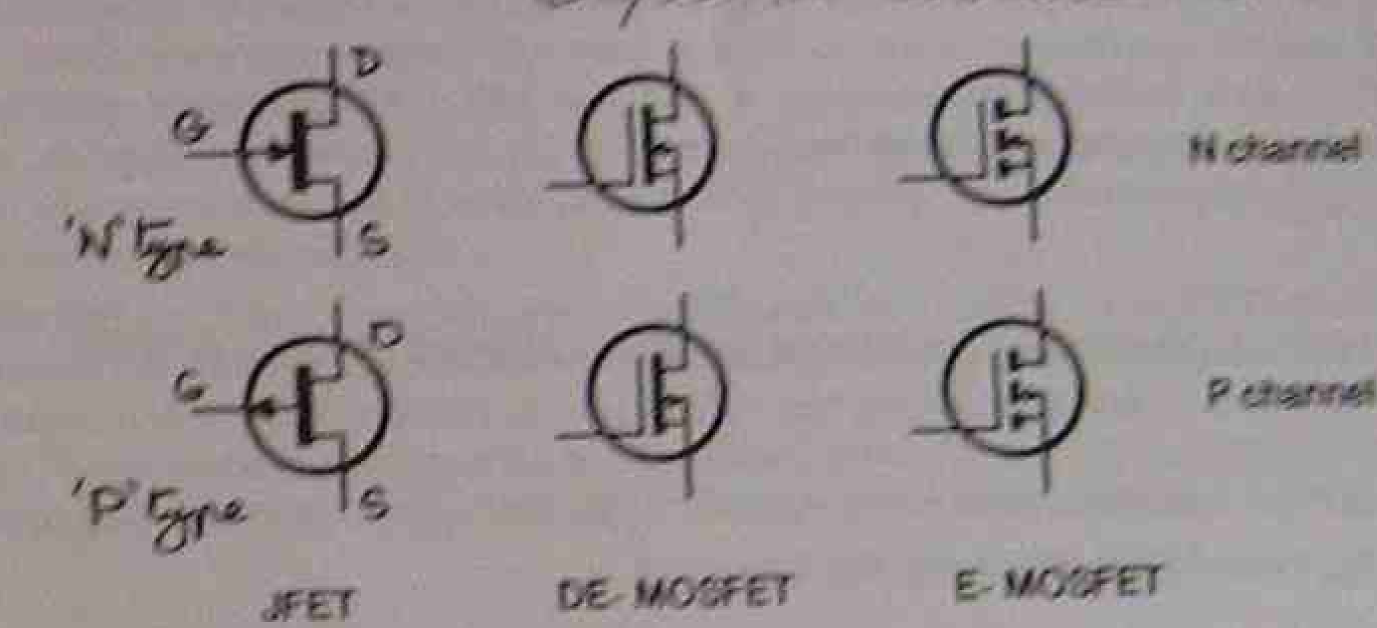


Fig.7: FET symbols

Because of the insulation between the gate and channel in a MOSFET, these devices have a much higher input resistance than a JFET, which although operated in reverse bias, still have a small leakage current between the gate terminal and the channel. Because of the high input resistance, the MOSFET must be handled carefully to prevent electrostatic discharge (ESD) from damaging the device. When soldering a MOSFET to a PCB, the soldering iron must be earthed, and if the device is to be handled, either wear an earth strap or discharge yourself by touching an earthed appliance.

- A useful mnemonic for remembering the symbol for the N channel FET is that the arrow points **IN** for **N**.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 10

1. For a P channel JFET, if the value of Vgs is increased from + 1V to + 3V:
   (a) the drain current .... (increases/decreases)
   (b) the depletion region .... (widens/narrows)
   (c) the resistance of the channel .... (increases/decreases)

2. Briefly explain why the gate-source junction of a JFET must be reverse biased.

3. Define the terms:
   (a) I_{DSS}
   (b) $V_{GS\,OFF}$
   (c) self biasing as applied to a FET amplifier.
   (d) gm for a FET.

4. Sketch the symbols for the N-channel and P-channel JFET, DE-MOSFET and E-MOSFET.

5. For the circuit of Fig.1, use the transfer curve shown to calculate:
   (a) values of Rg, Rs and Rd to effectively bias the circuit.
   (b) gm of the JFET.

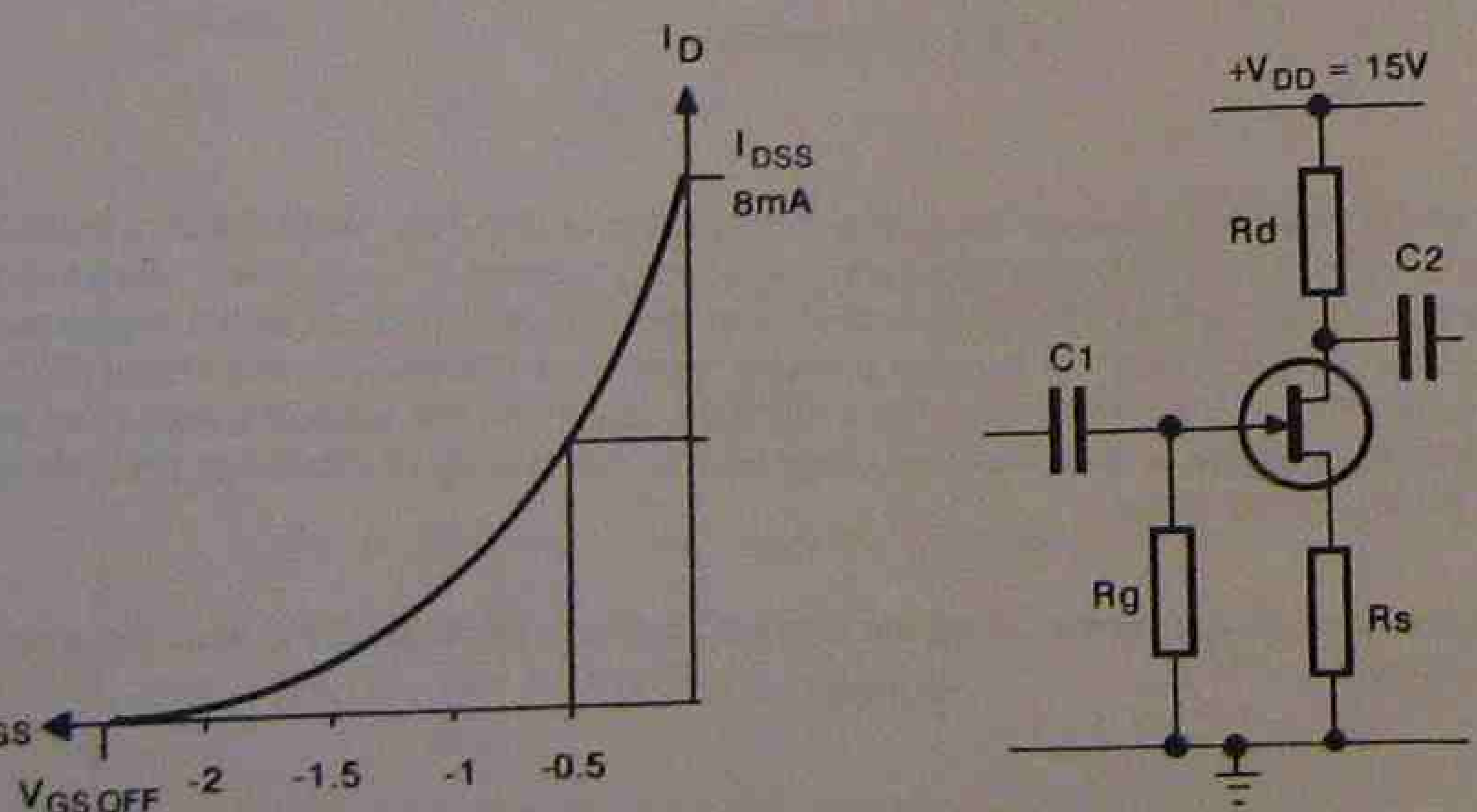


Fig.1

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 11

References: Electronic Devices, 2nd Ed. Floyd. Chapter 9.

JFET AMPLIFIERS

OBJECTIVES At the end of this lesson you will be able to:

- Describe the function of each component in the basic common drain (CD) and common source (CS) JFET amplifier configurations.
- Calculate the voltage gain, input resistance and output resistance for a JFET CS amplifier.
- List practical applications for CS and CD JFET amplifiers.

1. INTRODUCTION

The JFET amplifier is simpler than its BJT counterpart in that it uses less components and is voltage operated. As described previously, the gate-source voltage of a FET controls its drain current, unlike a transistor where the base current controls the collector current. Because of this, the input impedance of a FET amplifier is generally much higher than that for a BJT amplifier as negligible current is required at the input. This is the main reason for using a FET amplifier as compared to a BJT amplifier, the JFET amplifier has a lower voltage gain and often has a poorer frequency response.

As for the BJT amplifier, there are several configurations possible for a JFET amplifier and the main two are the common drain (CD) and the common source (CS). The CD amplifer is similar to the common emitter BJT amplifier configuration and the CS amplifier is similar to the emitter follower. These notes briefly describe the operation of the CS and CD amplifiers and show how the voltage gain, input resistance and output resistance can be calculated.

2. THE JFET CS AMPLIFIER

The circuit of a JFET common source amplifier is shown in Fig.1. The DC conditions were described previously in which the gate-source junction is reverse biased by the positive voltage present at the source. The gate is at ground potential and the drain terminal should have a DC voltage around half the supply voltage.

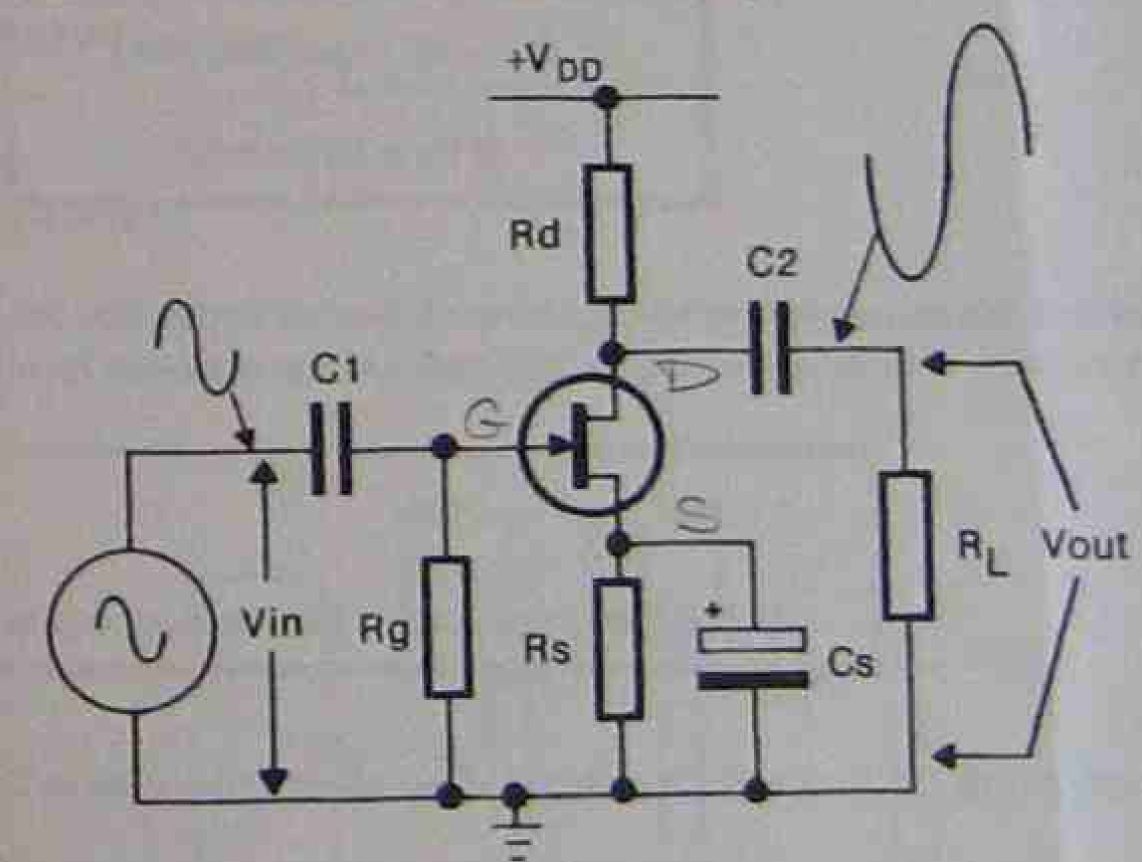


Fig.1: The JFET CS amplifier

The input voltage is applied between the gate and ground and the output signal appears at the drain terminal. Capacitors C1 and C2 are used to isolate the DC potentials. If the signal source has no DC component, C1 could be deleted, but is usually included for protection. This circuit is referred to a common source amplifier, as the source terminal is neither an input or an output and connects to ground. The bypass capacitor is optional, although its inclusion increases the voltage gain of the circuit. As in the BJT CE amplifier, the changes in the drain current caused by Vin varying Vgs are converted to a voltage change by Rd. Hence the output is 180° out of phase with the input.

3. AC CONDITIONS OF THE JFET CS AMPLIFIER

The operation of the common source amplifier is as follows:

- In Fig.1, if Vin increases in the positive direction, the gate voltage will rise. The source voltage remains constant at a predetermined positive value as Cs bypasses any AC component. Because of this, the voltage difference between the gate and source drops, allowing more drain current to flow. This causes the voltage drop across Rd to rise and Vd drops. Thus the output signal swings in the negative direction when the input signal rises in the positive direction.
- When the input signal goes negative, the potential across the gate-source junction increases, causing the drain current to fall. As the voltage drop across the drain resistor is now reduced, the voltage at the drain terminal rises. Thus a negative swing at the input causes a positive swing at the output.

Note how gm (the relationship between Vgs and I_D) forms the basis of the operation of the circuit. The more sensitive the JFET, or the higher its value of gm, the greater the gain of the circuit. The equation to calculate thevoltage gain (Av) of a JFET CS amplifier is:

$$\square\ A_V = \frac{g_m (R_d // R_L)}{1 + g_m R_s}$$

where: gm = mutual conductance in A/V Siemens
Rd = value of drain resistor
R_L = value of load resistor
Rs = value of unbypassed source resistor

If the amplifier has the source resistor bypassed as in Fig.1, the equation becomes:

$$\square\ A_V = g_m (R_d // R_L)$$

(if Rs is bypassed)

Note that this equation now has no denominator as bypassing the source resistor Rs results in a denominator of unity. If there is no load resistor, the equation is even simpler, becoming:

$$\square\ A_V = g_m R_d$$

(if Rs is bypassed and no load resistor connected)

The input resistance (r_{in}) for the circuit is the value of the gate resistor Rg: That is:

$$\square\ r_{in} = R_g \text{ ohms}$$

The output resistance (r_{out}) for the circuit is the value of the drain resistor Rd: That is:

$$\square\ r_{out} = R_d \text{ ohms}$$

WORKED EXAMPLE:

For thecircuit of Fig.2, given that the gm of the JFET is 3ma/V calculate the following:

(a) voltage gain of the circuit
(b) input resistance
(c) output resistance
(d) voltage gain if the bypass capacitor is removed

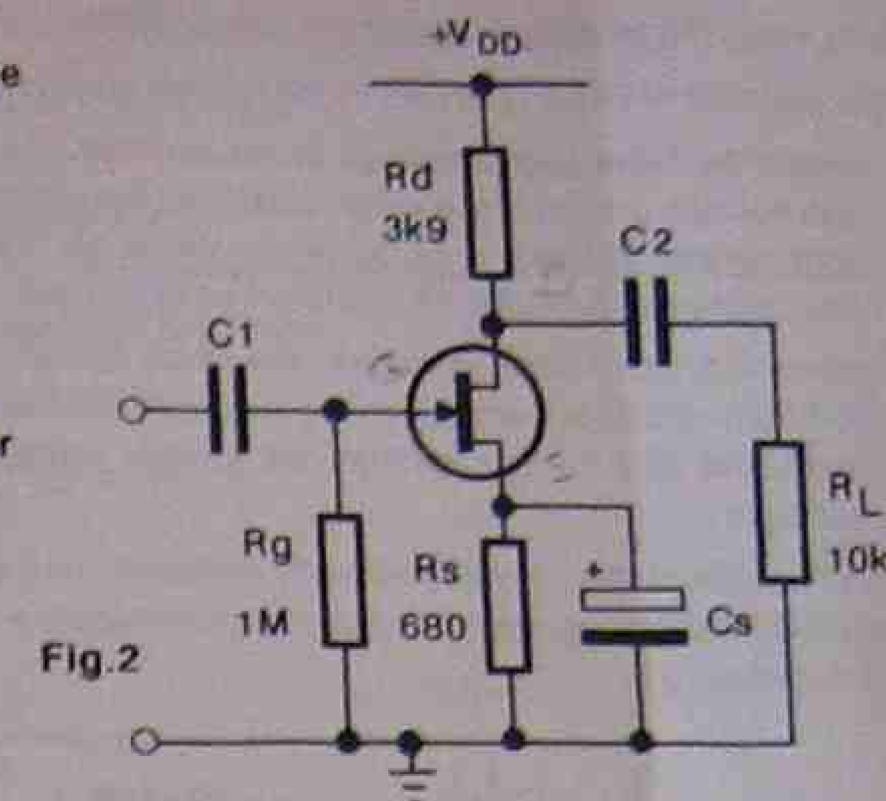


Fig.2

Solution:

(a) $A_V = \frac{g_m(R_d//R_L)}{1 + g_m R_s} = 3 \times 10^{-3} (3k9//10k) = 3 \times 10^{-3} \times 2k8 = \underline{8.4}$

(b) $r_{in} = R_g$ ohms = **1M ohm**

(c) $r_{out} = R_d$ ohms = **3k9 ohm**

(d) $A_V = \frac{g_m(R_d//R_L)}{1 + g_m R_s}$

$= \frac{3 \times 10^{-3}(3k9//10k)}{1 + (3 \times 10^{-3} \times 680)} = 8.4/3.04 = \underline{2.76}$

4. THE COMMON DRAIN JFET AMPLIFIER

The circuit of Fig.3 shows a common drain JFET amplifier. The input is applied between the gate and ground as for the CS amplifier, but the output is taken from the source terminal. The drain terminal is connected directly to the supply voltage and represents the common terminal of the amplifier. Like the emitter follower amplifier, the CD JFET amplifier features a high input resistance, a low output resistance and a voltage gain of approximately one. The quiescent DC potential at the source terminal cannot normally be arranged to equal half the supply voltage, as this would be too high a voltage to allow the JFET to conduct. To overcome this, some circuits use a potential divider biasing network to give a positive voltage at the gate, allowing a higher value of source voltage.

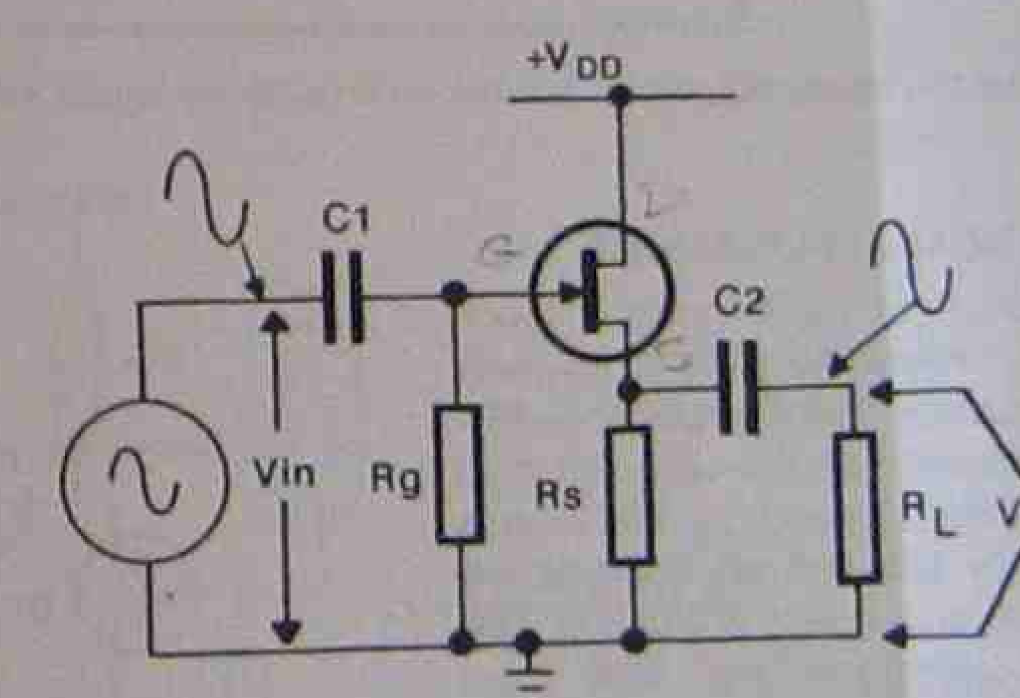


Fig.3: The common drain JFET amplifier

5. **AC CONDITIONS FOR THE CD AMPLIFIER**

The amplifier of Fig.3 operates in the following way:

- when the input signal swings in the positive direction, the drain current increases, causing the voltage across the source resistor to rise. However the circuit will operate to maintain the gate-source voltage at its quiescent value, so the change in voltage at the source will equal the change at the gate. This is referred to as source follower action.
- when the input voltage drops, the drain current through the FET will also drop, giving a reduced value of voltage at the source terminal. As the FET will operate to maintain the quiescent gate-source voltage, the source voltage will fall by the same amount as the input voltage.

Thus the gain of the circuit is virtually equal to unity and is the value of voltage gain that can be assumed for the purposes of these notes. However a more accurate equation, which is included for interest only is:

$$A_V = \frac{g_m (R_S // R_L)}{1 + g_m R_S}$$

where: g_m = mutual conductance in A/V
R_L = value of load resistor
R_S = value of source resistor

** not for exam purposes. Voltage gain can be assumed to equal one.

As for the common source amplifier, the input resistance of the CD amplifier is:

$$r_{in} = R_g \text{ ohms}$$

The output resistance of the common drain amplifier is determined by:

$$r_{out} = \frac{R_S}{1 + g_m R_S} \text{ ohms}$$

** not for exam purposes. Output resistance can be assumed to be low.

Note that the above equations assume Rs is relatively large (500 ohms or greater)

PRACTICAL APPLICATIONS

The circuit of Fig.4 shows a touch switch. When the plate is touched, the induced signal is amplified by FET Ql, smoothed by the capacitor then amplified by Q2 and Q3. The relay is driven by Q3, and the contacts will close while the plate is touched. This circuit relies on the high input impedance offered by the FET input stage.

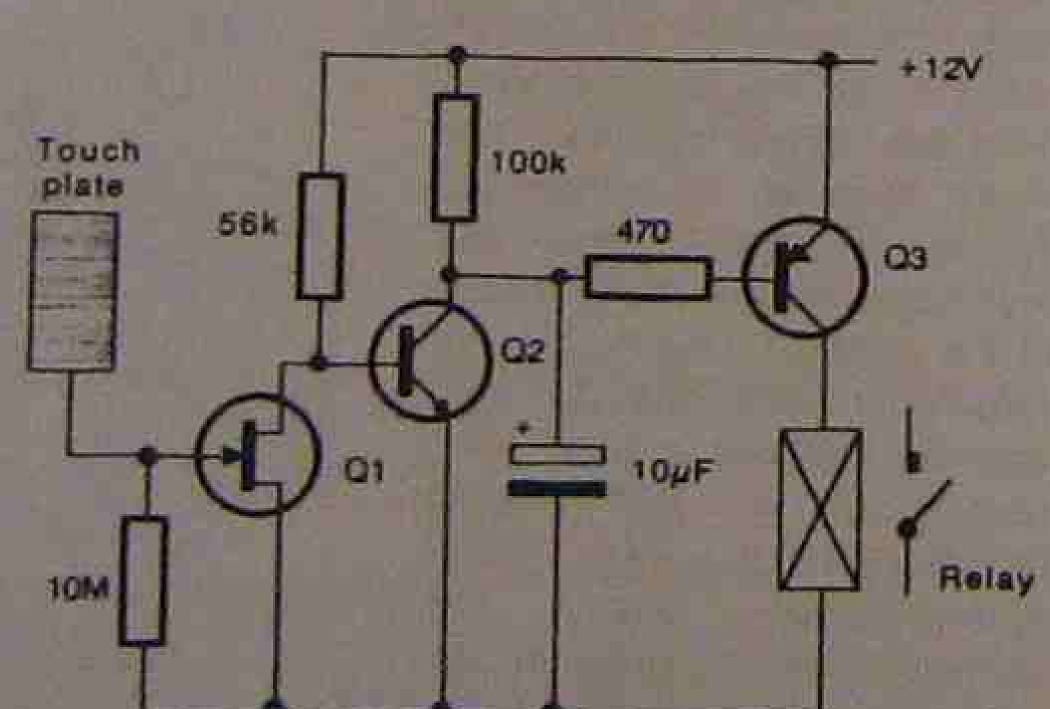


Fig.4: Touch switch with FET input stage

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 11

1. For the circuit of Fig.1, use the transfer curve shown to calculate:
   (a) the quiescent drain current .
   (b) suitable values for Rs and Rd. (Assume Vd = Vdd/2)
   (c) gm of the FET

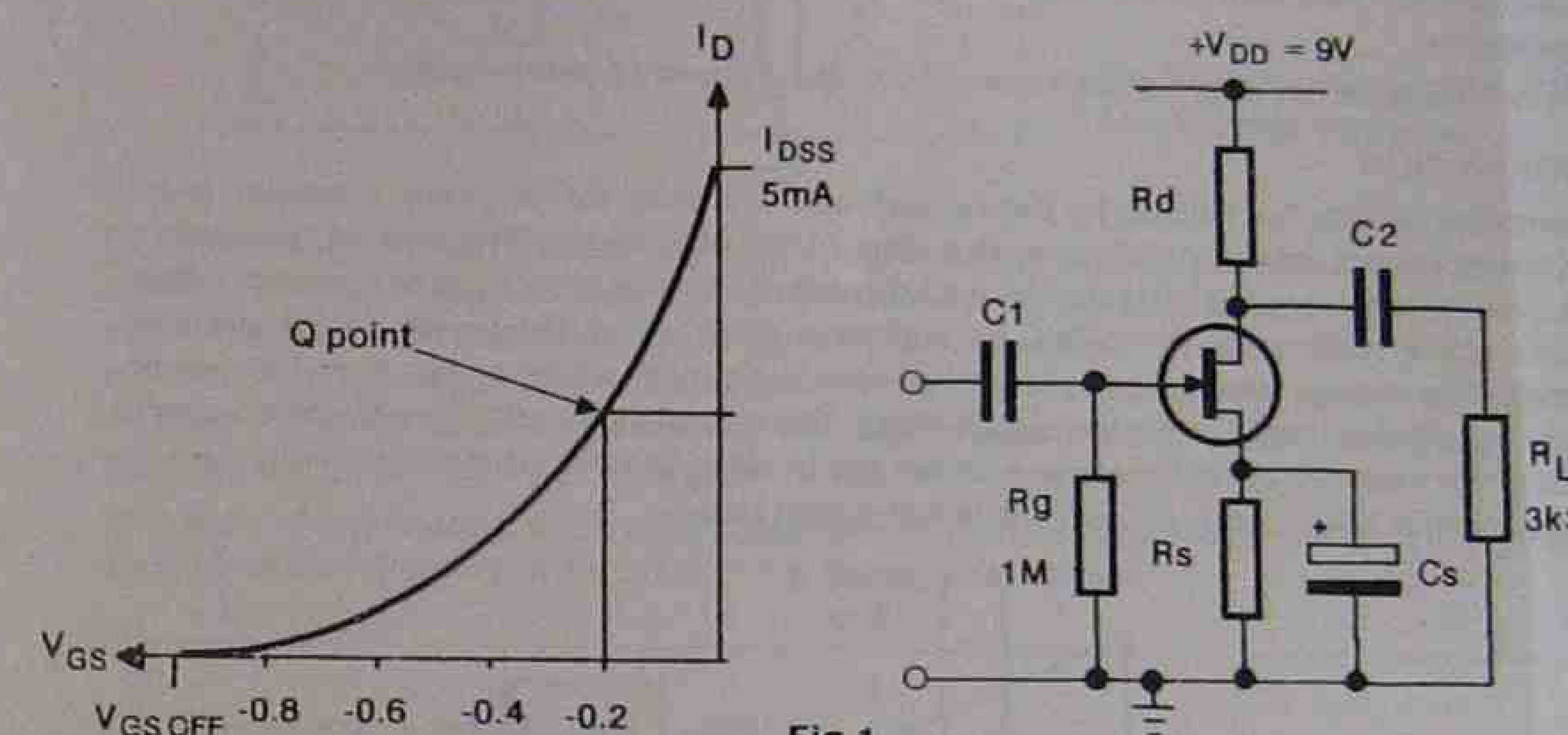


Fig.1

2. Using the resistance and gm values obtained in Q1, calculate the:
   (a) voltage gain of the circuit
   (b) input resistance
   (c) output resistance
   (d) voltage gain if the bypass capacitor is removed

3. For the circuit of Fig.2, calculate the approximate:
   (a) voltage gain.
   (b) input resistance.
   (c) output resistance.

4. Identify the circuit configurations of Figs.1 and 2.

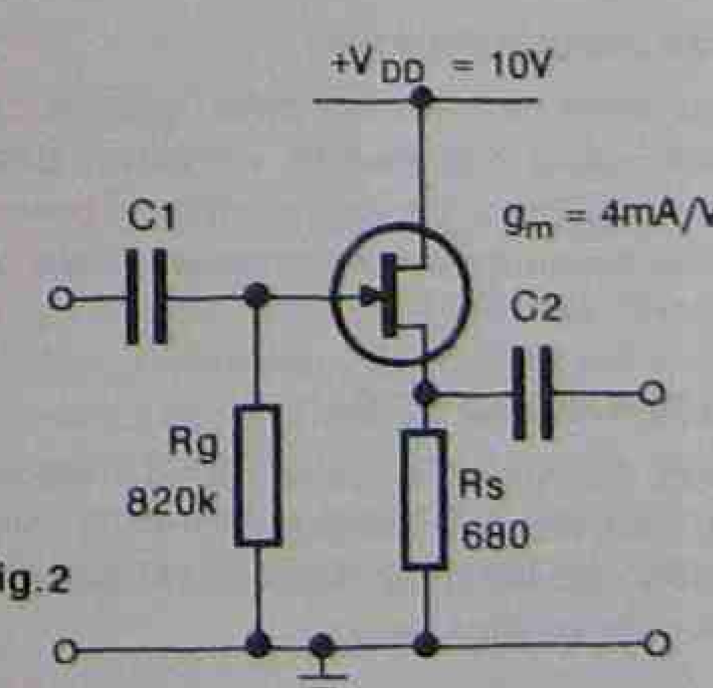


Fig.2

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 13

References: Electronic Devices, 2nd Ed. Floyd. Chapter 10.

POWER OUTPUT STAGES - PART 1

OBJECTIVES At the end of this lesson you will be able to:

- List the basic differences between class A and B power output amplifiers.
- Calculate the maximum power output of a power amplifier given the supply voltage and load resistance.
- Sketch the basic circuit of a class B complementary symmetry power amplifier.

1. INTRODUCTION

The amplifier circuits that have so far been described are suitable for low power operation only. That is they cannot deliver useful power to a load. In industry, an amplifier may be required to drive a servo motor, an indicating device, a loud speaker or some other form of transducer that requires power to operate. To do this, an additional stage called the power output stage is required. This section has the general form shown in Fig.1, in which power from the power supply is passed to the load via the output stage. The output signal will (ideally) be an exact replica of the input signal, and because a power output stage generally has unity voltage gain, a voltage amplifier will normally precede the power output stage.

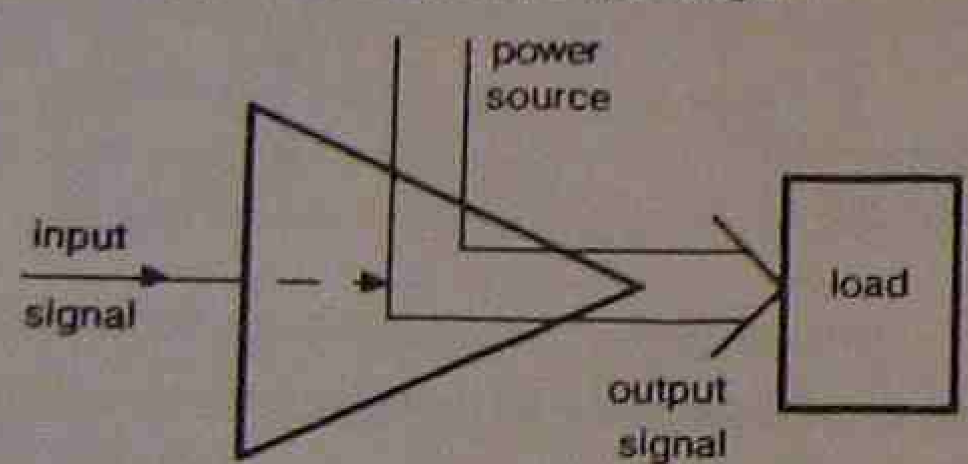


Fig.1: General form of a power output stage

The basic requirements of any power output stage are:

- high efficiency
- low distortion
- low output impedance

These notes describe the basic types of power output stages and show how to calculate the power output that can be expected, given the value of the supply voltage and the load resistance. There are various types of power amplifiers, categorised as Class A, Class B and so on. The differences between these classes of operation are also described.

CLASS A AMPLIFIERS

A class A amplifier is defined as an amplifier where the output device(s) conduct for 360°. This means the output transistor(s) (or FETs etc) of the circuit are always conducting current. All the amplifier circuits so far discussed in these notes are class A, although none have been used in a power output application.

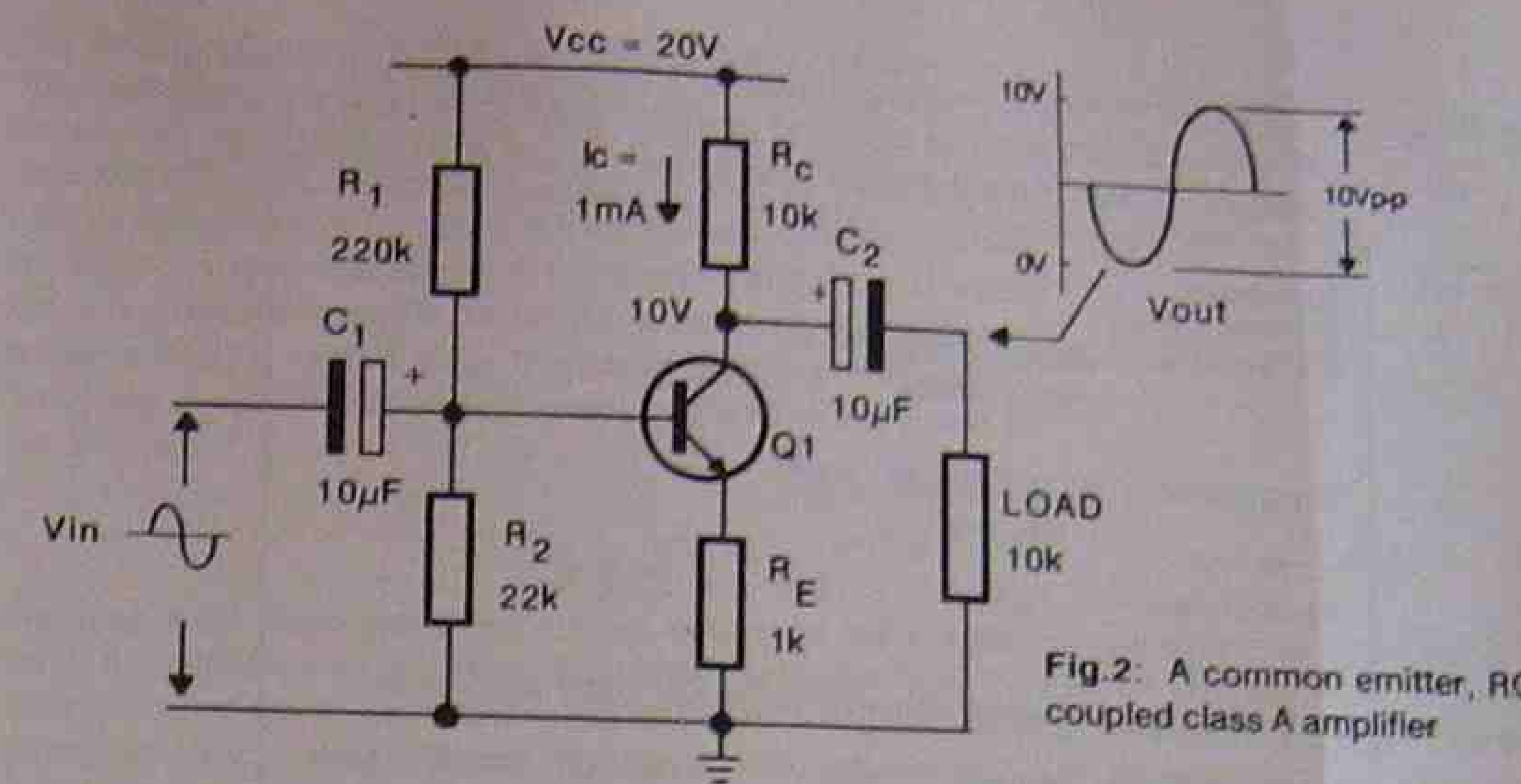


Fig.2: A common emitter, RC coupled class A amplifier

Fig.2 shows a common emitter amplifier, biased so the collector voltage is half the supply voltage. The DC voltage at the collector is isolated from the load by capacitor C2, and the theoretical maximum output voltage swing is from 0V (transistor Q1 fully on) to10V (Q1 turned off). Although this circuit is not able to deliver useful power, its efficiency can be calculated to illustrate the process. Note that the load resistance equals the output resistance (10k), as this gives the best power transfer. It also gives the output swing of Vcc/2 as shown.

Efficiency of any system equals the output power divided by the input power. The following shows how the efficiency of the circuit of Fig.2 can be calculated:

$$\text{Efficiency\%} = \frac{\text{output power}}{\text{input power}} \times 100$$

$$P_{in} = V_{cc} \times I_c = 20V \times 1mA = 20mW$$

(That is: the input power = Vcc multiplied by the quiescent collector current of 1mA)

$$P_{out} = \frac{(V_{orms})^2}{R_{load}}$$

$$V_{orms} = \frac{V_{op\text{-}p}}{2\sqrt{2}} = \frac{10}{2.83} = 3.53V$$

$$P_{out} = \frac{3.53V^2}{10k} = 1.25mW$$

$$\text{efficiency \%} = \frac{1.25mW}{20mW} \times 100 = 6.25\%$$

The 6.25% efficiency figure can be improved by making the collector resistor the load. This will give a maximum efficiency of 25% as V_{RL} now equals 20Vp-p, although a DC current will flow in the load, making this type of connection impractical in most cases. The 25% value is the best efficiency for any Class A amplifier, except for the transformer coupled type, which has a maximum theoretical efficiency of 50%. Note that these efficiencies are the maximum possible, and most class A amplifiers have a much lower efficiency. To show why efficiency is so important, consider the following example:

EXAMPLE 1: Calculate the input power required for an industrial sound system with an efficiency of 20% that has an output power of 50kW. (Note that this power output is typical of most outdoor sound systems used for pop concerts.)

Solution:

$$\text{Efficiency\%} = \frac{\text{output power}}{\text{input power}} \times 100$$

$$\text{input power} = \frac{\text{output power}}{\text{efficiency}} \times 100$$

$$\text{input power} = \frac{50\text{kW}}{20} \times 100 = \underline{250\text{kW}}$$

If the system is supplied with a single phase, 240V AC supply,

$$\text{the line current (I) equals} \quad \frac{Pin}{V} = \frac{250\text{kW}}{240\text{V}} = \underline{1041.67\text{A}}$$

To achieve a higher output power for the circuit of Fig.2, the values of the collector resistor and the load resistor could be lowered to, say 10 ohms and the emitter resistor replaced with a short circuit. However, to obtain the necessary 10V at the collector, a quiescent current of 1 amp would be required to drop 10V across Rc. Thus, under no signal conditions, the total power consumed by the circuit would equal 20V x 1A, which equals 20W. In this case, 10W would be dissipated by the collector resistor and 10W by the transistor. The maximum power output would now be 5W (as per the calculations used to calculate efficiency). However an advantage of the class A amplifier is its relatively low distortion.

CLASS A AMPLIFIER CHARACTERISTICS

- ☐ maximum efficiency = 25% (for RC coupled type)
- ☐ maximum efficiency = 50% (for transformer coupled type)
- ☐ low distortion (compared to class B)
- ☐ high quiescent current flows in the output device(s)
- ☐ output device(s) conduct for 360° of the input signal

3. CLASS B POWER OUTPUT STAGE

A class B amplifier requires a minimum of two output devices, connected as shown in Fig.3. A class B amplifier is one where the output devices conduct for exactly half the input cycle (180°). In Fig.3, Q1 will conduct for the positive half of the input signal, then turn off while Q2 conducts for the negative half. Thus the output signal is produced by both transistors, in which one is on while the other is off. Note also that when the input signal is zero, the output across R_L is also zero and both Q1 and Q2 are off.

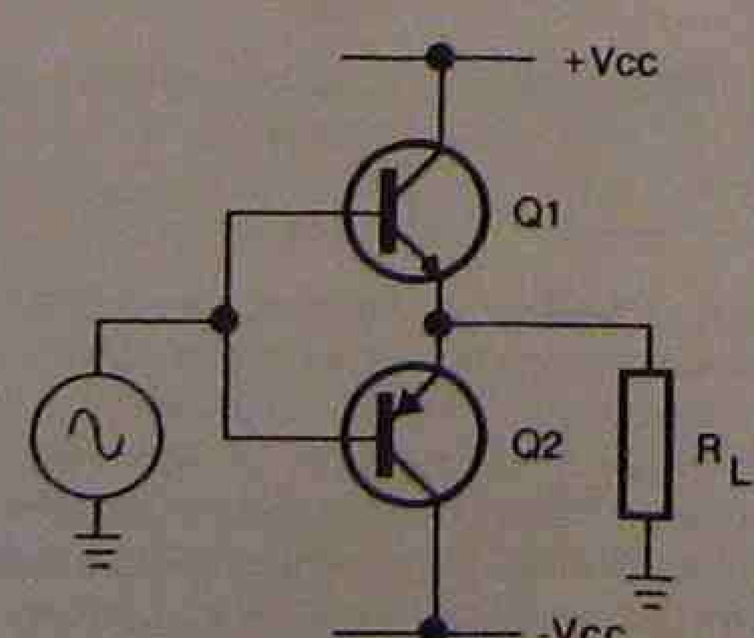


Fig.3: Class B output stage

The circuit of Fig.3 is powered by a dual polarity power supply. This type of power supply is effectively two individual supplies, connected as shown in Fig.4. With this type of supply, the amplifier will have a quiescent output voltage of zero, and be able to swing in both the positive and negative directions. When Q1 is on, current is taken from the positive supply, and when Q2 is on, current flows via the negative supply. It is possible to power a class B output stage from a single rail power supply, providing a coupling capacitor is used to connect the load to the amplifier.

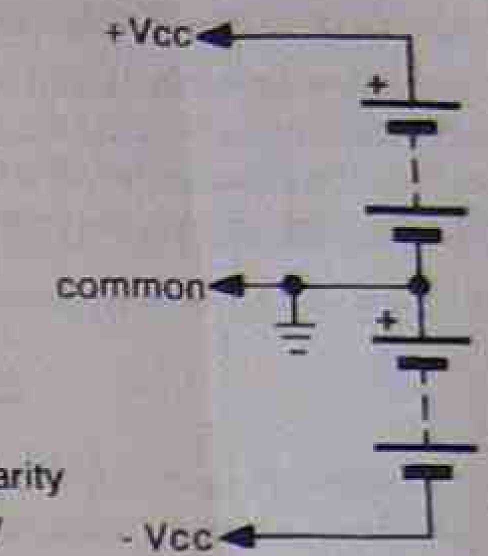


Fig.4: A dual polarity power supply

Because there is no quiescent current flowing in the output devices when the input signal is zero, the efficiency of the class B amplifier is higher than the class A configuration. It can be shown that the maximum efficiency is 78.5% for a class B amplifier. However the trade off for the higher efficiency is an increase in distortion. As shown in Fig.5, distortion occurs when one transistor is turning off and the other is turning on. This type of distortion is known as crossover distortion and is a direct result of the 0.6V needed to turn a transistor on.

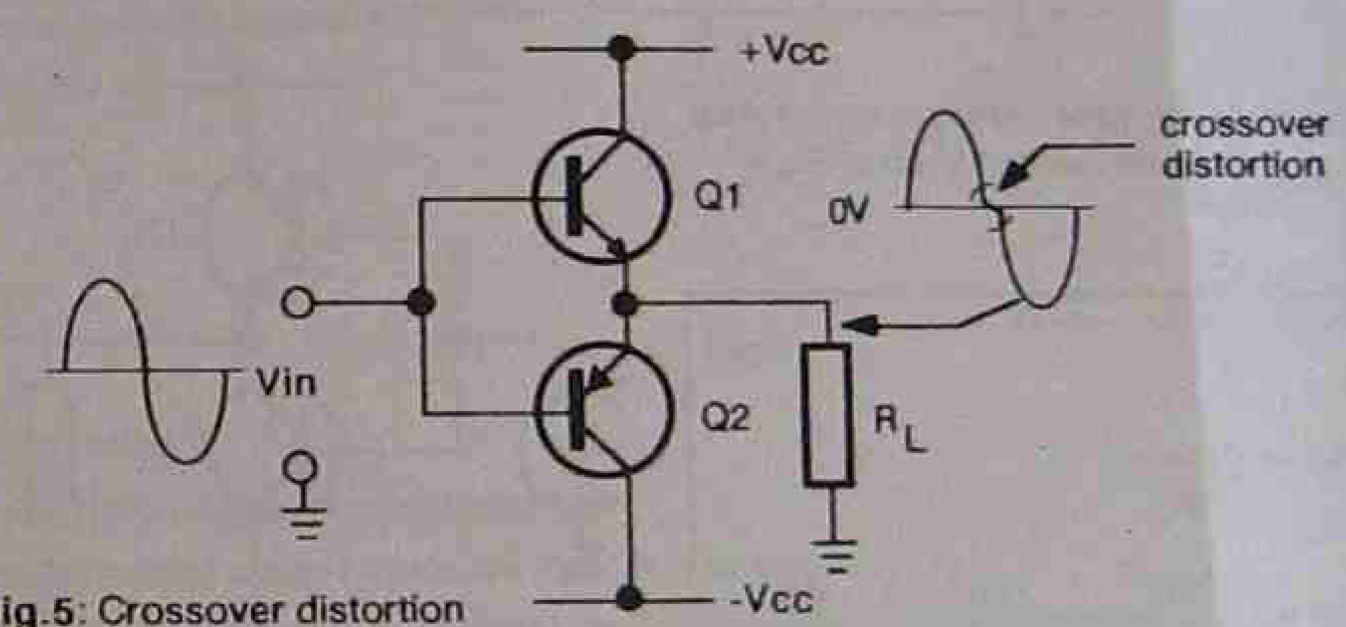


Fig.5: Crossover distortion

In Fig.5, the input signal provides the forward bias for the output transistors. When the input signal is greater than +0.6V, Q1 will be turned on and the output signal will follow the input as a result of emitter follower action. Likewise, when the input is more negative than -0.6V, Q2 will conduct and the output signal will be a replica of the input, again by emitter follower action. However, when the input signal is between +0.6V and -0.6V, neither of the output transistors can conduct, and the output will fall to zero. Thus, there is a gap in the output signal at the points when the input signal changes polarity. This can be overcome by applying forward bias to the circuit, as will be described in Part 2 of these notes. The characteristics of the class B amplifier are:

CLASS B AMPLIFIER CHARACTERISTICS

- ☐ maximum efficiency = 78.5%
- ☐ high distortion (compared to class A)
- ☐ lower power dissipation by the output devices
- ☐ two output devices needed, both conduct for 180°

4. POWER OUTPUT FOR THE CLASS B AMPLIFIER

The maximum possible output power for a class B amplifier is calculated in the same way as already described. That is, the maximum output voltage swing is first converted to an RMS value, then the equation V^2/R is used. However, an alternative equation can be used, as shown below. Note that $V_{op\text{-}p}$ is the voltage difference between the supply rails.

<u>Power output equation</u>

$$P_{out} = \frac{(V_{orms})^2}{R_{load}}$$

$$(V_{orms})^2 = \left(\frac{V_{op\text{-}p}}{2\sqrt{2}}\right)^2 = \frac{V_{op\text{-}p}^2}{8}$$

☐ **Pout** $= \frac{(V_{op\text{-}p})^2}{8R_L}$

<u>EXAMPLE 2</u>: Calculate the maximum output power for the circuit of Fig.6.

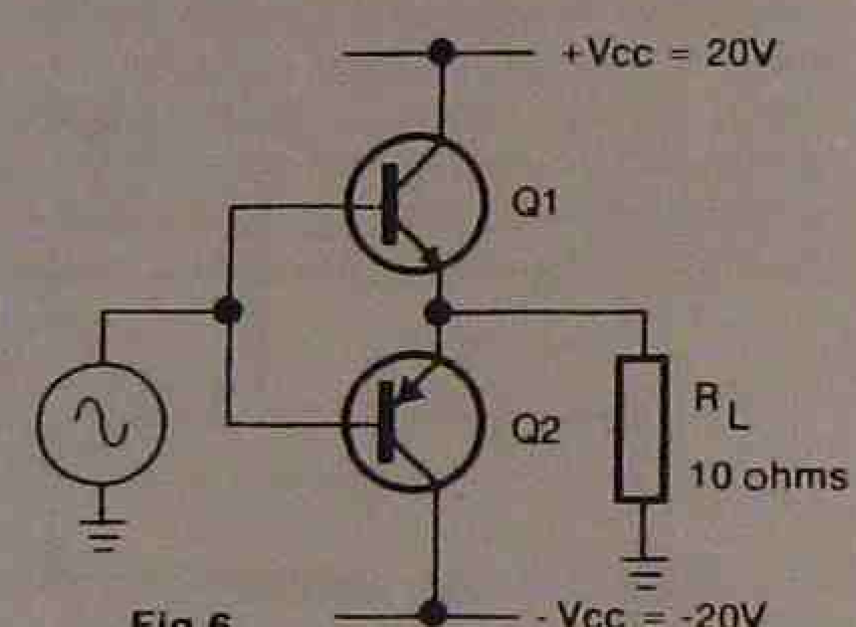


Fig.6

Solution:

$$P_{omax} = \frac{(V_{op\text{-}p})^2}{8R_L} = \frac{(40)^2}{8 \times 10} = \frac{1600}{80} = \underline{20W}$$

5. VOLTAGE GAIN AND POWER GAIN OF THE CLASS B AMPLIFIER

The circuit of Fig.6 is referred to as a <u>Complementary Symmetry</u> amplifier, as it uses an NPN and a PNP transistor (complementary pair). Their characteristics need to be matched, in which both transistors should have the same current gain (symmetrical). The circuit configuration is actually two emitter followers connected in 'push-pull'. Thus, the output signal voltage will be virtually equal to the input signal, giving a voltage gain of unity. In practice, the gain will be less than unity due to losses. As well, the maximum output voltage swing will be less than the supply rails, due to voltage drops across the transistors.

The power gain of a complementary symmetry amplifier is equal to the current gain of the transistors.

☐ Av = unity

☐ Ap = current gain of transistors (β)

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 13

1. Calculate the line current required to supply an audio amplifier system delivering 40kW with an efficiency of 45%. Assume the supply voltage is 240V AC.

2. List one reason why a class A power amplifier would be preferred to a class B amplifier in a particular application. Also give one reason why a class B amplifier might be preferred in another application.

3. For the circuit of Fig.1, calculate:
   (a) the theoretical maximum output power.
   (b) the power gain of the circuit.

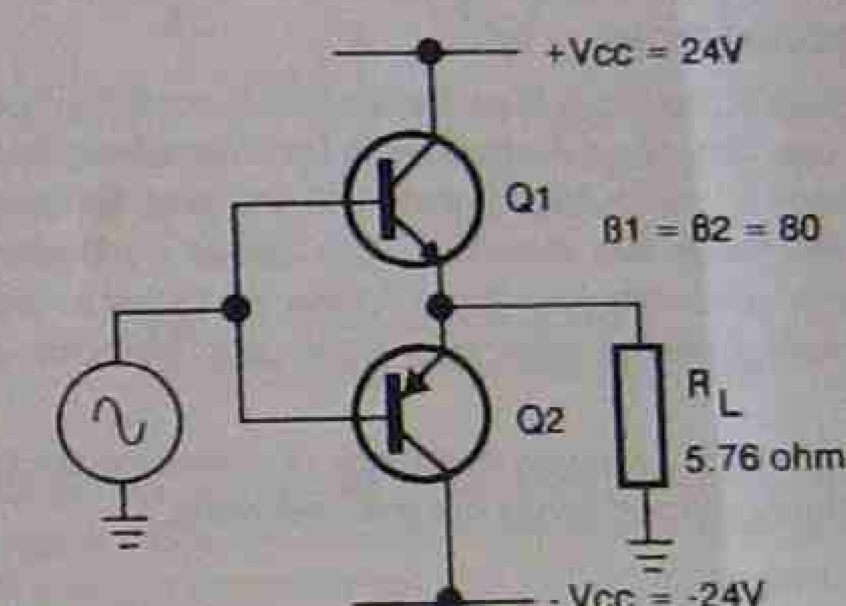


Fig.1

4. For the circuit of Fig.1:
   (a) State the class of operation of the circuit.
   (b) Sketch the output waveshape if the input is a sinewave.
   (c) Give two reasons why the theoretical maximum output power calculated in 3.(a) cannot be achieved in practice.

5. For the circuit of Fig.2:
   (a) State the optimum value of collector voltage.
   (b) Calculate the maximum output power.
   (c) Calculate the power being dissipated by Q1 when the input signal is zero.
   (d) State the maximum efficiency the circuit can achieve.

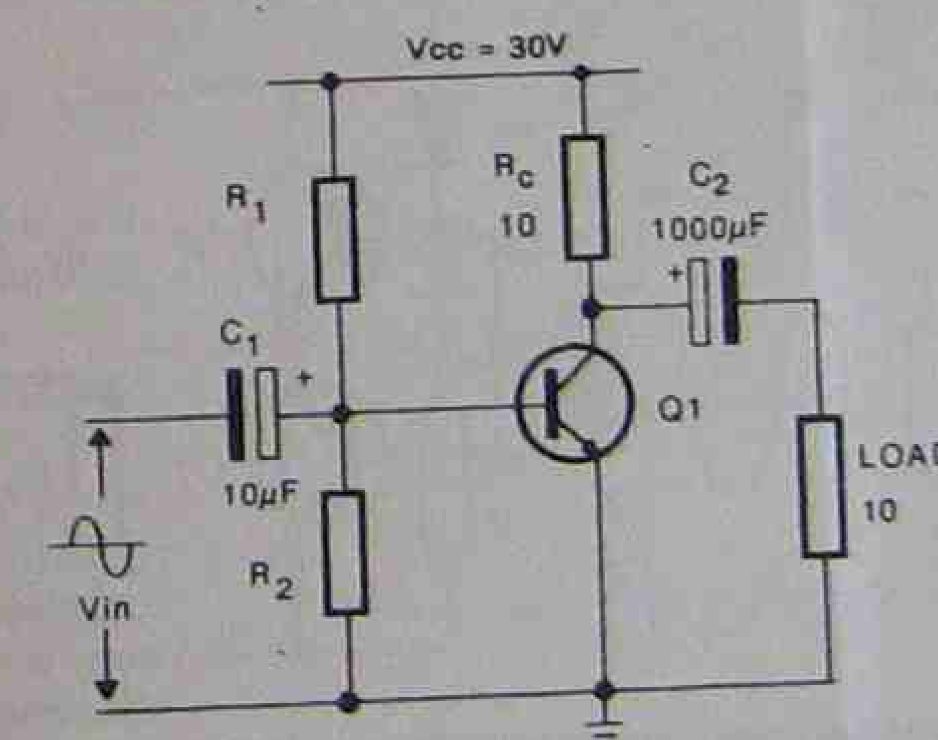


Fig.2

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 14

References: Electronic Devices, 2nd Ed. Floyd. Chapter 10.

POWER OUTPUT STAGES - PART 2

OBJECTIVES At the end of this lesson you will be able to:

- List the operating characteristics of a class AB complementary symmetry power amplifier.
- Calculate the DC voltages of a quasi-complementary power output amplifier.
- List practical limitations of a power amplifier.

1. INTRODUCTION

The class A and class B power amplifiers described previously both have limitations that restrict their use. The class A amplifier is very inefficient, and the class B amplifier has a high level of distortion in the output signal. The practical solution is a power amplifier that compromises between these two extremes. This circuit configuration is referred to as a class AB power amplifier, and requires the addition of forward bias to the basic class B complementary symmetry amplifier described previously. Methods of achieving this are described in these notes.

There are other practical limitations of a power output stage, and these, along with the methods to minimise the limitations are also described.

2. CLASS AB AMPLIFIERS

The circuit of Fig.1 shows the basic class B, complementary symmetry power output stage. The points to note about this circuit are:

- maximum efficiency equals 78.5%
- each transistor conducts for exactly 180° of the input signal
- there is crossover distortion in the output signal
- the forward bias for the transistors is supplied by the input signal

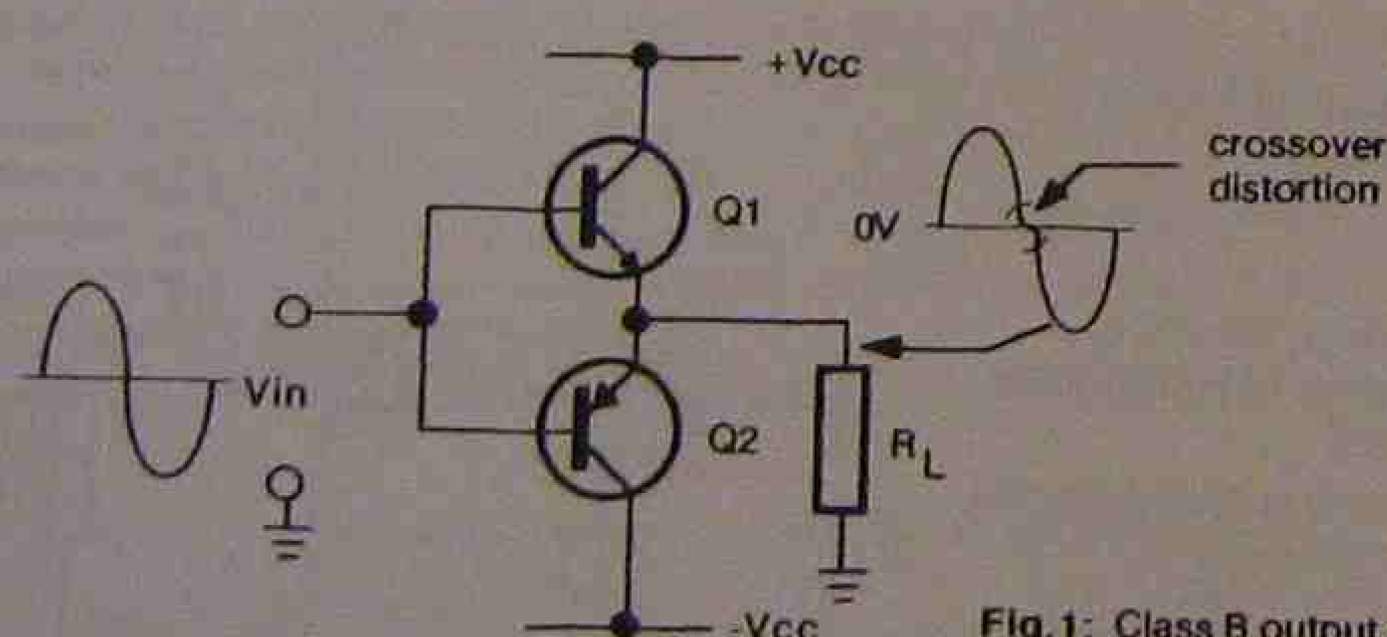


Fig.1: Class B output stage

To eliminate the crossover distortion, forward bias of 0.6V needs to be applied to both transistors. This will allow the transistors to still conduct when the input signal is less than 0.6V, and if the bias is correctly applied, the crossover distortion can be reduced to a negligible level. However, the transistors will now conduct for more than 180° of the input cycle, but for less than 360°. This class of operation is therefore between class A and class B, and is referred to as class AB operation.

The amount of forward bias determines how close the circuit operates to either of the extremes, and the compromise depends on the required efficiency and the amount of distortion that can be tolerated.

The circuit of Fig.2 shows how forward bias can be applied to the basic class B configuration shown in Fig.1. Note the following about Fig.2:

- This circuit is powered by a single rail supply, and a coupling capacitor is required to isolate the load resistor from the DC potential at the junction of the emitters of Q1 and Q2. The circuit can also be powered from a dual polarity supply.
- The input signal is applied to the base of Q2, but because both diodes are forward biased by R1, they are an AC short circuit and the input signal therefore also appears at the base of Q1.
- As for the basic class B circuit, the circuit of Fig.2 is effectively two emitter followers connected so that Q1 passes the positive half cycle of the input signal to the load, and Q2 handles the negative half cycle. However because there is now forward bias applied to the circuit, both transistors will conduct for more than 180° of the input signal, and crossover distortion is therefore eliminated.

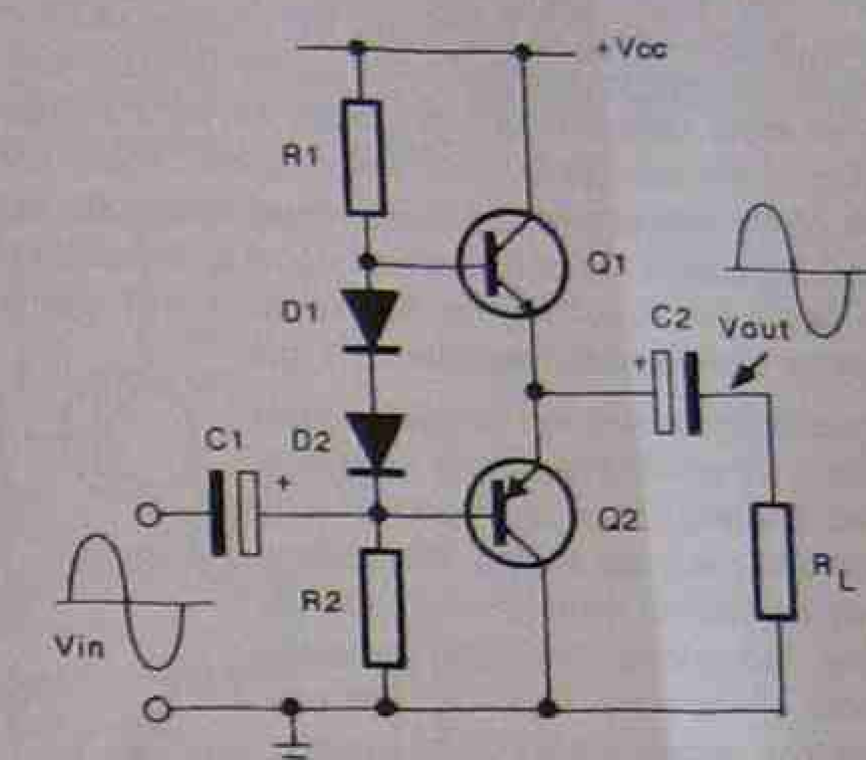


Fig.2: The basic class AB, complementary symmetry amplifier

3. DC CONDITIONS

The DC conditions for the class AB amplifier are shown in Fig.3. Note the following:

- The DC voltage at the junction of the emitters of Q1 and Q2 is half the supply voltage.
- R1 equals R2.
- A quiescent current (I_{CQ}) flows in both transistors.
- The diodes are used to ensure there is a difference of 1.2V between the voltages at the base of Q1 and Q2.
- Both transistors have a forward bias of 0.6V across their base-emitter junctions.
- It is possible to replace the diodes with a resistor. The value of the resistor would need to be such that a 1.2V drop occurs across it. If the resistor value was increased to give more than 1.2V, a higher value of quiescent current would flow in the transistors, pushing the operation towards class A.
- Diodes are preferred to a resistor as the voltage drop across them varies with temperature, keeping the DC conditions constant with a change in temperature.

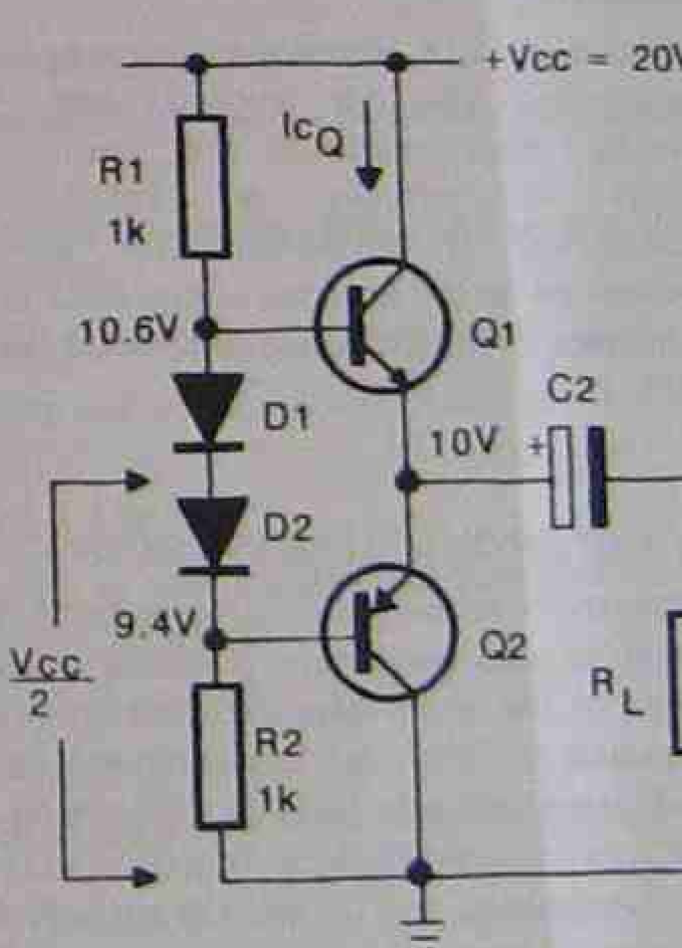


Fig.3: DC conditions

4. **BOOTSTRAPPING**

A practical limitation of the circuit of Fig.3 is that the output signal swing is limited by various losses in the circuit. In particular, the positive half cycle of the output signal can never equal Vcc, as the emitter voltage must always be 0.6V less than the base voltage at Q1. Because current must always flow in R1, there will be a voltage drop across R1 which will prevent the voltage at the base of Q1 from reaching Vcc. There are also losses to prevent the negative half cycle of the output signal from reaching zero volts, but the losses are not as great. Thus, the overall voltage swing of the output signal is limited more by the losses in the positive half cycle than by those occurring for the negative half cycle. To overcome this, a 'bootstrapping' capacitor can be added as shown in Fig.4.

This capacitor provides a form of positive feedback in which the output voltage is applied to the junction of R1 and R2, via a capacitor, adding to the DC voltage already present at their junction. On the positive half cycle, the total potential available to forward bias Q1 will now exceed Vcc, allowing the output signal to virtually reach Vcc. On the negative half cycle, the total bias voltage for Q2 will drop, allowing Q1 to turn on harder, pulling the output closer to zero volts.

Bootstrapping is a term used to describe a condition where the circuit helps itself to achieve the required function. (From the rather impossible analogy of lifting yourself from the ground by your bootstraps!)

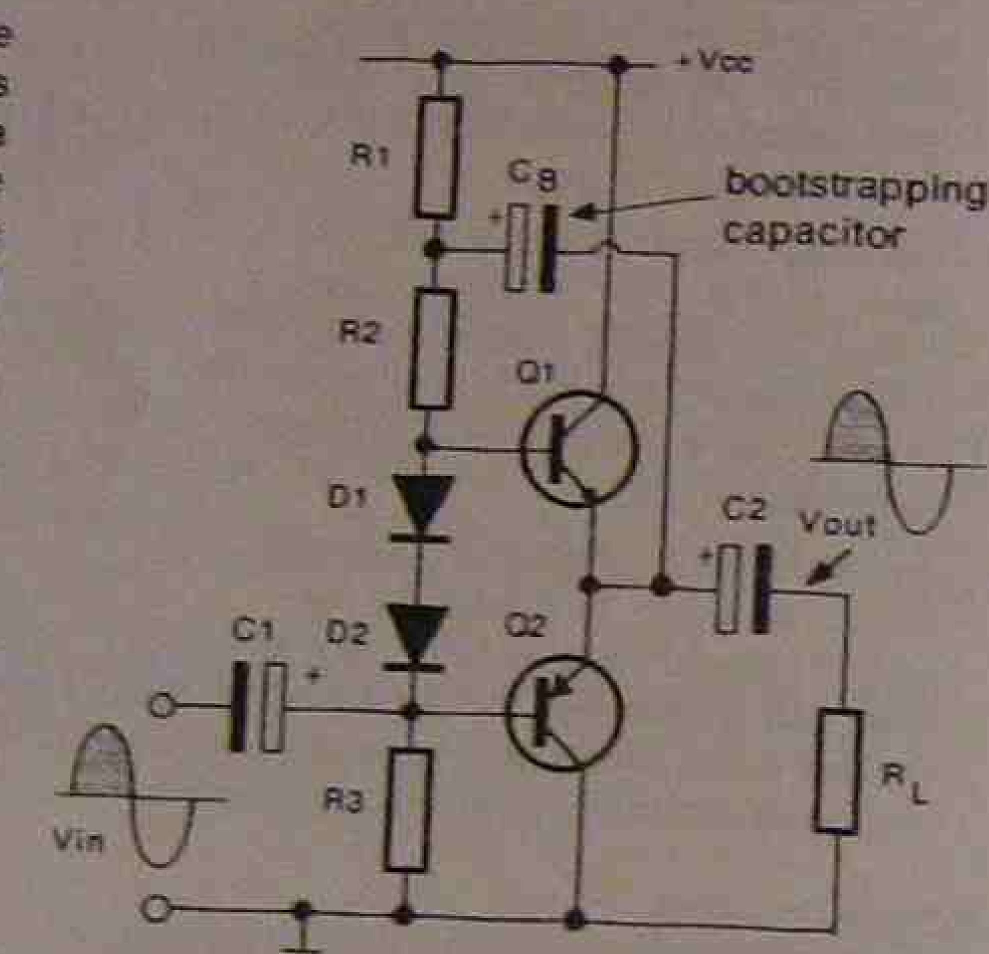


Fig.4: Adding bootstrapping to obtain a greater output signal swing

The circuit of Fig.4 is not the only way bootstrapping can be applied, and numerous other methods are used. However in virtually all cases, bootstrapping is achieved with a capacitor connected from the output back to some point in the circuit. A capacitor is required to give isolation of the DC voltages between the points of connection.

5. **DARLINGTON OUTPUT TRANSISTORS**

To increase the current gain of the output transistors, it is common to use darlington pairs as shown in Fig.5. Because of the extra bias required to overcome two base-emitter junctions, additional diodes are necessary in the biasing network. The DC voltages for a supply of 20V DC are included in Fig.5. Transistors Q1 and Q4 are low power devices, and Q2 and Q3 are high power types. Darlington power transistors, containing the two transistors in the one package are commonly used in power amplifiers.

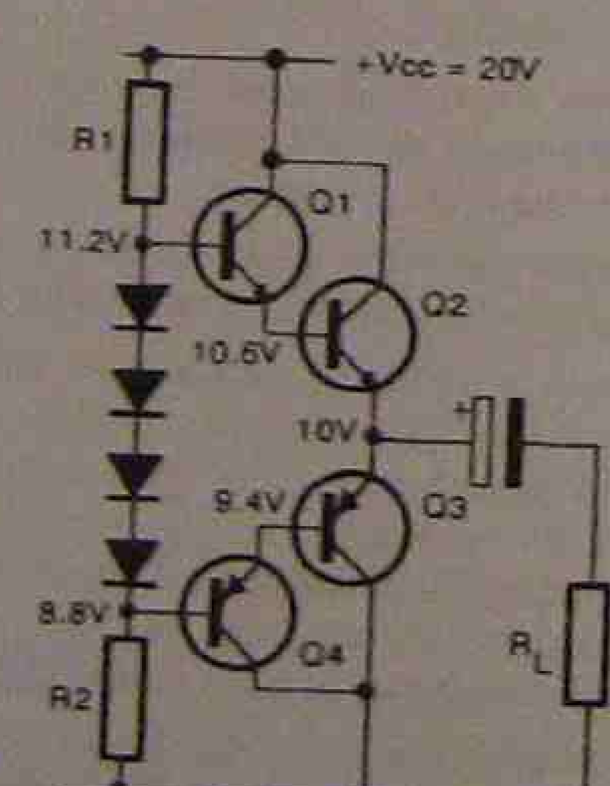
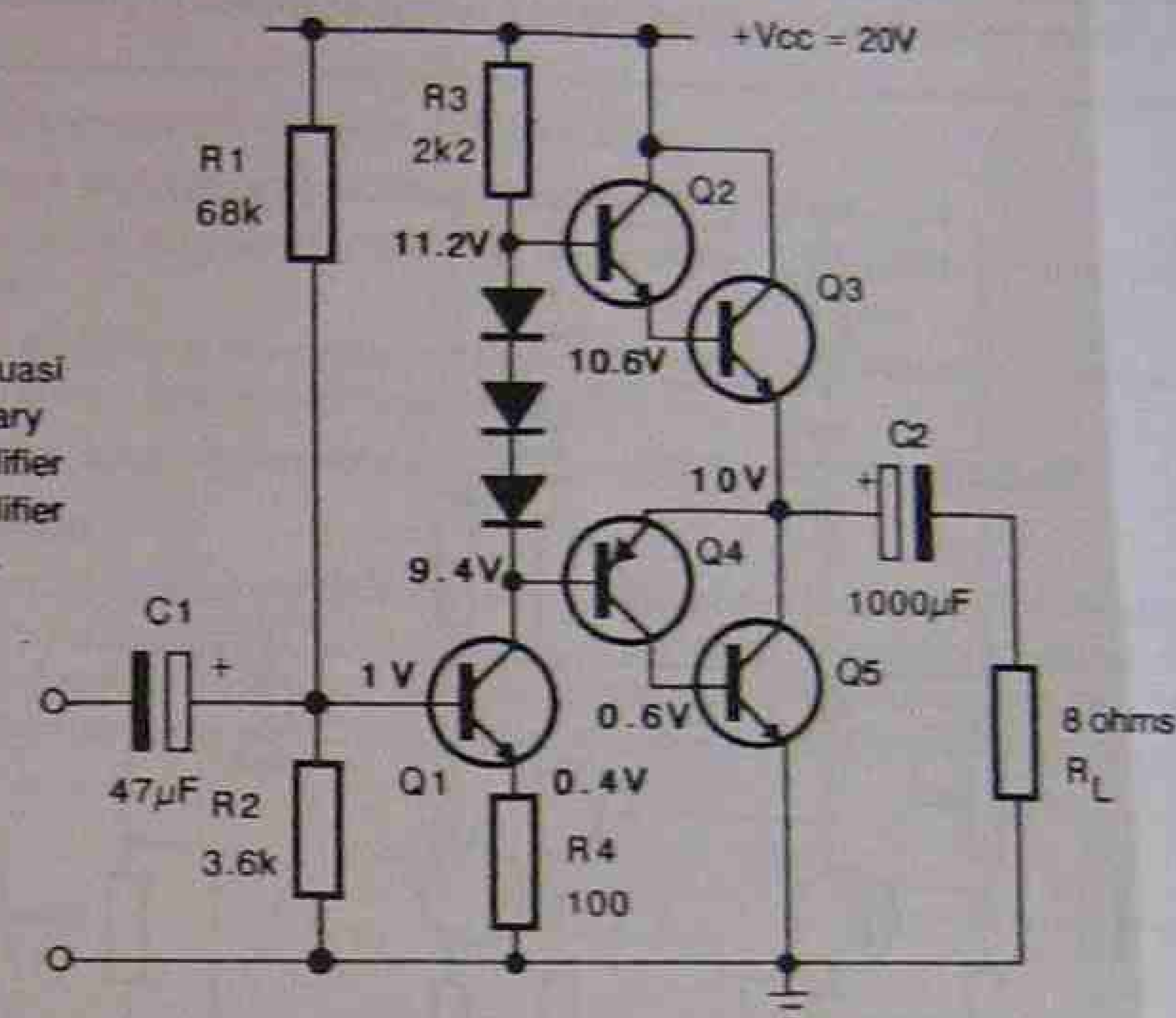


Fig.5: Darlington output

6. **QUASI-COMPLEMENTARY AMPLIFIER**

A variation on using darlington transistors as the output devices is the circuit shown in Fig.6. This circuit is called a 'quasi-complementary symmetry' amplifier, where quasi means 'sort of'. The complementary transistors are Q2 and Q4, which need only be low power transistors making it easier to match the transistors. The power transistors (Q3 and Q5) are both NPN types. This circuit also contains a voltage amplifier/driver stage around Q1.

Fig.6: The quasi complementary power amplifier with an amplifier -driver stage.



The DC voltages are included for the circuit and can be calculated as follows.

- The quiescent DC voltage at the junction of the emitter of Q3 and the collector of Q5 should equal half the supply voltage.
- The voltages at the base, emitter and collector of Q2 to Q5 can be determined on the basis that Vbe = 0.6V.
- The current through R3 can now be determined using Ohm's law. (Equals 4mA)
- The emitter voltage of Q1 can be calculated as the current in R3 also flows in R4. (This assumes the base current of Q2 and Q4 is negligible).
- The base voltage of Q1 equals the emitter voltage plus 0.6V.

Note that a practical circuit will include emitter resistors to help stabilise the operation of the circuit. These are usually small enough not to affect the DC voltages.

7. **POWER OUTPUT CALCULATION**

The maximum output power for all the power amplifier circuits described in these notes can be calculated as described previously. That is:

$$\square \quad P_{out} = \frac{(V_{op\text{-}p})^2}{8R_L}$$

where:

$V_{op\text{-}p}$ = Vcc

.... Potential between the supply rails.

THEORY ASSIGNMENT 14

1. State the basic difference between class B and class AB biasing as applied to power amplifiers.

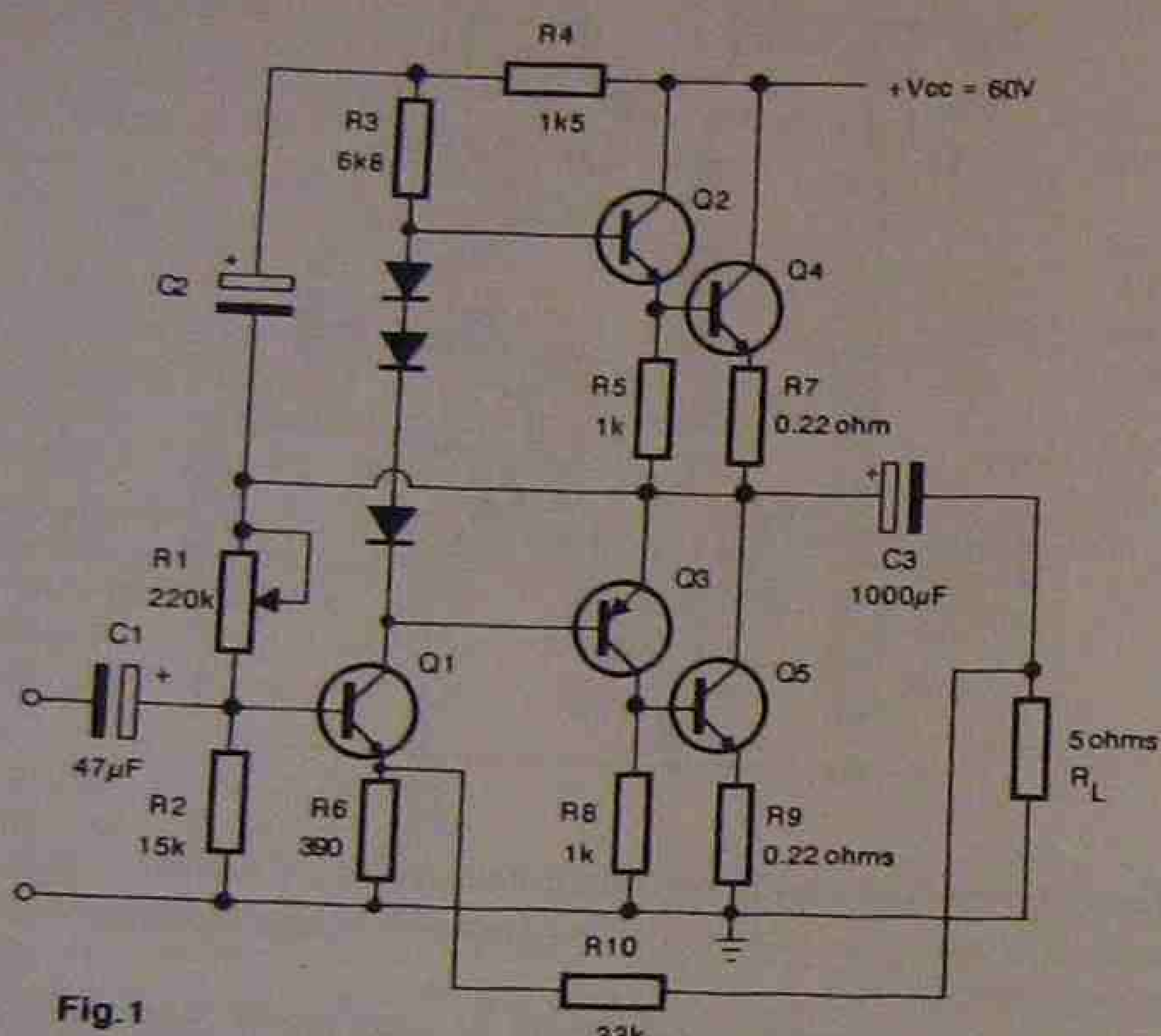


Fig.1

2. For the circuit of Fig.1:
   (a) calculate the approximate value of the DC voltages present at the terminals of each transistor. Assume the collector voltage of Q5 is 30V and that Vbe = 0.6V.
   (b) calculate the theoretical maximum output power of the circuit.
   (c) calculate the approximate value for R1.
   (d) state the purpose of R10.
   (e) state the purpose of C2.
   (f) state the type of circuit configuration.
   (g) state the purpose of Q1.
   (h) state the purpose of R5, R7, R8 and R9.

****

THEORY LESSON 15

References: Electronic Devices, 2nd Ed. Floyd. Chapter 3.

POWER SUPPLIES - PART 1

OBJECTIVES At the end of this lesson you will be able to:

- Draw a block diagram representing a regulated power supply.
- Draw the circuit diagrams for common single phase rectifier/filter circuits, and associated waveforms.
- Calculate the value of filter capacitor required to give a specified output voltage, and explain the effect of this capacitor on diode current.
- Calculate the output voltage and percentage regulation for a power supply.

1. INTRODUCTION

Most electronic equipment requires some form of DC power to operate. If this equipment is to be operated from a standard AC outlet then a DC power supply is needed. Fundamentally, a power supply takes electrical power from a distribution system and converts it to the desired form of power. The conversion of an AC supply to a DC supply is done in four basic steps as shown in Fig.1

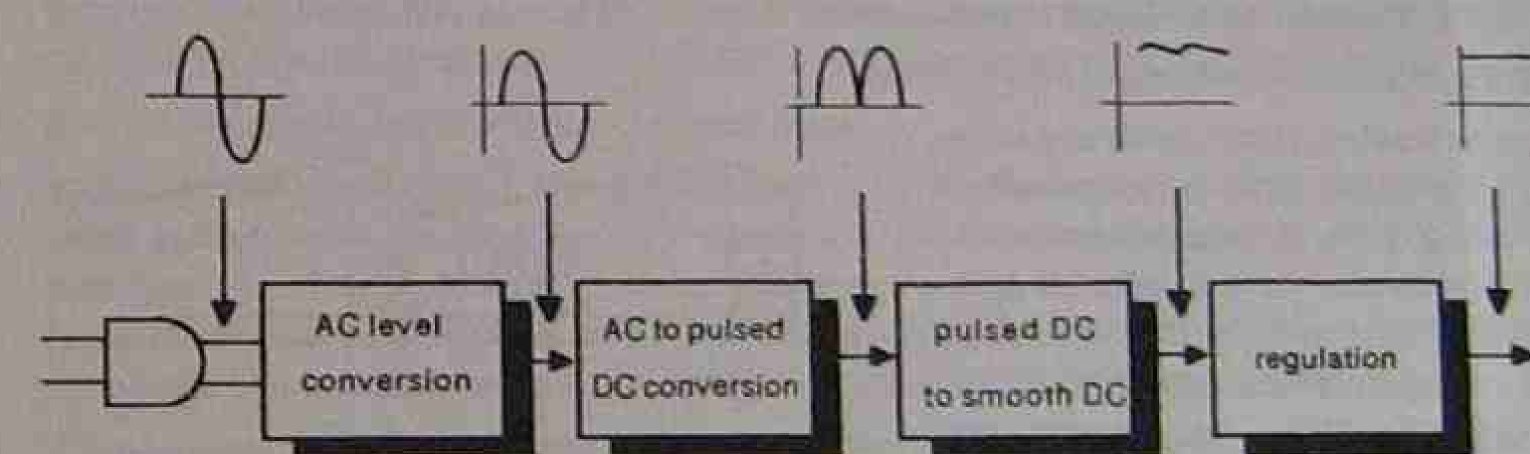


Fig.1: Block diagram of a regulated power supply

In the above block diagram the input is mains supplied 240V AC, 50Hz and the output is a steady DC voltage. The power supply contains a transformer which usually steps down the mains voltage to a level approximately equal to the DC voltage required (by using a transformer maximum power transfer with minimum losses results).

The rectifier section ensures that the sinewave output from the transformer is converted to only positive going DC pulses. These positive pulses are then filtered to minimise the ripple at the top of the resultant waveform, before being regulated to provide a steady DC voltage output. These notes examine typical single phase rectifier circuits and the filter network.

2. REVISION OF THE DIODE

The basic function of a diode is to allow current to flow in one direction and to block it in the other. For a diode to conduct it must therefore be forward biased. In a power supply, substantial currents are often passed by the diodes used in the rectifier section, and a voltage drop of up to 1V is usual for a silicon diode passing several amps. The important specifications for a diode used in a power supply are its forward current rating and its Peak Inverse Voltage rating (PIV). In some critical applications, the switching speed and reverse current of the diode alos need to be considered.

3. SINGLE PHASE HALF WAVE RECTIFIER

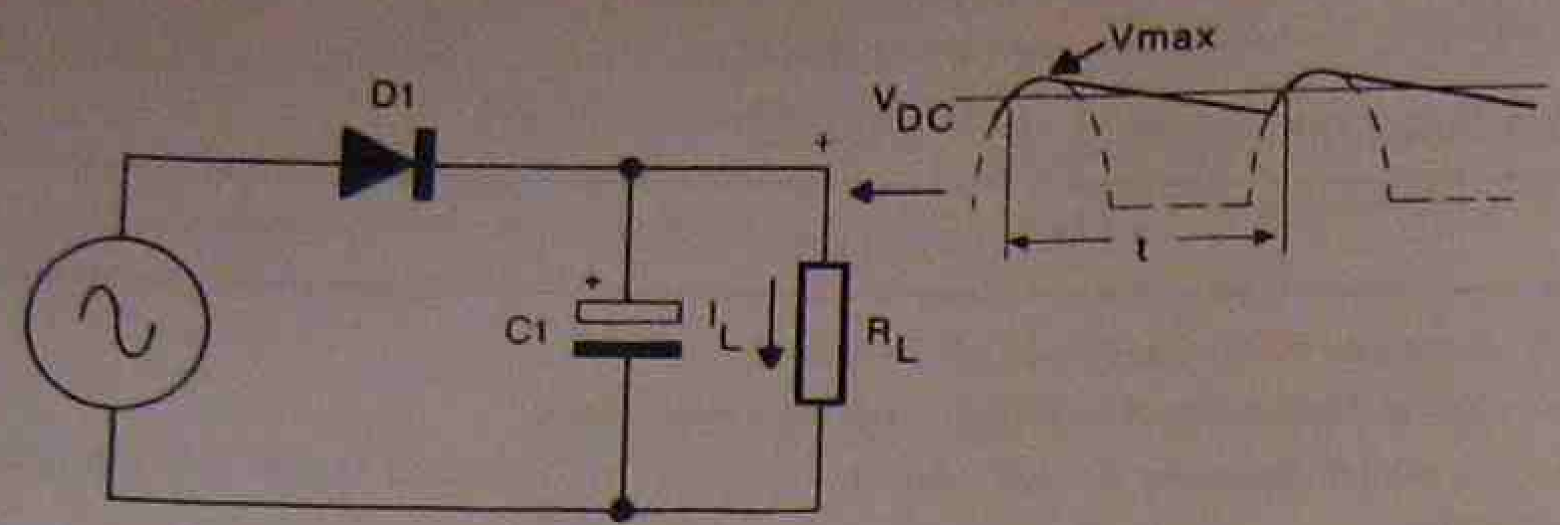


Fig.2: Half wave rectifier with filter capacitor

The circuit of Fig.2 shows a halfwave rectifier with a single capacitor as a filter. The important characteristics of this circuit are:

- only half the input waveform is used
- the ripple frequency (f_r) equals the supply frequency (f_s). Note that frequency equals 1/t, where t = period of waveform.
- the approximate DC output voltage (V_{DC}) equals Vmax, where Vmax = 1.41Vrms. The actual DC output voltage is less than this value and equals Vmax - (V_{ripple}/2). For the purposes of these notes, V_{DC} can be assumed to equal Vmax.
- the PIV the diode must withstand equals Vmax
- the current flowing in the diode is a series of short duration, high value pulses as shown in Fig.3. As an approximation, the current pulses can have a value of up to 10 times the load current. Thus, the diode current rating must be greater than the load current. Also, the transformer secondary current rating should exceed the load current.

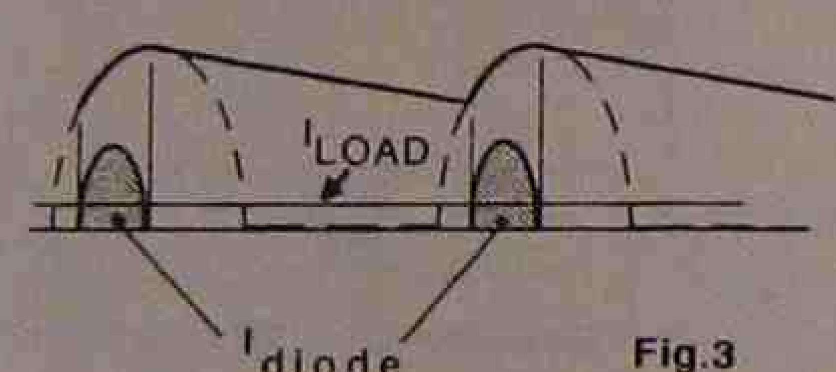


Fig.3

- If the capacitor is removed, the DC output voltage equals 0.318 x Vmax, or 0.45 x Vrms.

The half wave rectifier is only suitable as a supply for loads with a small current consumption as the load voltage will drop significantly as the load current increases. (See definition of load regulation further in these notes.)

4. SINGLE PHASE FULL WAVE RECTIFIER

An improvement over the half wave rectifier is the full wave rectifier where both halves of the AC input cycle are used to produce the DC output. There are two basic circuit configurations:

- the full wave bridge rectifier
- the centre-tapped transformer type

Both of these circuits have particular advantages, and the choice of circuit depends on the application. However, the full wave bridge circuit is more commonly used as it doesn't require a special transformer.

5. THE BRIDGE RECTIFIER

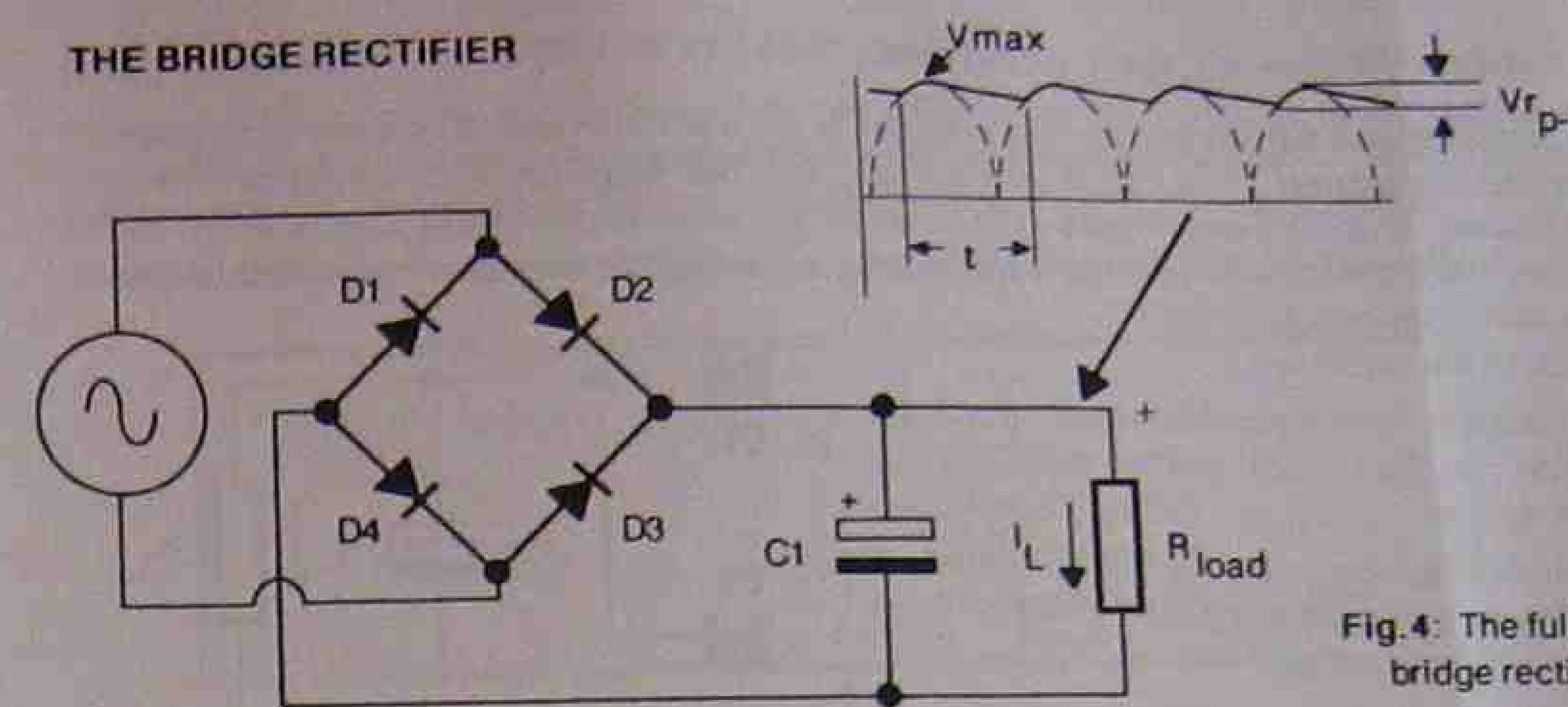


Fig.4: The full wave bridge rectifier

The bridge rectifier is shown in Fig.4 in which four diodes are connected in a bridge configuration across an AC source. Typically, the AC source will be a transformer. The points to note about this circuit are:

- both halves of the input waveform are used.
- the ripple frequency (f_r) equals twice the supply frequency (f_s). Note that frequency equals 1/t, where t = period of waveform. For a 50Hz AC input, the ripple frequency will equal 100Hz.
- as for the half wave circuit, the approximate DC output voltage (V_{DC}) equals Vmax, where Vmax = 1.41Vrms. The actual DC output voltage is less than this value and equals Vmax - (V_{ripple}/2). For the purposes of these notes, V_{DC} can be assumed to equal Vmax.
- the PIV the diodes must withstand equals Vmax
- as for the half wave circuit, the current flowing in the diode is a series of short duration, high value pulses which charge the filter capacitor C1. As an approximation, the current pulses can have a value of around 5 times the load current, which is half that for the half wave rectifier as the current pulses occur twice for every cycle of the input. Again the diode current rating and the transformer current rating must be greater than the load current.
- If the capacitor is removed, the DC output voltage equals 0.636 x Vmax, or 0.9 x Vrms.
- The circuit operates in the following way:
  - for the positive half cycle of the input, diodes D2 and D4 conduct, with current flowing from positive to negative through the filter capacitor, charging it with the polarity shown.
  - for the negative half cycle, diodes D1 and D3 conduct, causing the current to again flow from positive to negative through the filter capacitor.
  - the filter capacitor supplies a steady value of current to the load, thereby partially discharging between each pulse of current supplied as described above.
  - the current in the secondary of the transformer alternates in direction. Thus, AC flows in the transformer secondary and DC flows in the load.

The bridge rectifier has the disadvantage that the current flows through two series connected diodes, causing a voltage drop of up to 2V. The advantage is simplicity, as a transformer with a single secondary winding can power the circuit.

Rather than use four individual diodes, packages containing four diodes connected as a bridge are usually employed. These are available with current ratings from 1A to 50A or more, in which the higher current types are packaged in a metal casing suitable for mounting on a heatsink. A bridge rectifier passing 50A or so can dissipate up to 100W of heat, requiring a large heatsink to avoid its destruction.

6. THE CENTRE-TAPPED FULL WAVE RECTIFIER

The full wave, centre-tapped transformer rectifier is shown in Fig.5 in which two diodes are connected to a centre-tapped transformer. This circuit is effectively two half wave rectifiers connected to a common transformer, in which current flows in one half of the secondary winding at a time, in the directions shown. The current in the primary is alternating, as a result of the combined effect of the currents in the secondary winding. The points to note about the circuit are:

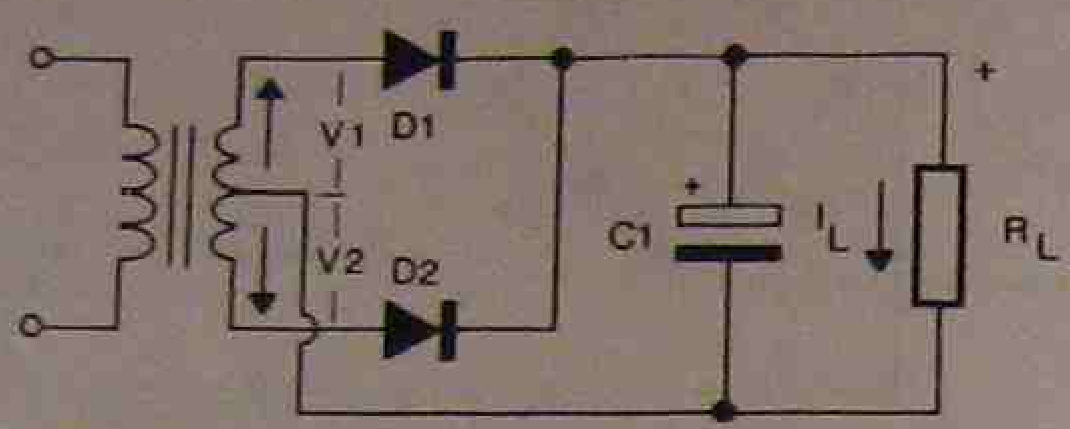


Fig.5: Centre-tapped transformer, full wave rectifier

- The output waveform is identical to that from the bridge rectifier.
- the ripple frequency (f_r) is twice the supply frequency (f_s) as for the bridge rectifier.
- the approximate DC output voltage (V_{DC}) equals Vmax, where Vmax = 1.41Vrms. In this circuit Vrms = V1 = V2.
  As before, the actual DC output voltage is less than this value and equals Vmax - ($V_{ripple}/2$). For the purposes of these notes, V_{DC} can be assumed to equal Vmax.
- the PIV the diodes must withstand equals 2 x Vmax. (where Vmax = V1 or V2.) For example, if V1 = 10V RMS, the peak inverse voltage the diodes must withstand = 2 x 1.41 x 10V = 28.4V.
- as for the bridge rectifier circuit, the current flowing in the diodes is a series of short duration, high value pulses. This current charges the capacitor, which in turn supplies current to the load.
- If the capacitor is removed, the DC output voltage equals 0.636 x Vmax, or 0.9 x Vrms, where Vrms = V1 or V2.
- The circuit operates in the following way:
  - for the positive half cycle of the input, diode D1 conducts, with current flowing through the top section of the secondary then from positive to negative through the filter capacitor. The filter capacitor supplies current to the load as described for the bridge rectifier.
  - for the negative half cycle of the input, diode D2 conducts and current now flows in the lower section of the secondary winding in the direction shown. Thus the charging current through the filter capacitor is in the same direction as before.

This circuit has the disadvantage that a transformer with a centre tapped winding is required, usually resulting in a larger transformer than for the bridge rectifier. However, there are less losses, as the current flows through a single diode rather than two series connected diodes as in the bridge rectifier.

The following table summarises the three rectifier circuits. Note that they all produce the same output voltage, although their characteristics vary as described in these notes.

Summary of the half and full wave rectifier circuits

Circuit	V_{DC}	PIV of diodes	ripple frequency
half wave	V_{MAX}	V_{MAX}	f_{SUPPLY}
bridge	V_{MAX}	V_{MAX}	2 x f_{SUPPLY}
C.T full wave	V_{MAX}	$2V_{MAX}$	2 x f_{SUPPLY}

7. CALCULATING THE SIZE OF THE FILTER CAPACITOR

In the preceding circuits, a capacitor is used as a filter, in which it charges to the peak of each output half cycle and slowly discharges through the load between the peaks. The ripple voltage (Vr) produced is dependent on the size of the load resistance and filter capacitance.

The output voltage of a rectifier circuit can be calculated using the following expression:

☐ $V_{DC} = V_r f_r R_L C$ where: V_{DC} = DC output voltage
V_r = p-p ripple voltage
f_r = ripple frequency
C = capacitance
R_L = load seen by the capacitor

Example: Calculate the load resistance and capacitor size of a full wave rectifier that supplies 40V DC with a 2% ripple voltage (peak to peak) at 250mA to a resistive load. Assume the rectifier circuit is supplied with 50Hz AC.

Solution:

$$R_L = \frac{V_O}{I_L} = \frac{40}{0.25} = \underline{160 \text{ ohms}}$$

$$V_r = 40V \times \frac{2}{100} = \underline{0.8V}$$

$$V_{DC} = V_r f_r R_L C \quad \text{or} \quad C = \frac{V_{DC}}{V_r f_r R_L} = \frac{40}{0.8 \times 100 \times 160} = \underline{3125\mu F}$$

It should be noted that diodes in a rectifier circuit only become forward biased when the supply exceeds the capacitor voltage. The capacitor effectively is "pumped" up to the peak voltage at every half cycle, causing high peak diode currents to flow.

8. LOAD REGULATION

A measure of how well a DC output can be maintained regardless of changes in the loading conditions is called Load Regulation. It is calculated using the difference between full and no load conditions and is usually expressed as a percentage where:

$$\square\ \%\ \text{load Regulation} = \frac{V_{DC}(\text{no load}) - V_{DC}(\text{full load})}{V_{DC}(\text{full load})} \times \frac{100}{1}$$

Thus, a power supply with a no load voltage of 20V and a full load output of 18V has a load regulation of: $\frac{20 - 18}{18} \times 100 = \underline{11.1\%}$

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES.

THEORY ASSIGNMENT WEEK 15

1. Sketch the block diagram of a typical power supply, label each block, and briefly describe its main function.

2. A single phase, full wave bridge rectifier with a filter capacitor supplies 24 volts at 750mA to a load. Calculate the value of load resistance and filter capacitance if the ripple is 1.5% of the DC output voltage. Assume a 50Hz supply frequency.

3. The no load voltage of a power supply is measured at 48V DC. With a resistive load connected the voltage is measured at 42V DC. Determine the % load regulation.

4. For the circuit of Fig.1:

   (a) Identify the circuit configuration.

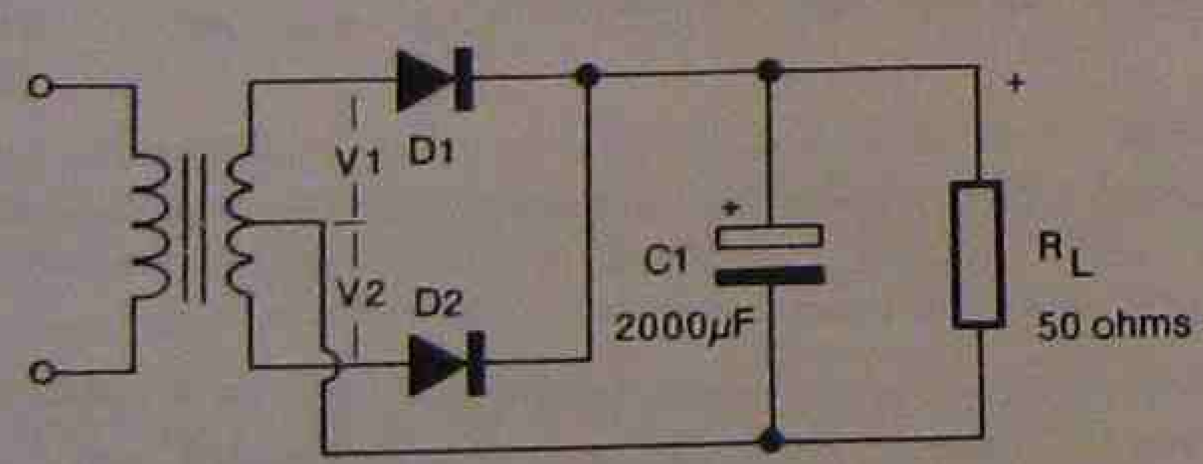
   (b) Determine the peak voltage across each half of the secondary of the transformer.

   (c) Sketch the waveform across the load.

   (e) Calculate the approximate DC voltage across the load.

   (f) Calculate the PIV the diodes must withstand.

   (g) Calculate the approximate DC output voltage if capacitor C1 is disconnected.



V1 = V2 = 10VRMS Fig.1

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

THEORY LESSON 16

References: Electronic Devices, 2nd Ed. Floyd. Chapter 17.

POWER SUPPLIES - PART 2

OBJECTIVES At the end of this lesson you will be able to:

- Explain the operation of a basic discrete component series voltage regulator.
- Calculate the output voltage, current limit value and transistor power dissipation for a basic series regulator.
- List the advantages and characteristics of a three terminal fixed voltage regulator.

1. INTRODUCTION

The purpose of voltage regulation is to maintain a constant voltage across a load regardless of any change in load conditions or supply voltage. The main types of regulation are series, shunt and switching, but for the purpose of studying the principles of regulation, only series regulators will be discussed.

The basic regulating device is the zener diode, which will be described in detail in another subject. A brief summary of this device is included along with a simple transistor series regulator. The three terminal fixed voltage regulator is also described, as these devices find considerable application due to their simplicity of use.

2. THE ZENER DIODE

A zener diode is a diode capable of conducting in both directions as depicted in Fig.1. When it is forward biased, (anode positive, cathode negative) the zener diode behaves as a conventional diode, with a forward voltage drop of around 0.6 to 1V, depending on the forward current (IF) passing through it (Fig.1(a)). If the voltage is reversed, current will flow in the reverse direction, providing the reverse voltage exceeds the zener voltage of the device. As shown in Fig.1(b), a current referred to as the zener current (Iz) flows from cathode to anode. Zener voltages can range from 3V up to several hundred, although values around 5V to 40V are more common. The maximum value of the zener current depends on the power rating of the zener diode and is usually limited to less than 1amp. Typical power ratings are 400mW, 1W and 5W.

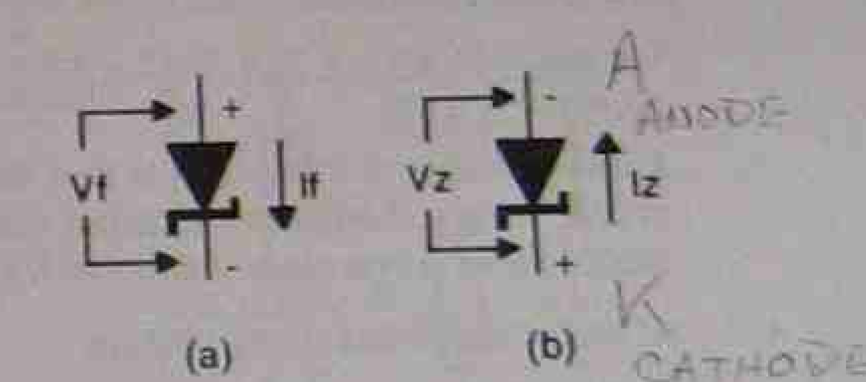


Fig.1: The zener diode

The important feature of the zener diode is that the reverse voltage is relatively constant regardless of the value of the zener current. It is this feature that makes the zener useful as a voltage regulator. The basic series zener regulator is shown in Fig.2. In this circuit:

- the output voltage across the load RL equals the zener voltage Vz.
- The output voltage is kept constant despite changes in Vin and IL because the zener current varies to change the voltage across the series resistor Rs to compensate. An increase in IL will cause a decrease in Iz, and a decrease in Vin will cause Iz to drop. In both cases, the change in the voltage drop across Rs will maintain Vo at a constant value. The limit of operation of the circuit is determined by the power rating of the zener diode.

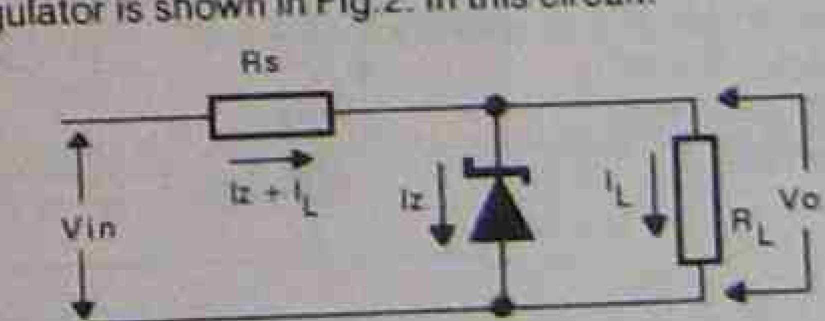


Fig.2: Basic series zener regulator

3. **SERIES TRANSISTOR REGULATOR**

To increase the power handling capability and to improve the regulating characteristics the zener diode can be used with a series pass transistor as shown in Fig.3.

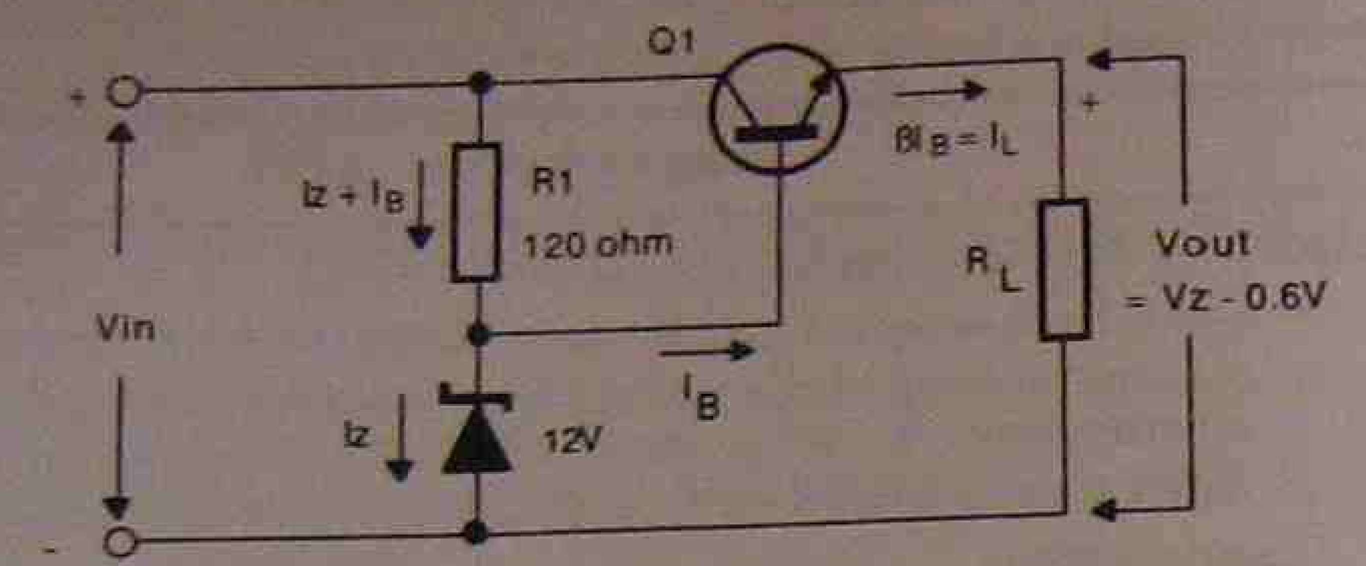


Fig.3: Basic series regulator using a series pass transistor and a zener diode.

In the circuit of Fig.3, the output voltage equals the zener voltage (Vz) less Vbe (Vz - 0.6V). For the values shown, Vout will equal 11.4V. Resistor R1 supplies current to the zener diode and to the base of the transistor. The circuit relies on the zener voltage and Vbe remaining constant despite changes in the input voltage and the load current. An increase in the load current will cause the zener current to reduce, as more base current is required by the transistor. If the input voltage changes, the zener current will also change, but the zener voltage will remain constant, keeping the output voltage constant. The input voltage must therefore be higher than the required output voltage by at least 2V.

The power dissipated by the series pass transistor equals $I_L \times V_{CE}$ where V_{CE} = Vin - Vout.

For example, if Vin = 20V, Vout = 11.4V and I_L = 1A, the power dissipated by Q1 will equal (20 - 11.4) x 1 = 8.6W.

A disadvantage of this circuit is that there is no feedback between the load and the regulating circuit. Thus a change in the output voltage for a particular reason will not be corrected, as the circuit works on the assumption that the output voltage can't change.

4. **REGULATORS WITH FEEDBACK**

To allow feedback an amplifier is required, in which a reference voltage is fed to one terminal and a sample of the output is fed to the other. If these two voltages are different, the amplifier will change the base current it supplies to the transistor to correct the error. Fig.4 shows the basic circuit of such a regulator.

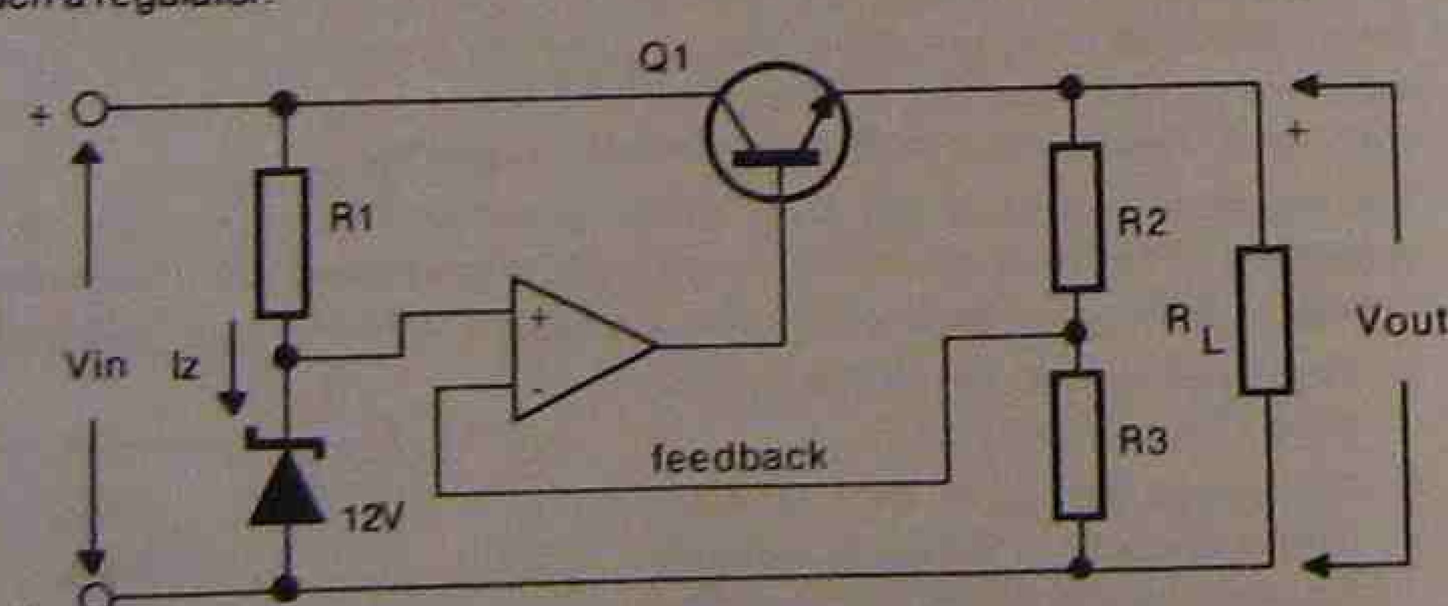


Fig.4: Series regulator with feedback

In this circuit, the reference voltage (Vz) is connected to the terminal marked +, and a sample of the output is taken from the junction of R2 and R3 to the terminal marked -. In Fig.4, if R2 equals R3, the output voltage will equal 24V, assuming the zener voltage is 12V.

As long as Vin is several volts higher than the required output voltage, both Vout and the power dissipation (Pd) of the series pass transistor can be calculated. The equations are:

- $Vout = (\frac{R2}{R3} + 1) \times Vref$ where: Vref = Vz
- $Pd = (Vin - Vout) \times I_L$

Note: The current in R2 and R3 can usually be ignored

This circuit produces a regulated output voltage that will have reduced ripple. If a potentiometer is connected between R2 and R3 with the feedback taken from the wiper of the pot, the output voltage can be varied. However the circuit has no protection if a short circuit is applied across the output. In this case the series pass transistor will probably burn out as the regulator will attempt to compensate for the drop in the output voltage by turning on the transistor to allow it to supply as much current to the output as it possibly can. To overcome this, some form of current limiting is required.

5. **CURRENT LIMITING**

The circuit of Fig.5 is identical to that of Fig.4 except current limiting has been added with Rsc and Q2.

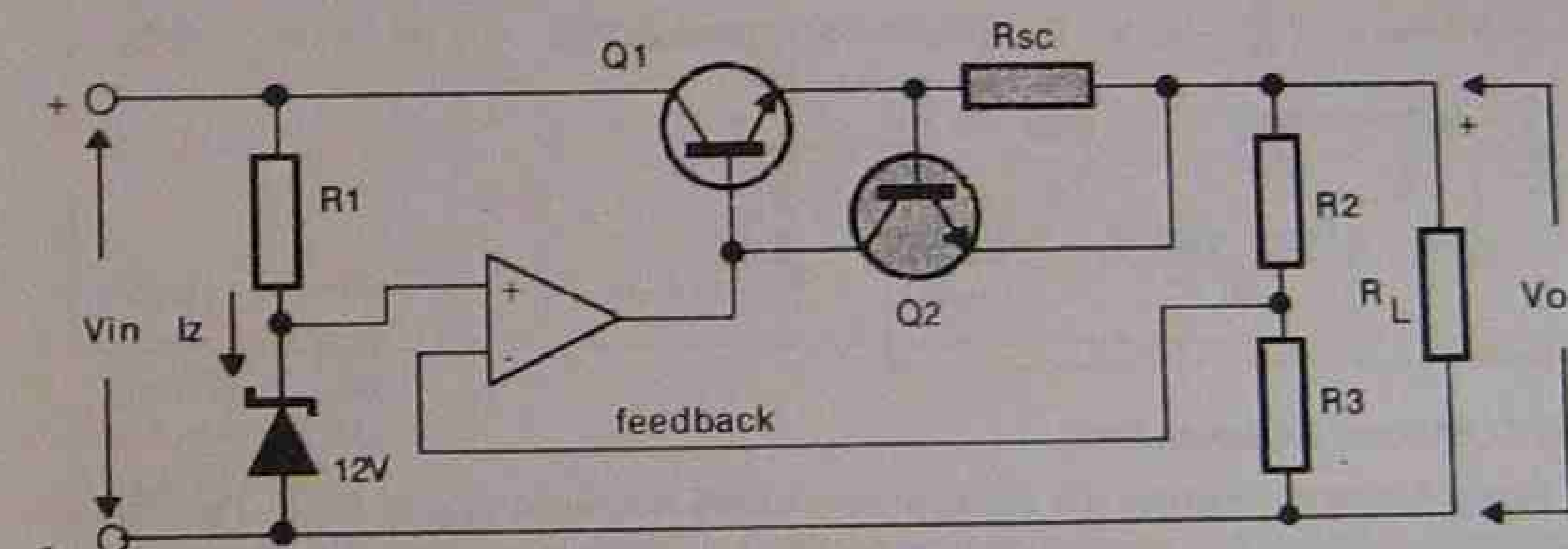


Fig.5: Series regulator with current limiting

In Fig.5:

- The load current passes through Rsc.
- If the voltage developed across Rsc exceeds 0.6V, Q2 will be turned on as it is now forward biased by the voltage drop across Rsc.
- When Q2 turns on, it will rob Q1 of base current, turning Q1 off. As a result, the load current will be limited to a value that produces 0.6V across Rsc. Thus:

$$Isc = \frac{0.6V}{Rsc}$$

EXAMPLE: For the circuit of Fig.6, calculate:

(a) The maximum and minimum output voltages

(b) The current that will flow if the output is short-circuited

(c) The power dissipated by Q1 if the output is short-circuited

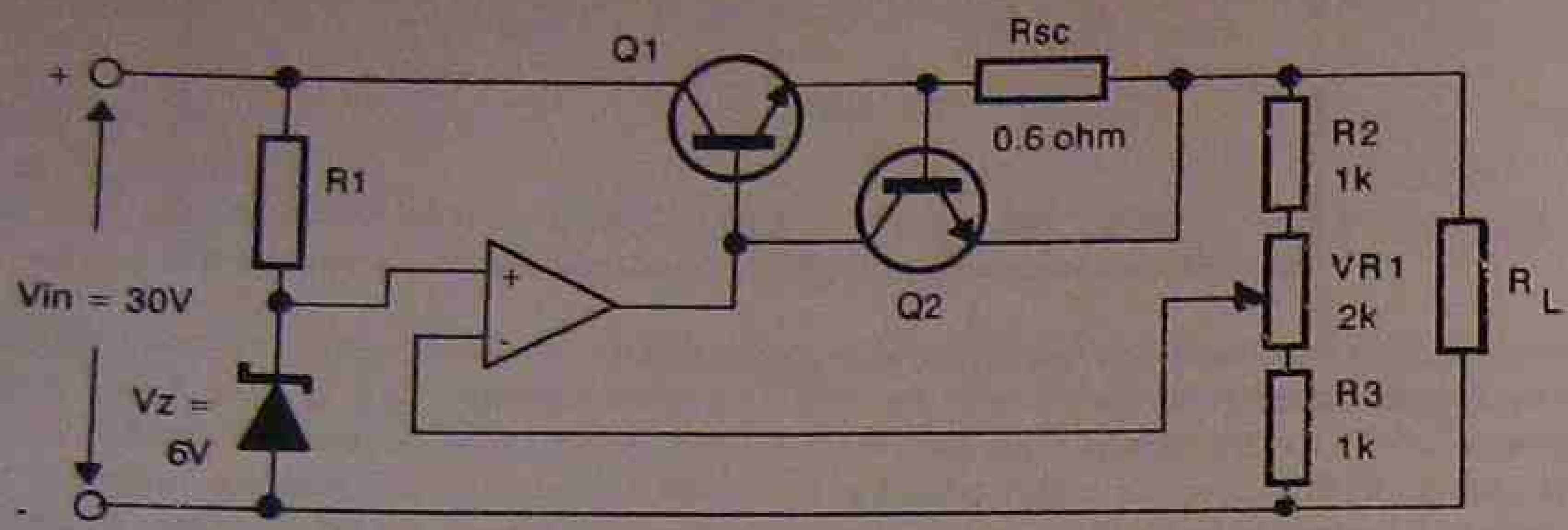


Fig.6

Solution

(a) $V_{out} = (\frac{R2}{R3} + 1) \times V_{ref}$ where: $V_{ref} = V_z$

$V_{omin} = (\frac{1k}{3k} + 1) \times 6$ (wiper moved towards R2) $= (1.33) \times 6 = \underline{8V}$

$V_{omax} = (\frac{3k}{1k} + 1) \times 6$ (wiper moved towards R3) $= (4) \times 6 = \underline{24V}$

(b) $I_{sc} = \frac{0.6V}{R_{sc}} = 0.6V = \underline{1A}$

(c) $P_d = (\underline{V_{in} - V_{out}}) I_L = 1(\underline{30 - 0}) = \underline{30W}$

6. THREE TERMINAL REGULATORS

Three terminal voltage regulator ICs incorporate circuitry similar to that of Fig.5. They have internal current limiting, thermal shutdown protection and come in a range of package styles and output voltages. They are extensively used as they simplify the design of power supplies. The most common fixed voltage three terminal regulators are the 78XX series (positive output) and the 79XX series (negative output). These regulators can pass currents of up to 1.5A, but other regulator types can pass up to 10A (LM 338).

Fig.7 shows the pin connections for the TO-220 and TO-3 packages (78XX series) and the TO-220 style 79XX series. Note the different pin connections for the 78XX and 79XX, TO-220 packages. Other smaller packages are also manufactured for low power use.

TYPE	POL	CASE	$V_{IN (max)}$	I_{OUT} (A) (nom)	P_{TOT} (W)	TOL %	SIMILAR TYPES
78XX	POS	TO-220	35	1		4	LM340 - T XX
79XX	NEG	TO-220	−35	1		4	LM320 - T XX
78H05	POS	TO-3	20	5	50	4	LM323
LM317	POS	TO-3	V_{OUT}+40V	1.5	20		78G

Note: Replace XX by output voltage e.g. - 7812, 7905 etc

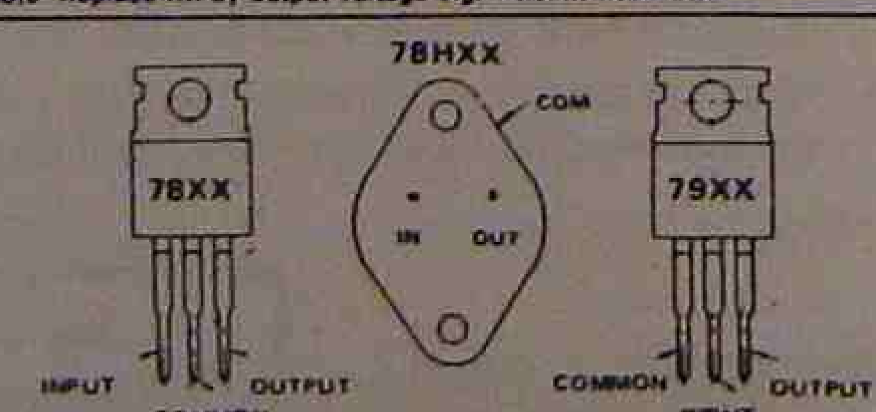


Fig.7: Three terminal, fixed voltage regulators

The main features of three terminal regulators are:

- very low quiescent currents
- minimal external components needed
- internal thermal overload protection
- internal short circuit current limiting
- excellent regulation characteristics
- high ripple rejection
- the typical range of output voltages are: 5V, 6V, 8V, 8.6V, 10V, 12V, 15V, 18V, 22V and 24V. A 5V positive regulator would be marked as type 7805, and a 7905 is a 5V negative regulator. These values apply to the 78XX series, as the 79XX series has a smaller range of output voltages.

Fig.8 shows the circuit of a 5V regulated supply using a 7805 voltage regulator. Note that a small value capacitor is connected across the output. If the regulator is mounted remotely from the main filter capacitor, another small value capacitor (0.1uF ceramic) should be connected across the input terminals of the regulator. These two capacitors improve the regulating and ripple rejection characteristics of the regulator.

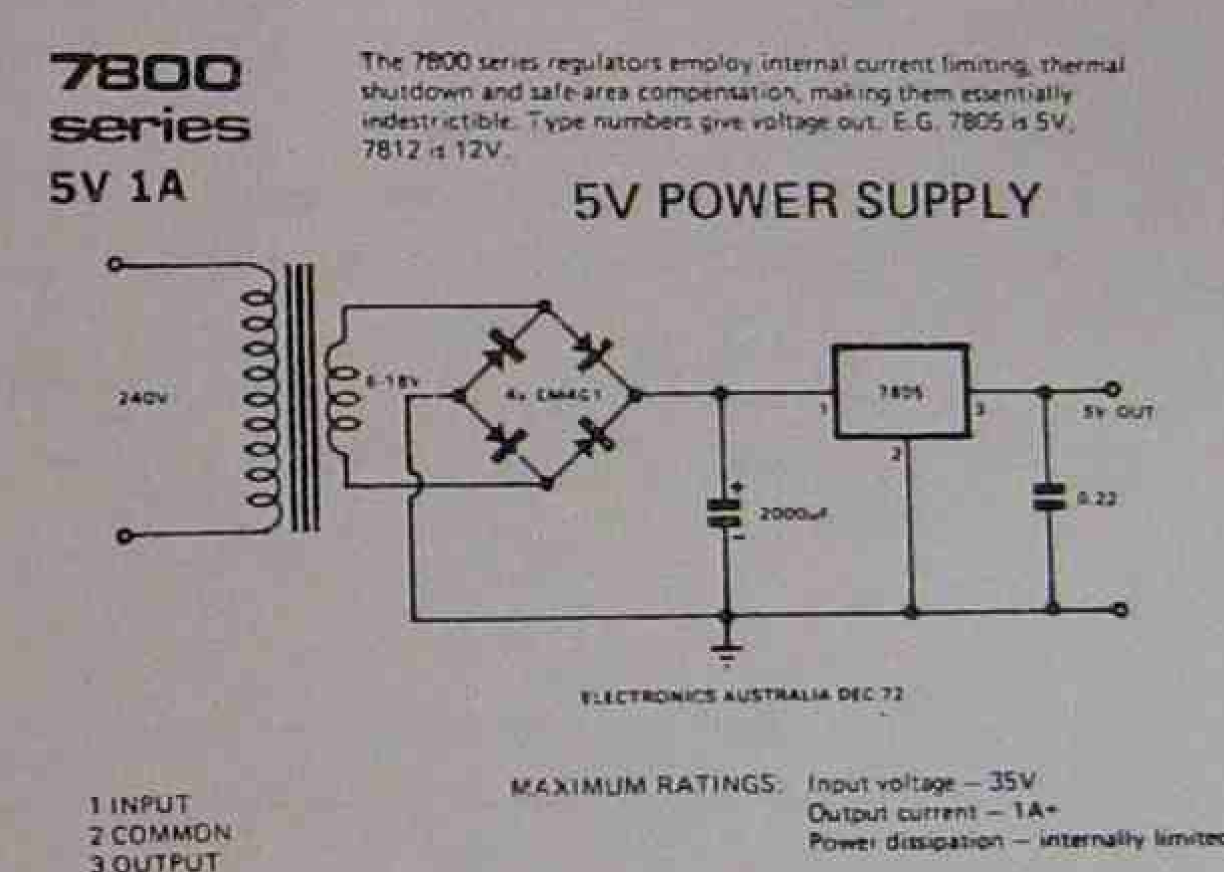


Fig.8: A 5V power supply suitable for powering a TTL digital circuit

When using a three terminal regulator, the input DC voltage must be at least 2V higher than the output of the regulator. Thus a 5V regulator must be supplied with at least 7V DC. If ripple on the input causes the voltage differential across the regulator to drop below 2V, the output voltage will drop, as regulation is no longer provided. However the power dissipation of the regulator will increase as the voltage across the regulator is increased, so a compromise between stable regulation and heat dissipation is required. In most cases a heat sink should be fitted to the regulator. The topic of voltage regulators will be covered more fully in other subjects within the course.

Variable voltage three terminal regulator ICs are also manufactured, and a complete regulated power supply can be constructed using one of these devices. These regulators will also be covered in detail in other subjects.

****