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

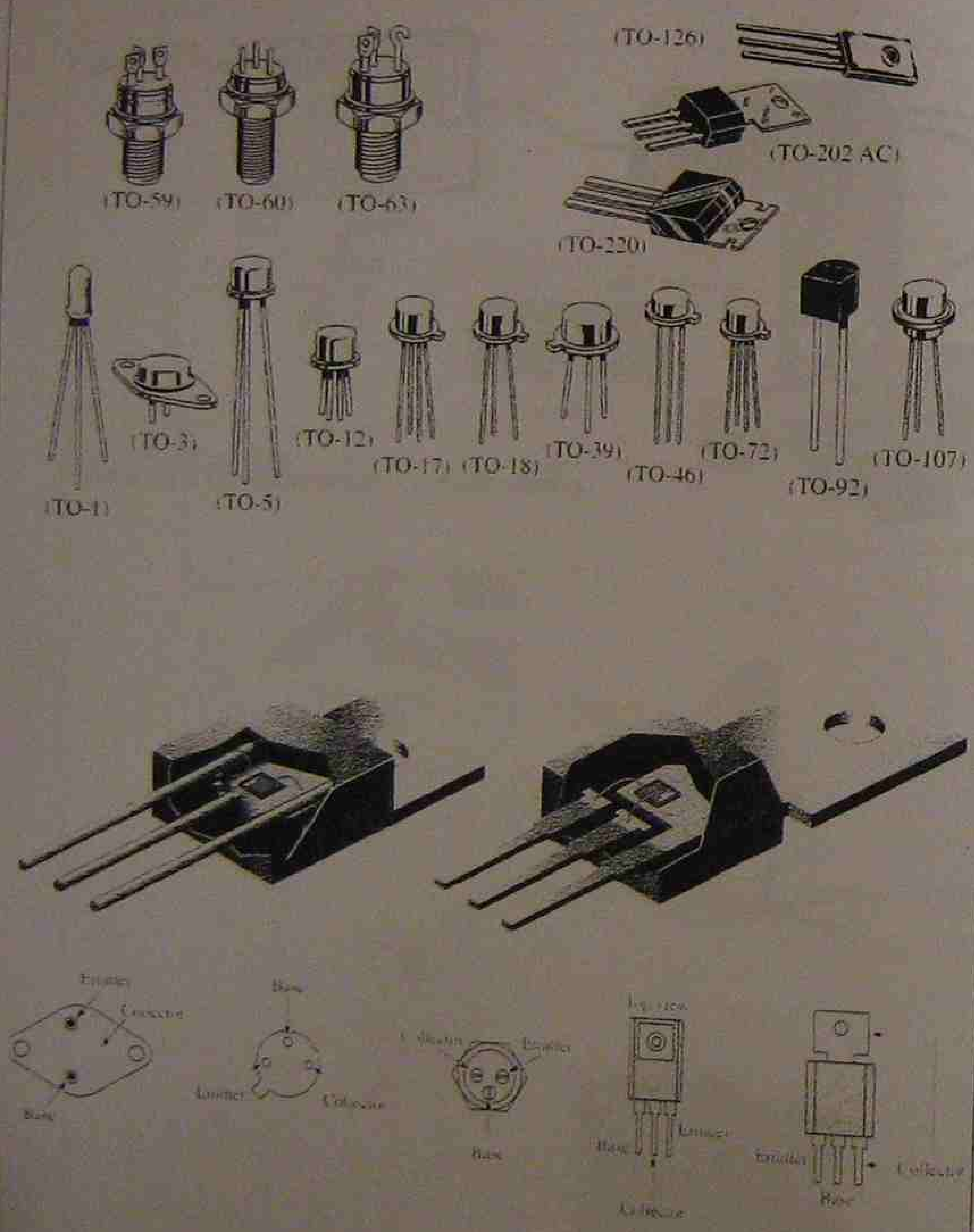


Figure 1.6 An IC amplifier (operational amplifier)

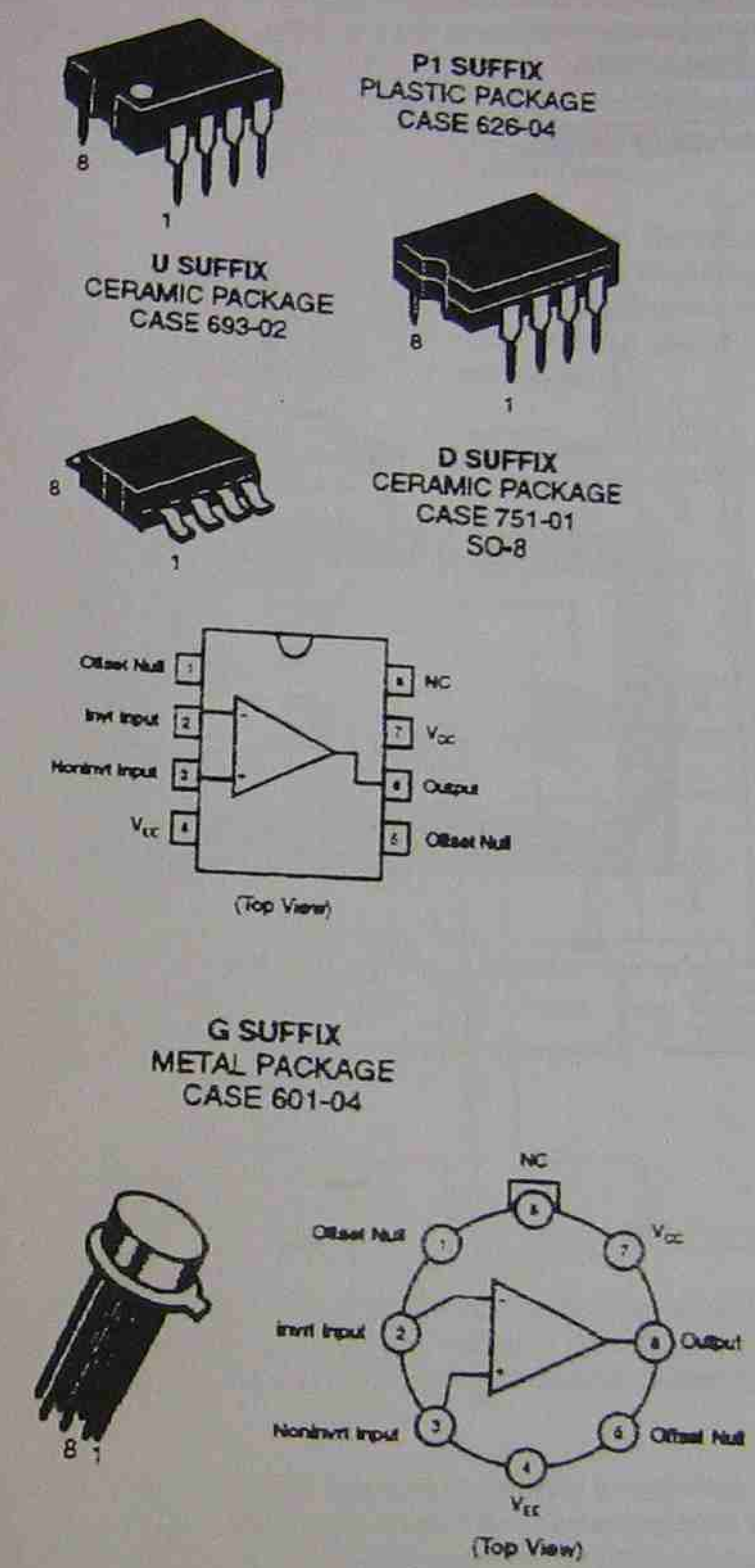
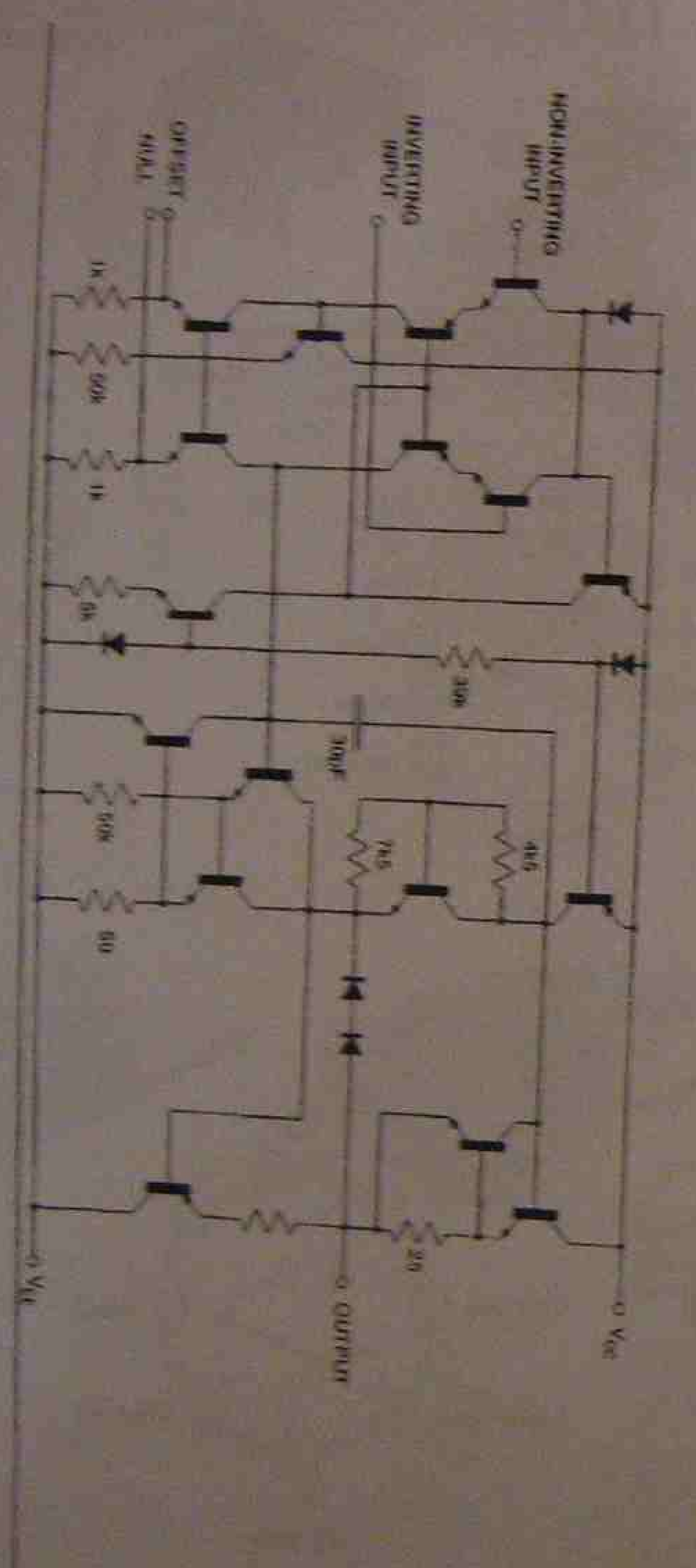


Figure 1.7 shows the internal circuit diagram of an LM741 general purpose operational amplifier. It consists of transistors, resistors, diodes and a capacitor. All discrete components are fabricated in a tiny silicon chip and typical packages are shown in Figure 5 and 6.

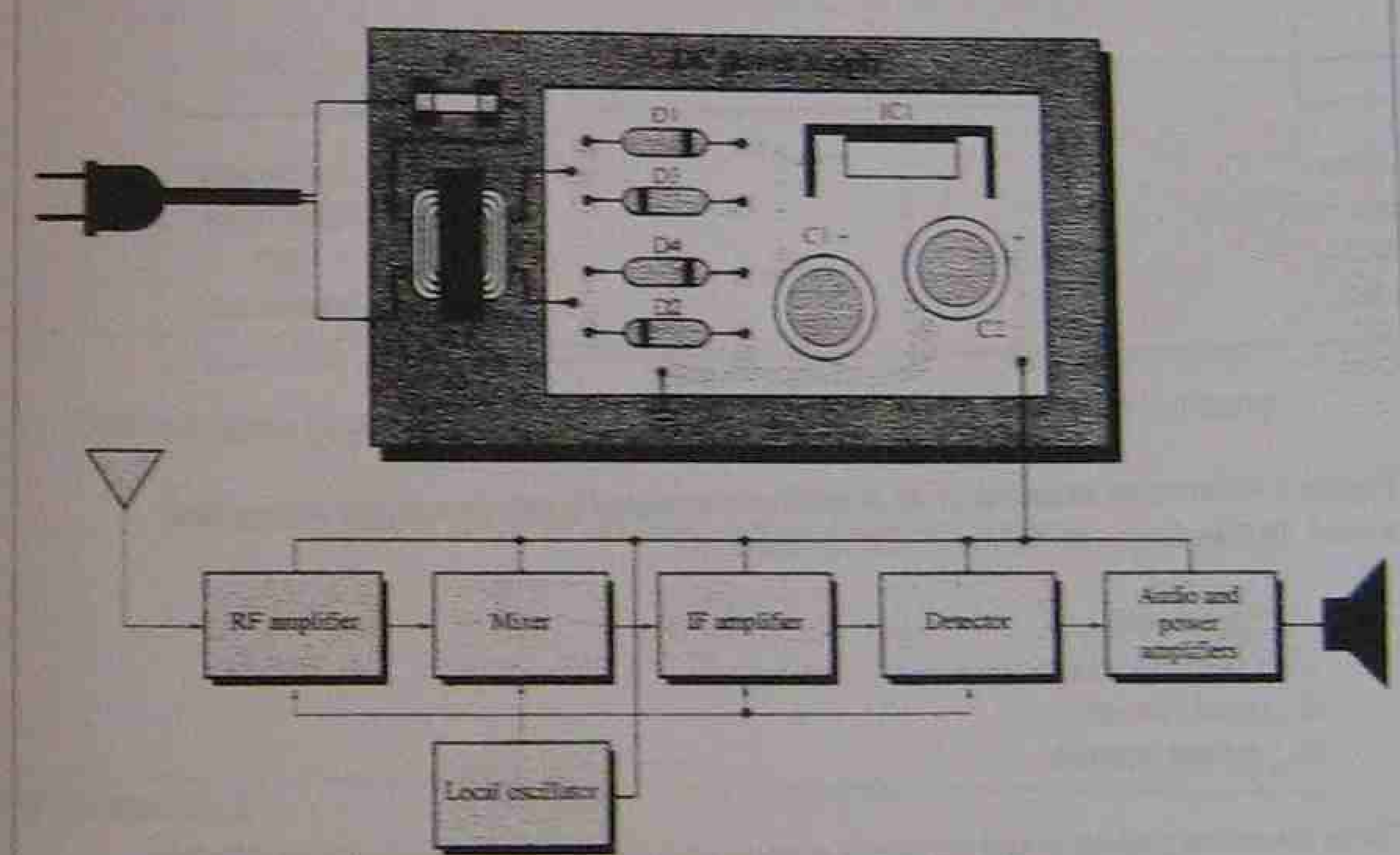
Figure 1.7 LM741 operational amplifier



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

Figure 1.9

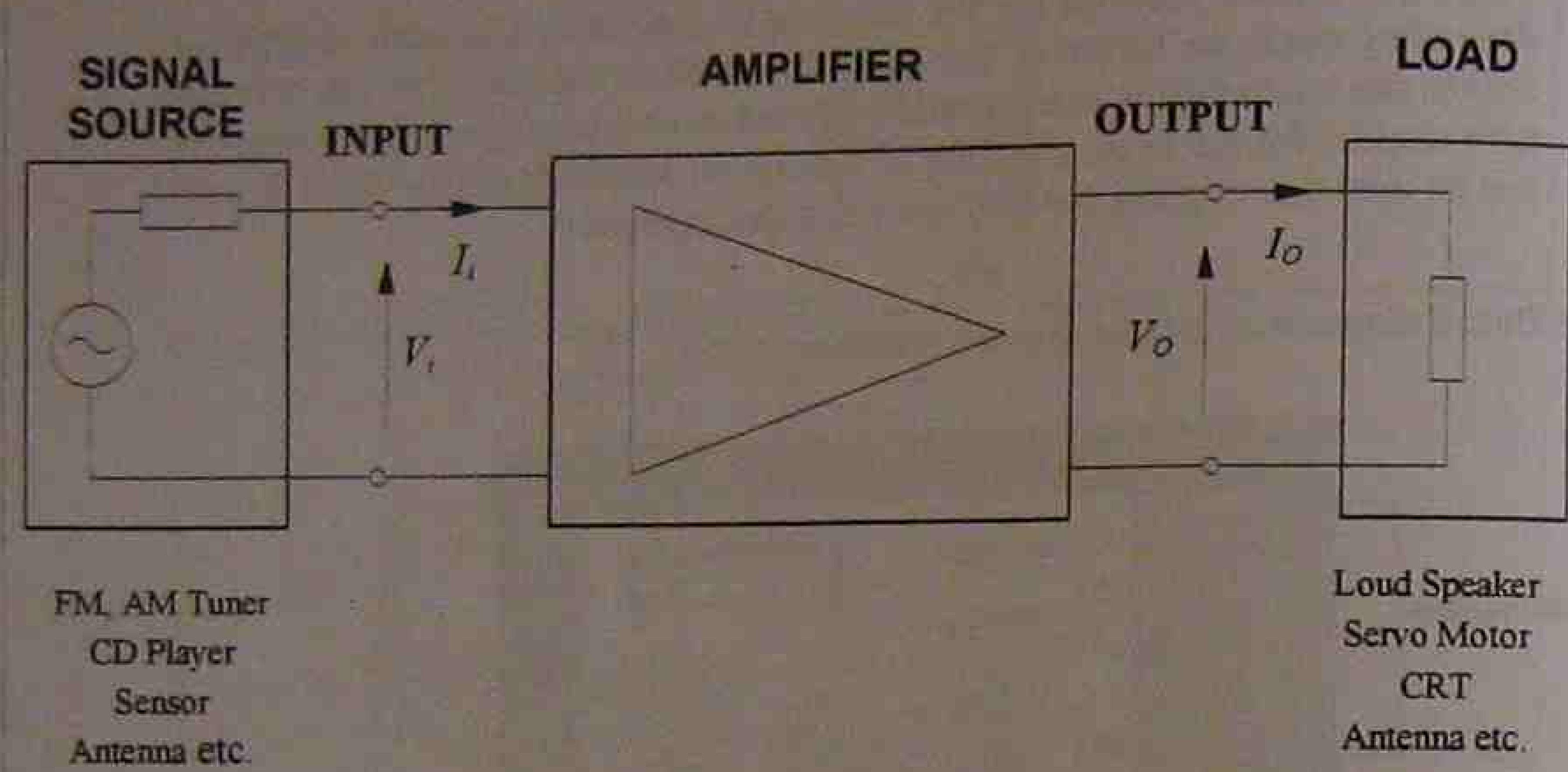


Figure 1.9 shows an example of an amplifier connected between a signal source and a load. In this diagram, V_i, V_O, I_i and I_O represent:

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

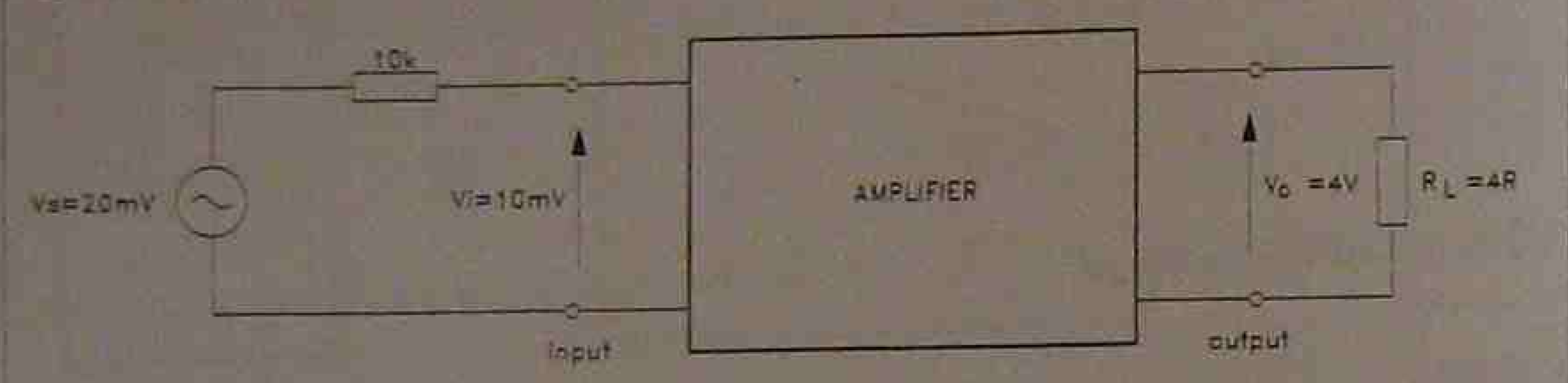
(b) $I_i = 0.15\mu\text{A}$, $I_O = 0.3\text{mA}$

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.

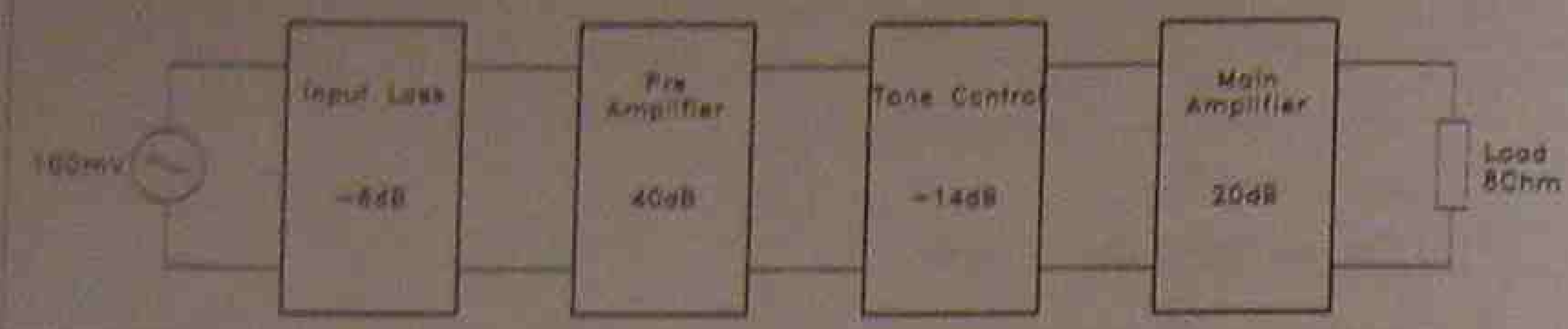
(a) Voltage gain $A_V = 50$ (dB)

(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

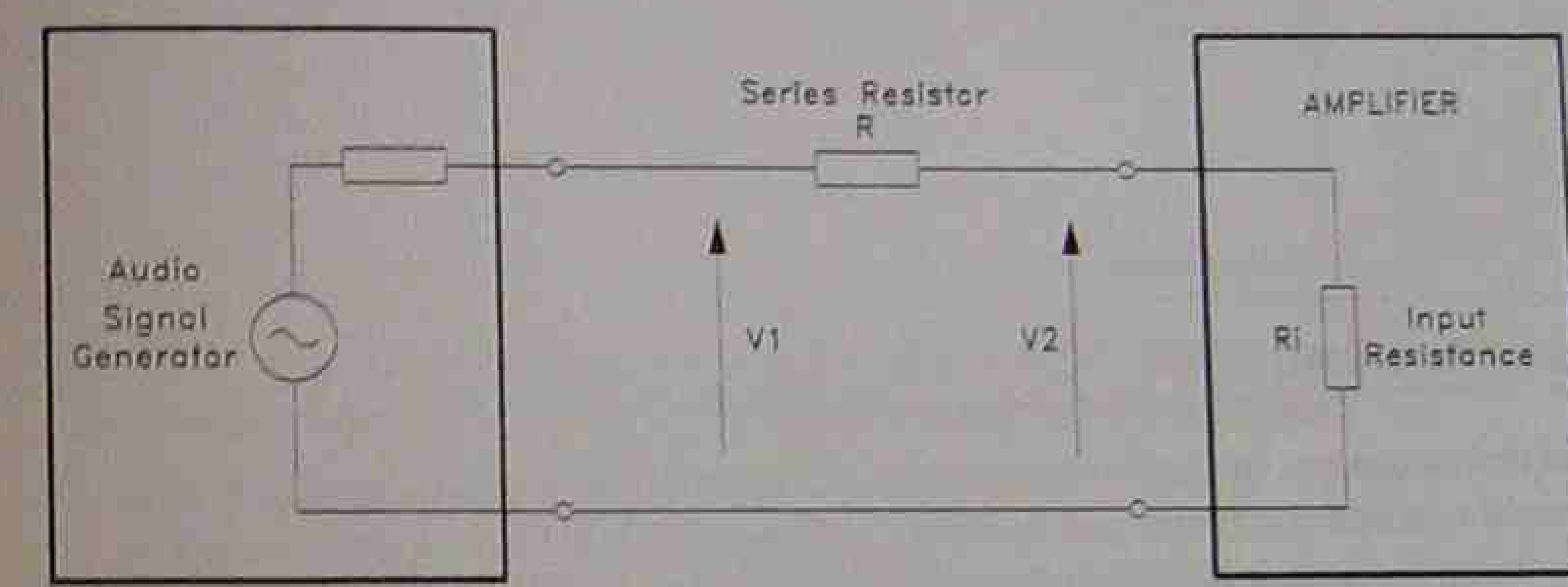
Input Resistance

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:

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

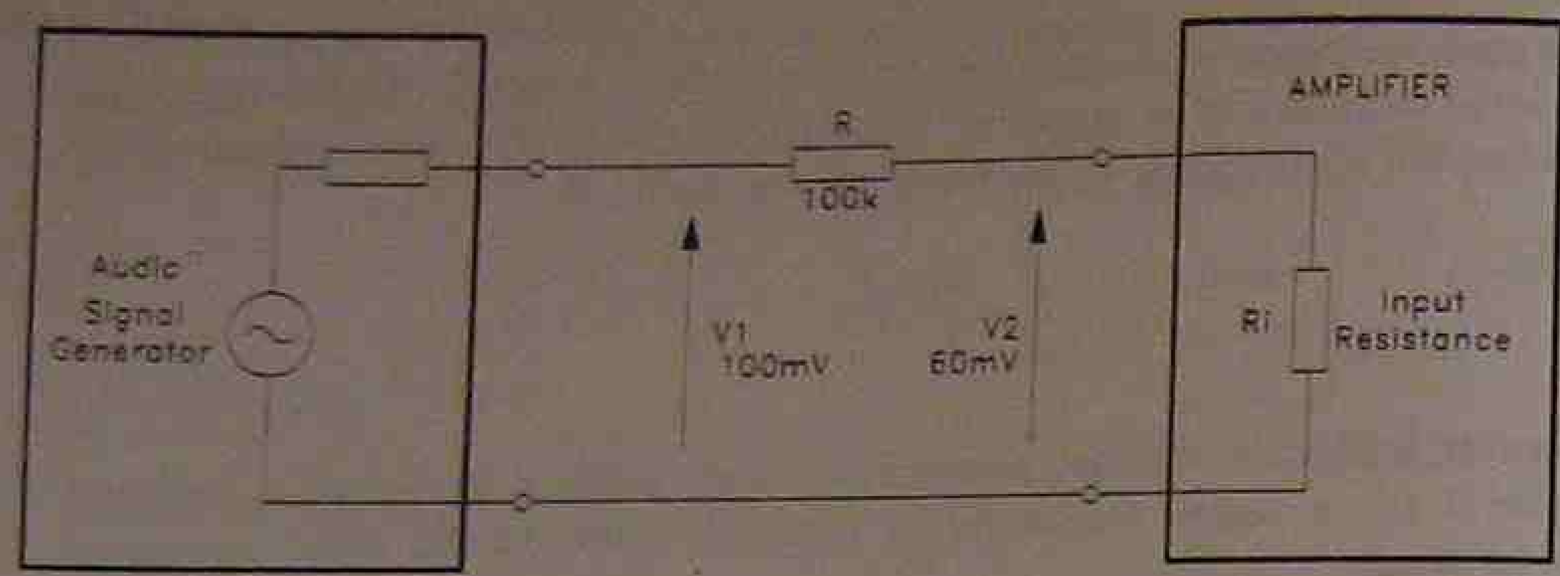
Also, V_2 is the input voltage of the amplifier V_i. Therefore, the input resistance of the amplifier can be determined:

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

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

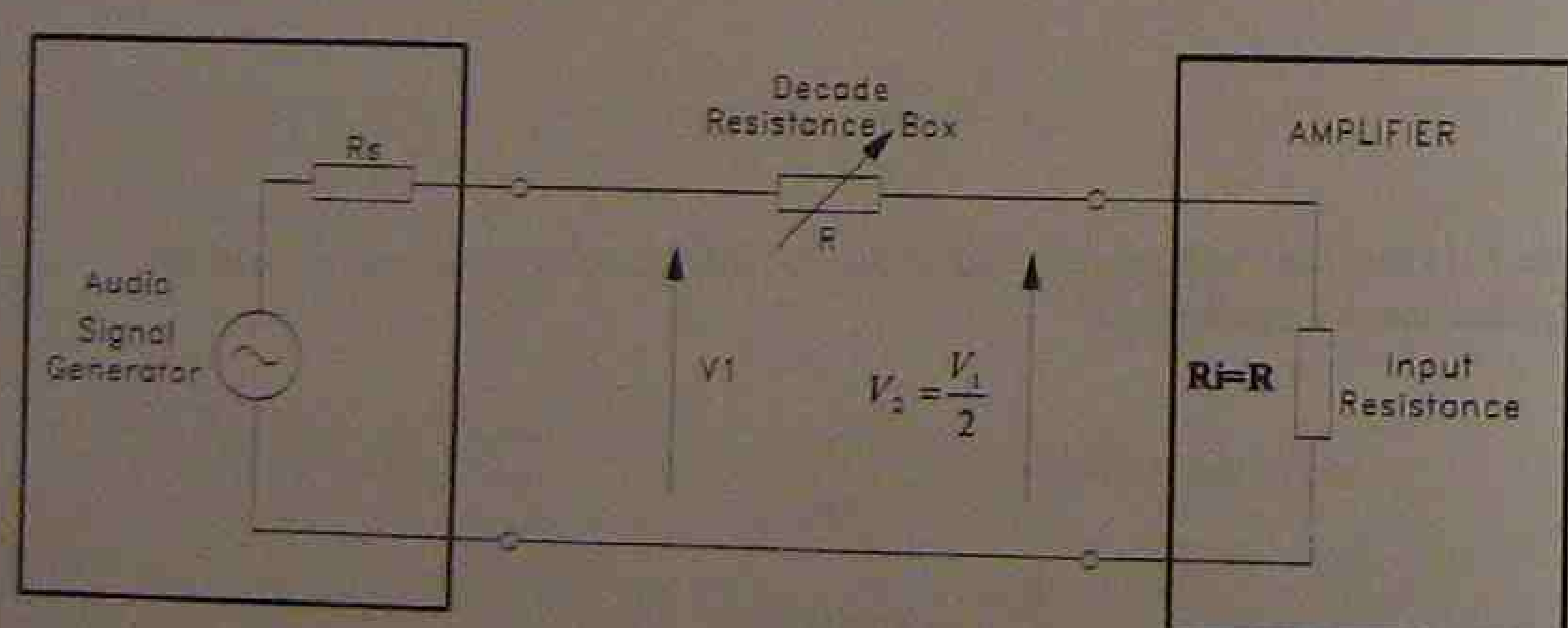
Drill Question 10

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.

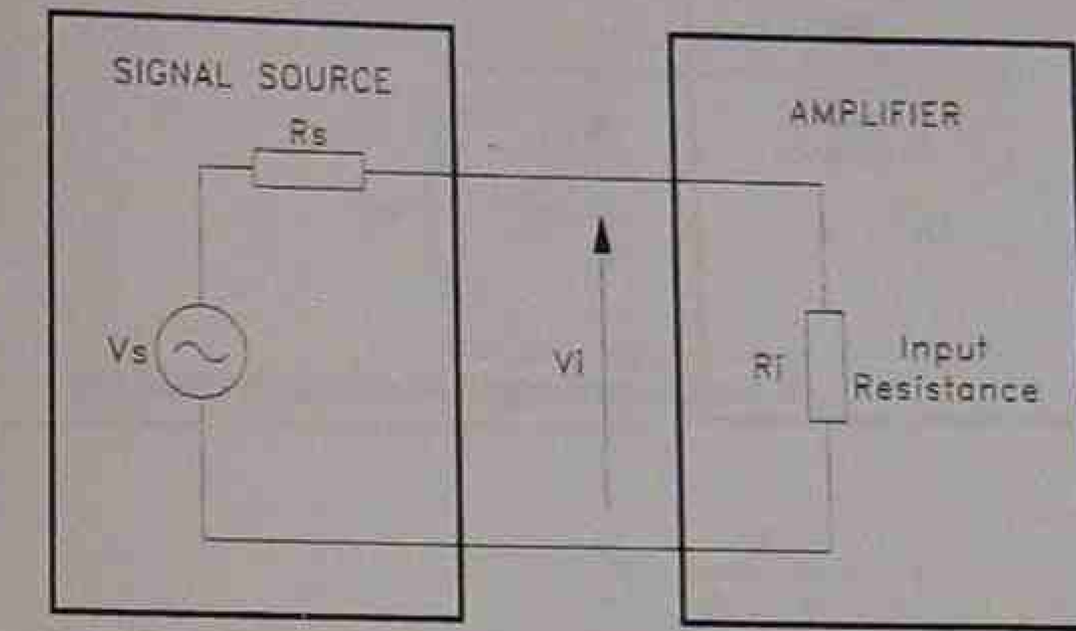


Loss due to the Input Resistance

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.

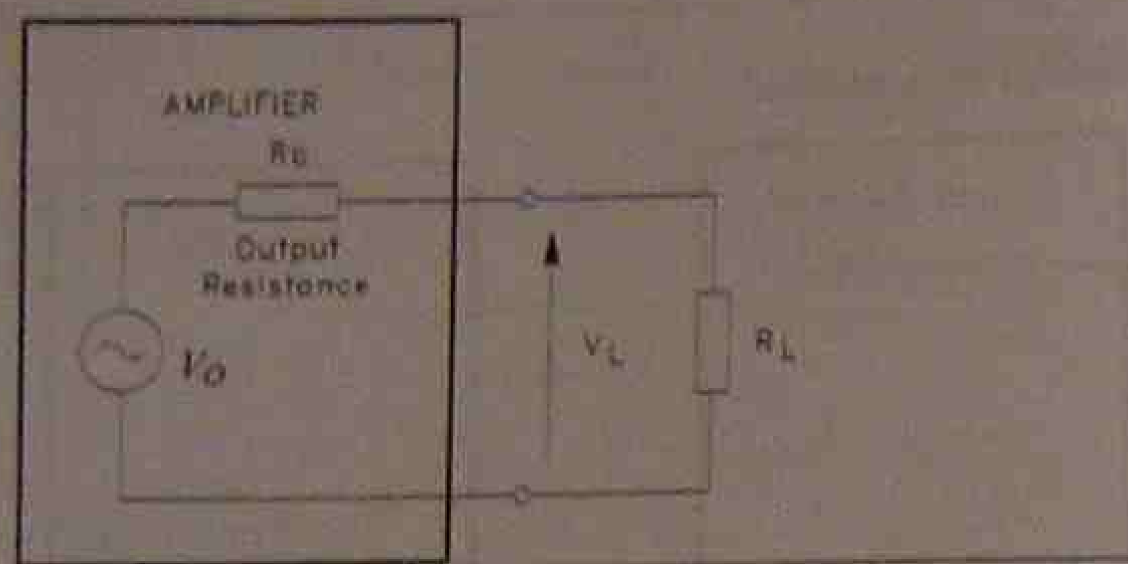
(b) Input loss in dB.

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.

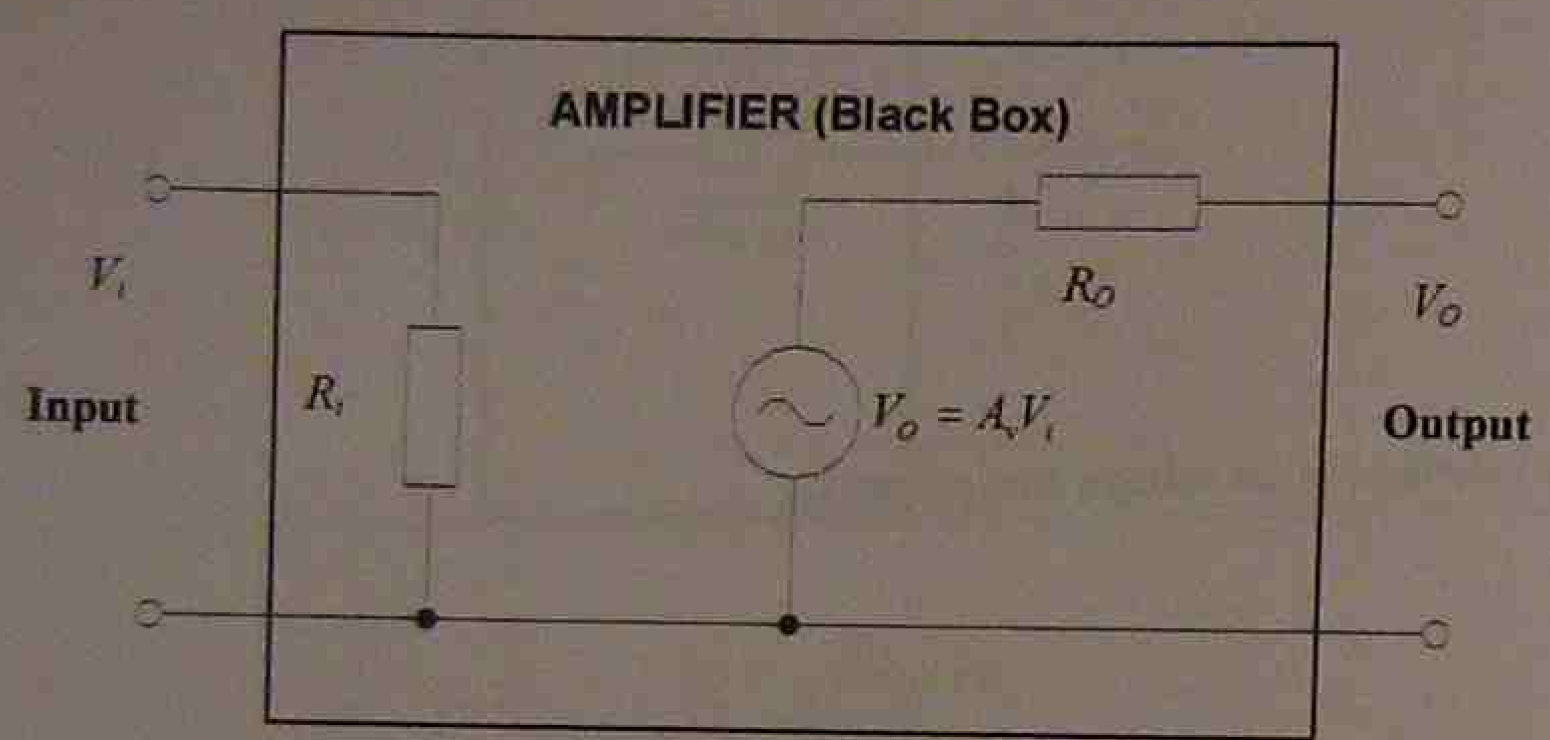
(d) Determine the value of output resistance of the amplifier.

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

1.5 Amplifier Equivalent Model

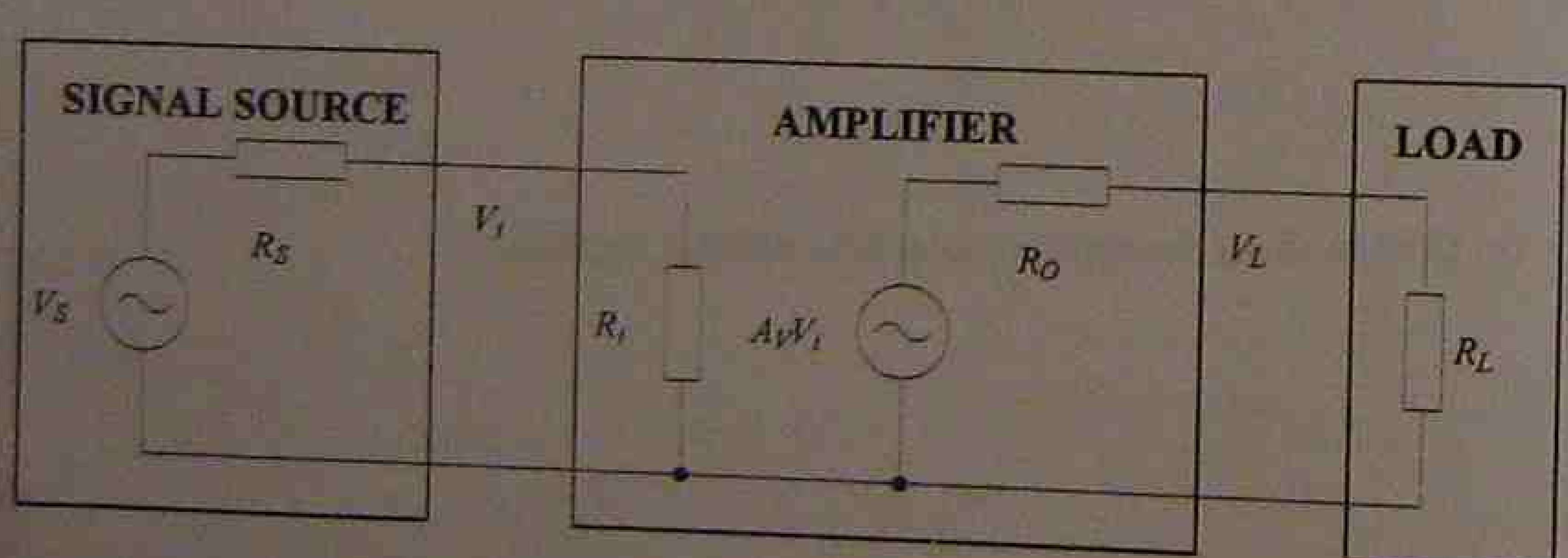
If we assume that an amplifier can be regarded as a black box, then it can be expressed with its input resistance (R_i), output resistance (R_O) and output voltage without load (V_O) regardless what electronic devices are employed to construct the amplifier.

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.

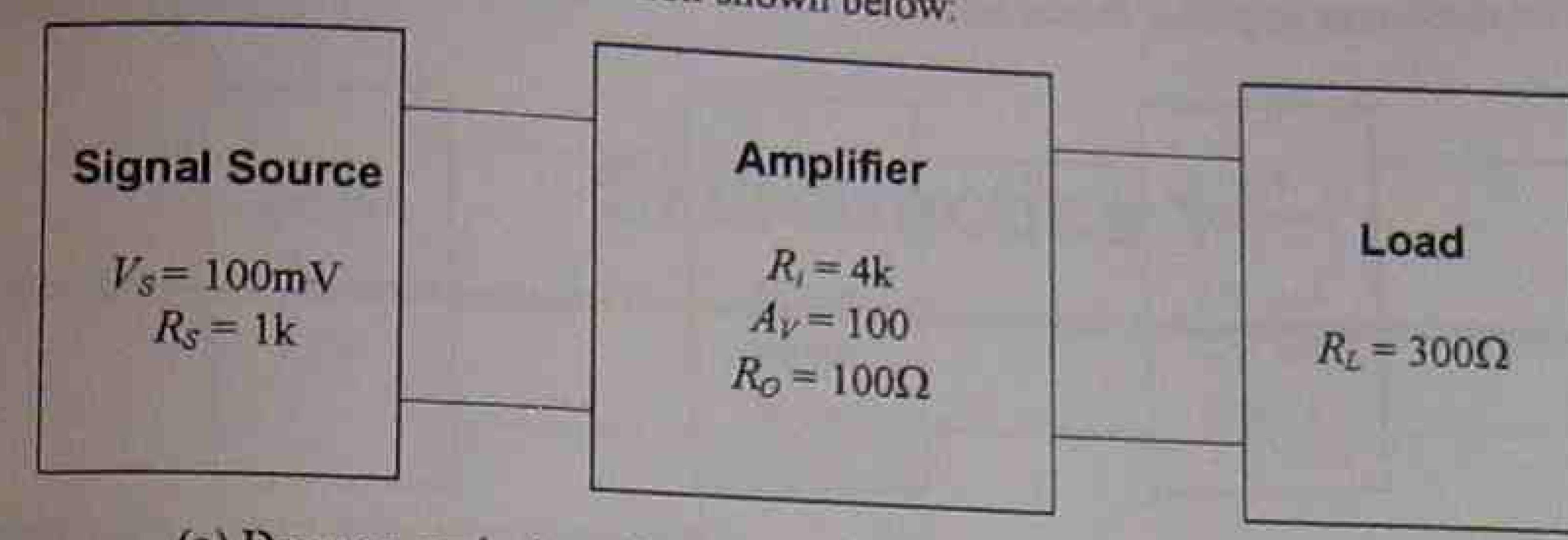
Figure 1.17 An amplifier between a signal source and a load.



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

Drill Question 13

For the amplifier connection shown below:



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

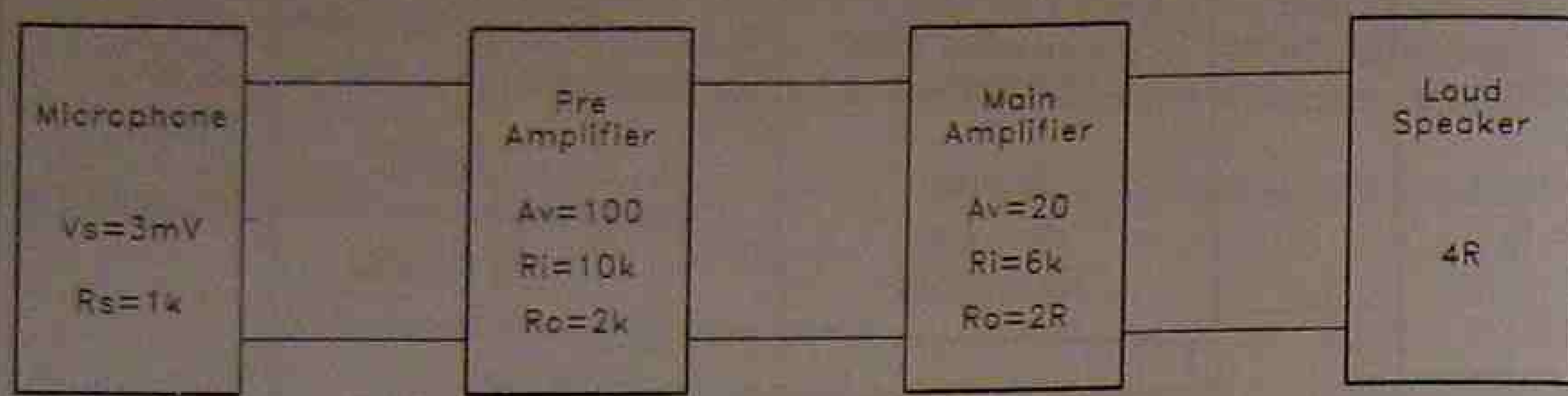
(b) Determine the input voltage of the amplifier.

(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.

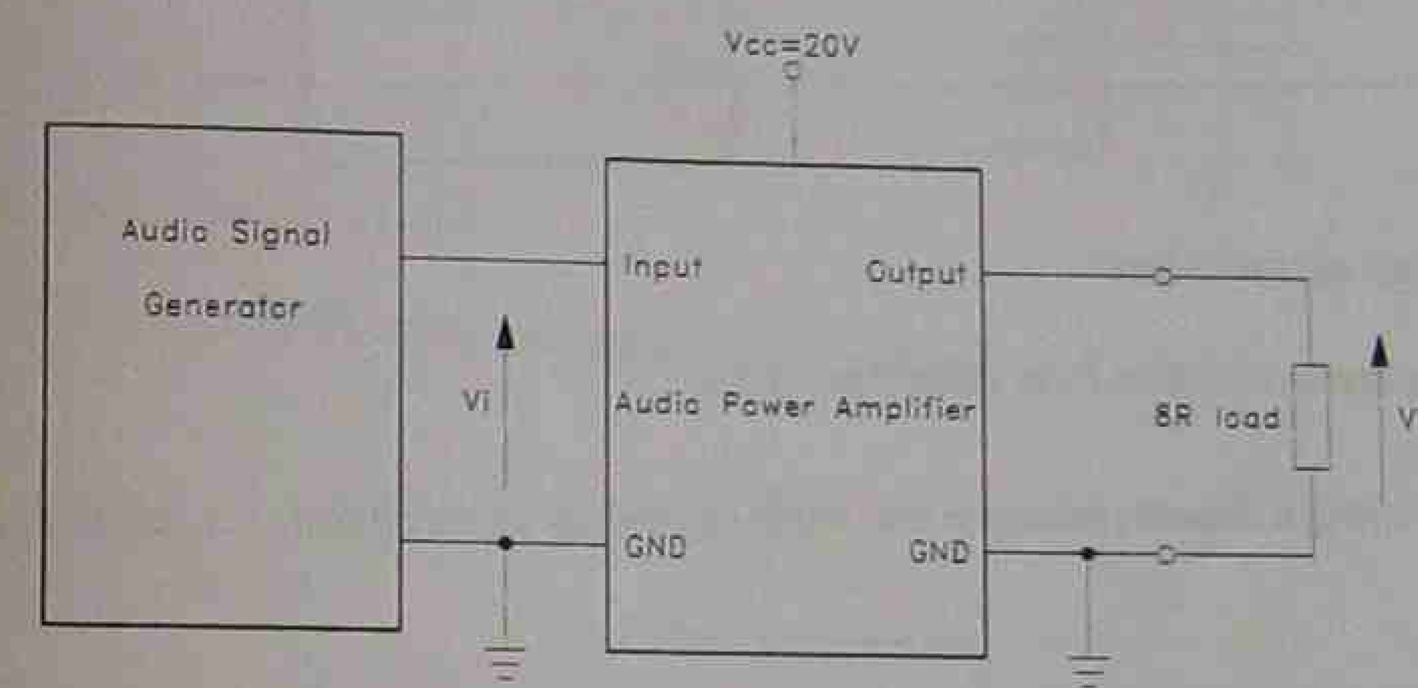
(e) Determine the output power.

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.

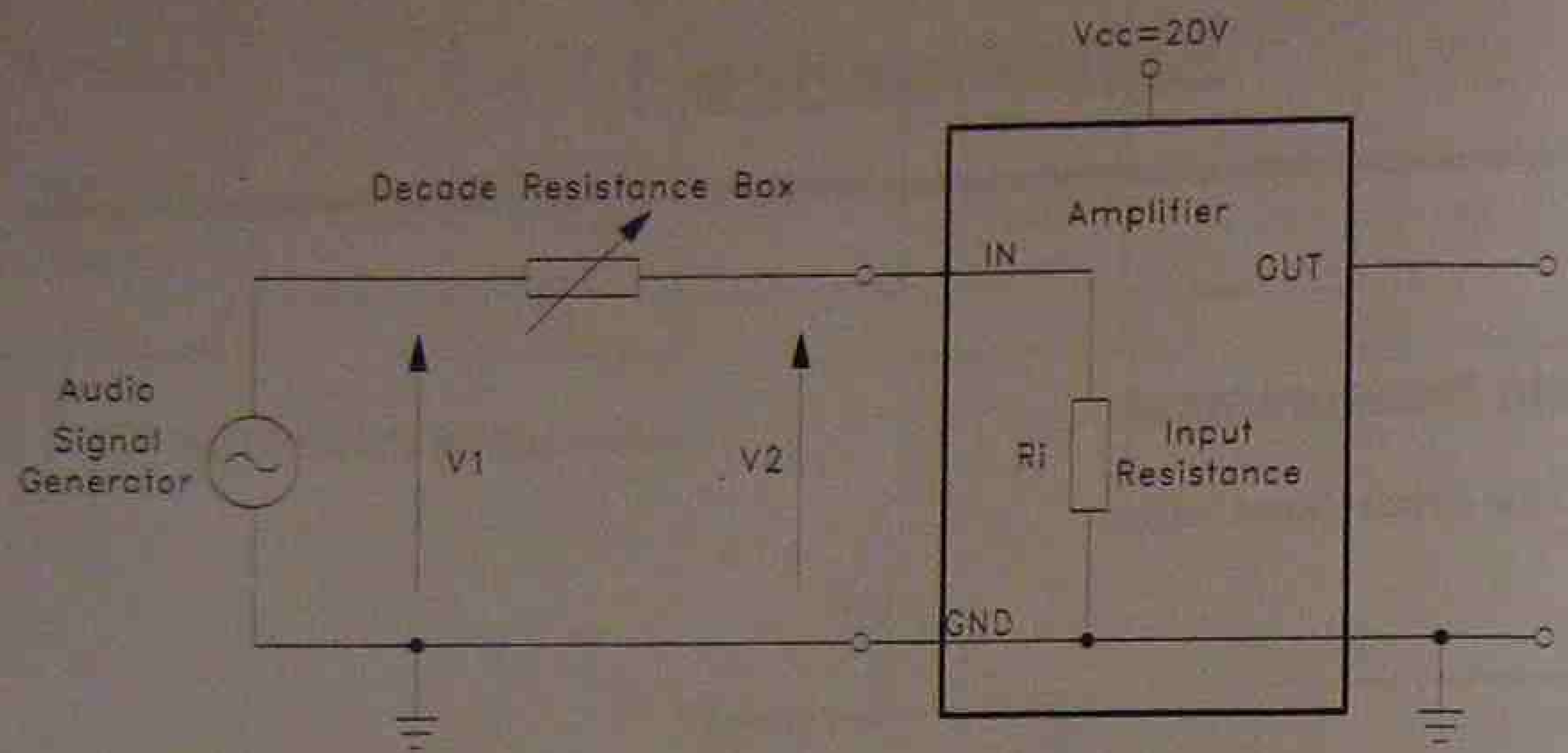
Vo =

(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 =

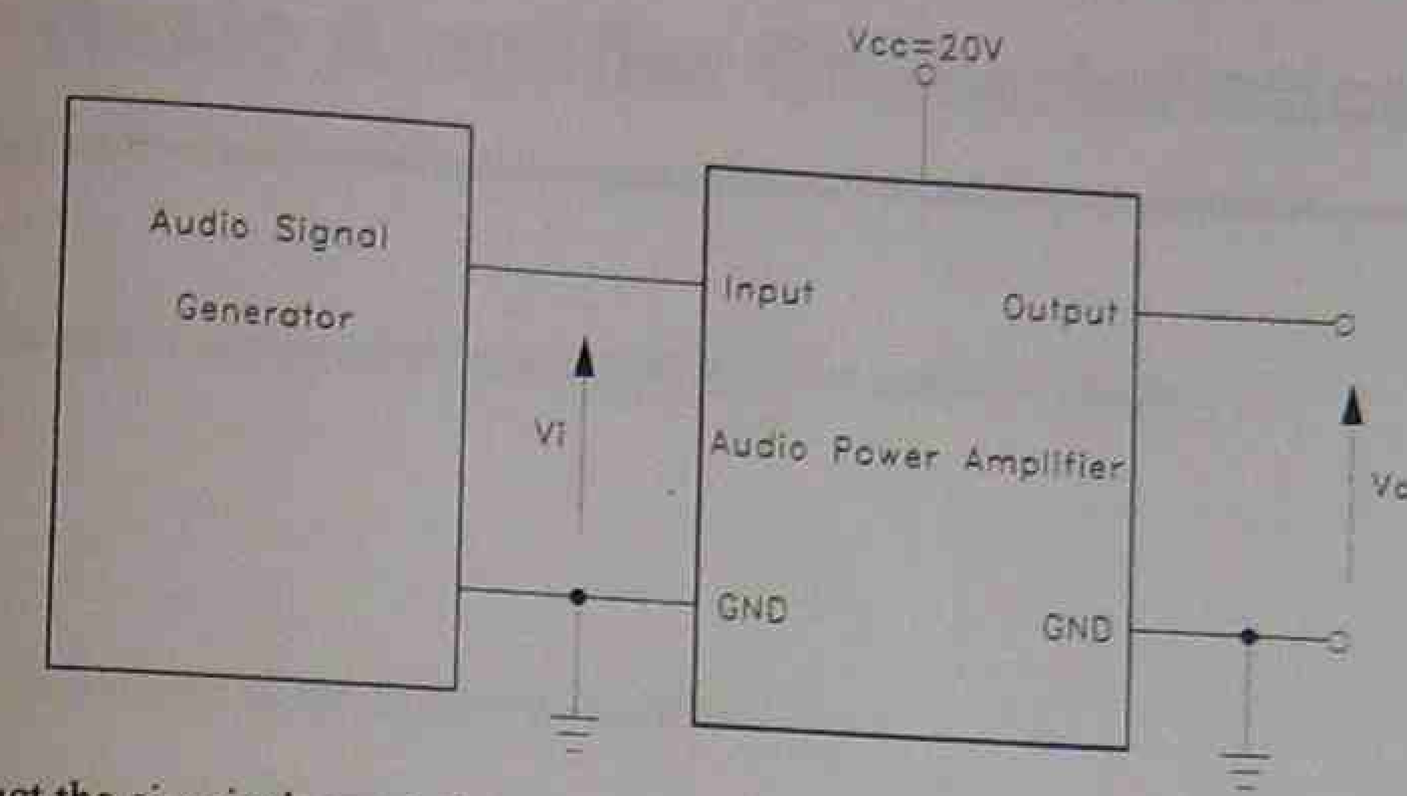
(e) Repeat (a) ~ (d) for:

100Hz input

10kHz input

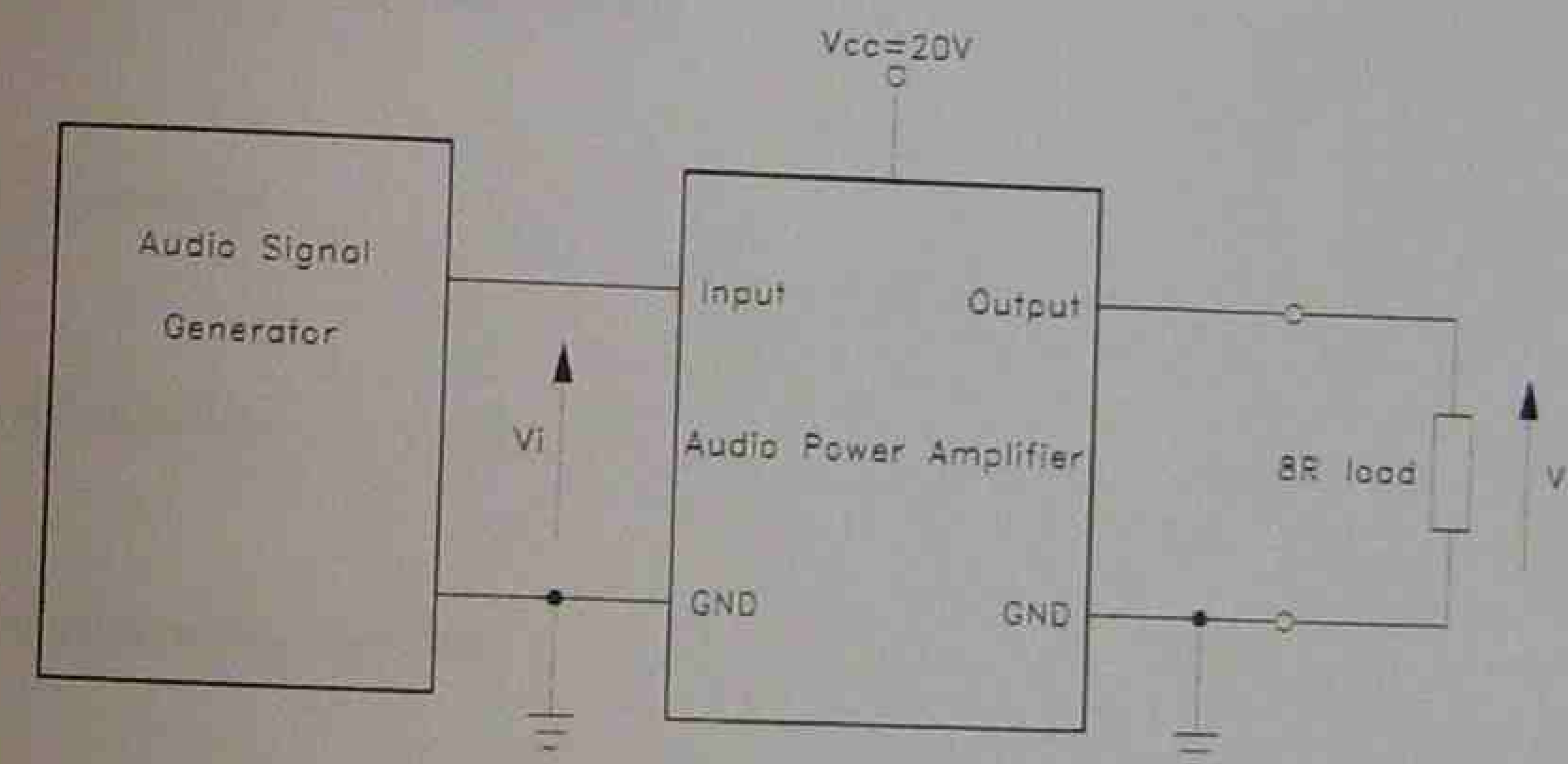
100kHz input

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.

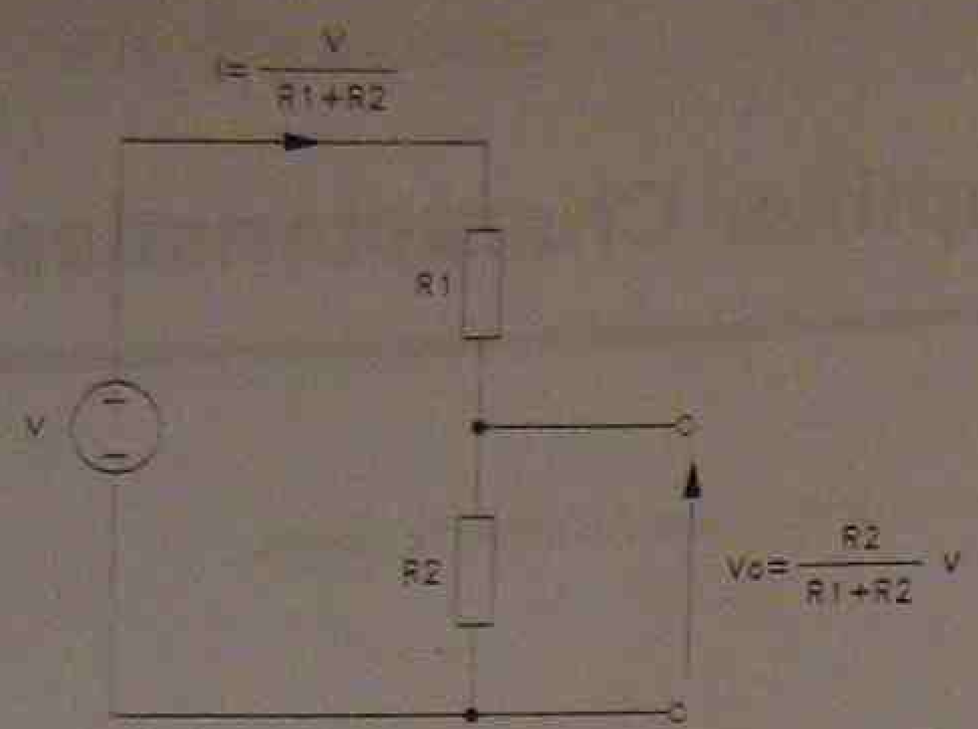
(c) DC amplifier.

(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:



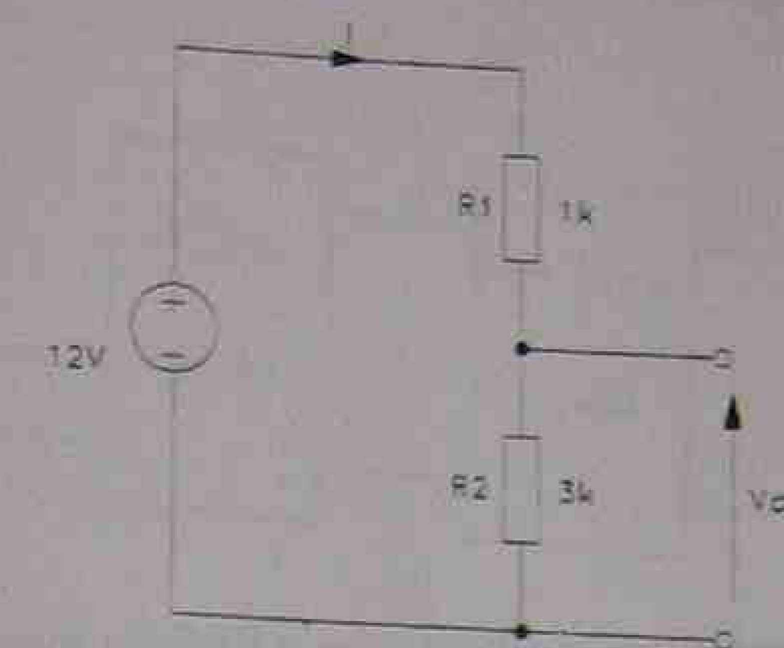
(a) V=10V, R1=600Ω, R2=1.4k

(b) V=12V, R1=4Ω, R2=8Ω

(c) V=15V, R1=0Ω, R2=2k

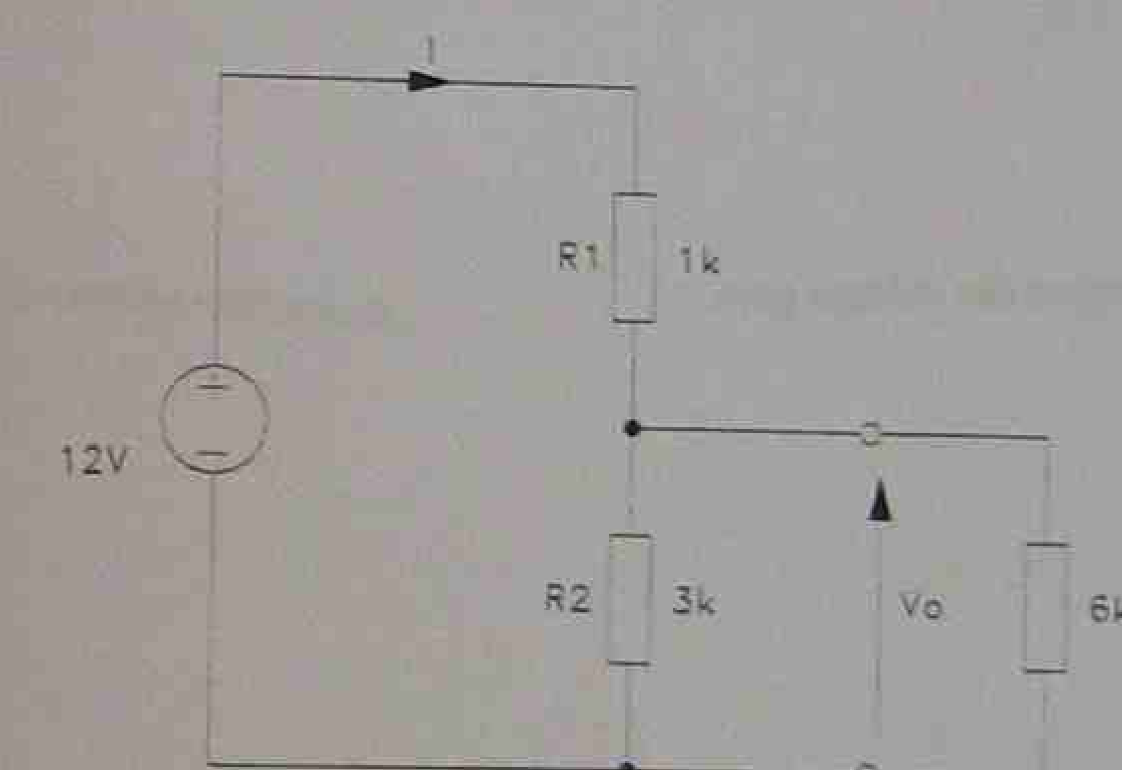
(d) V=18V, R1=2k, R2=0Ω

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.

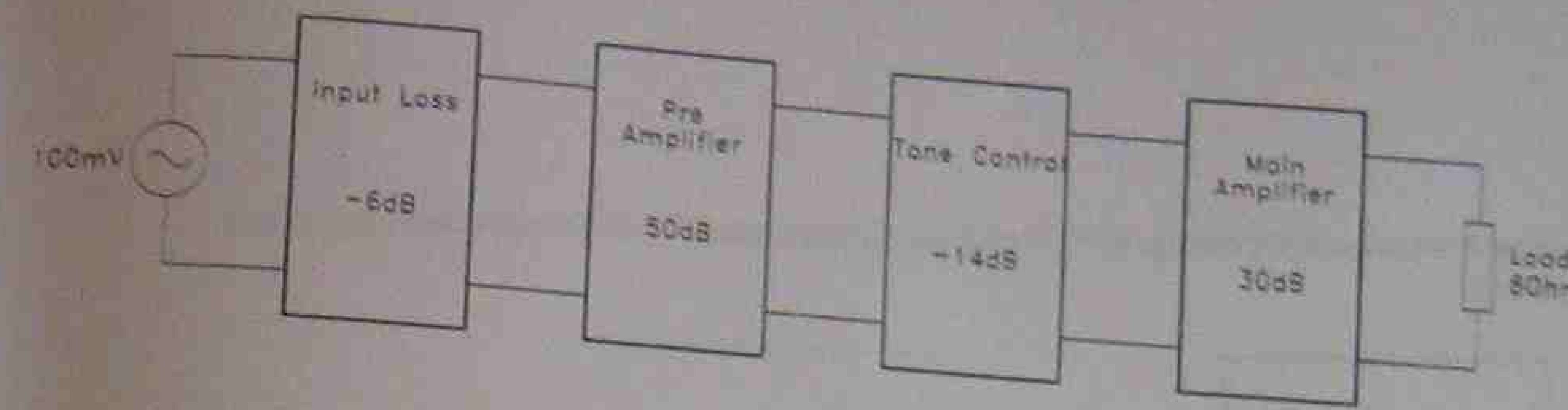


Q6. What is meant by the term 'voltage gain' in an amplifier?

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.

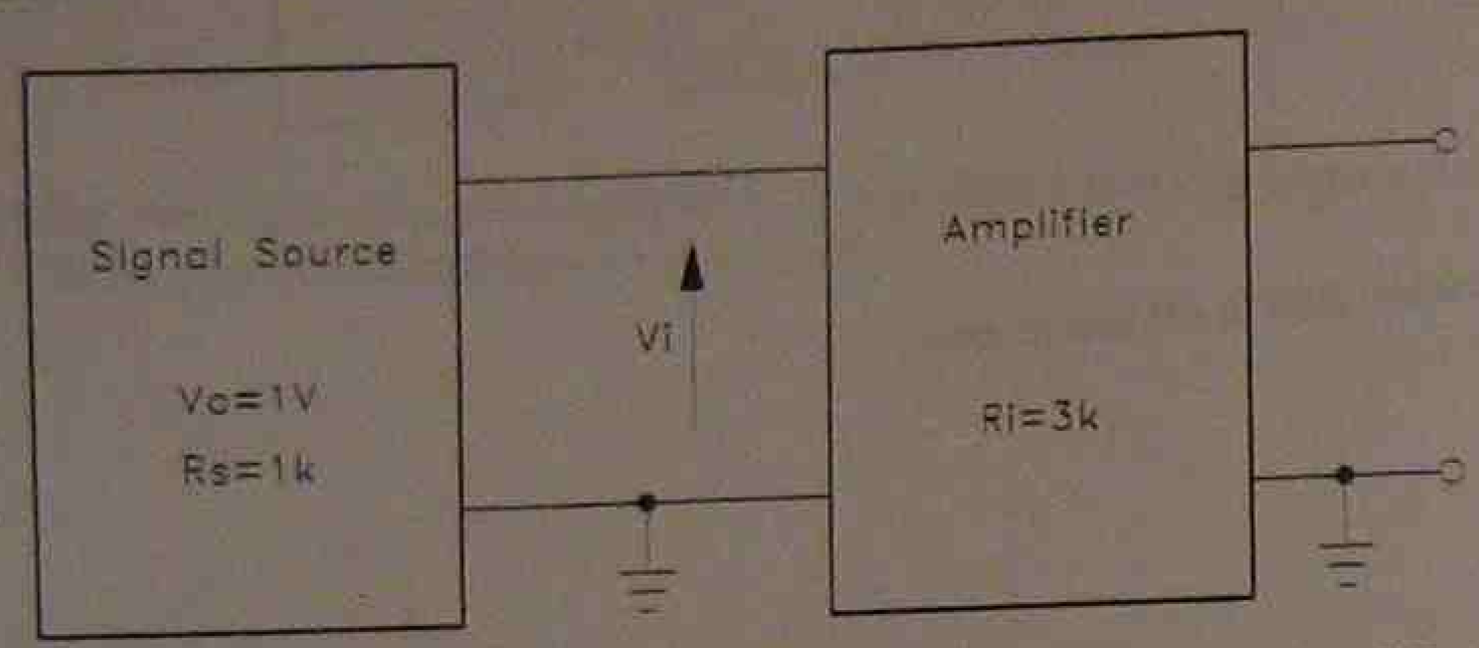


(a) Determine the total voltage gain in dB and in ratio.

(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.

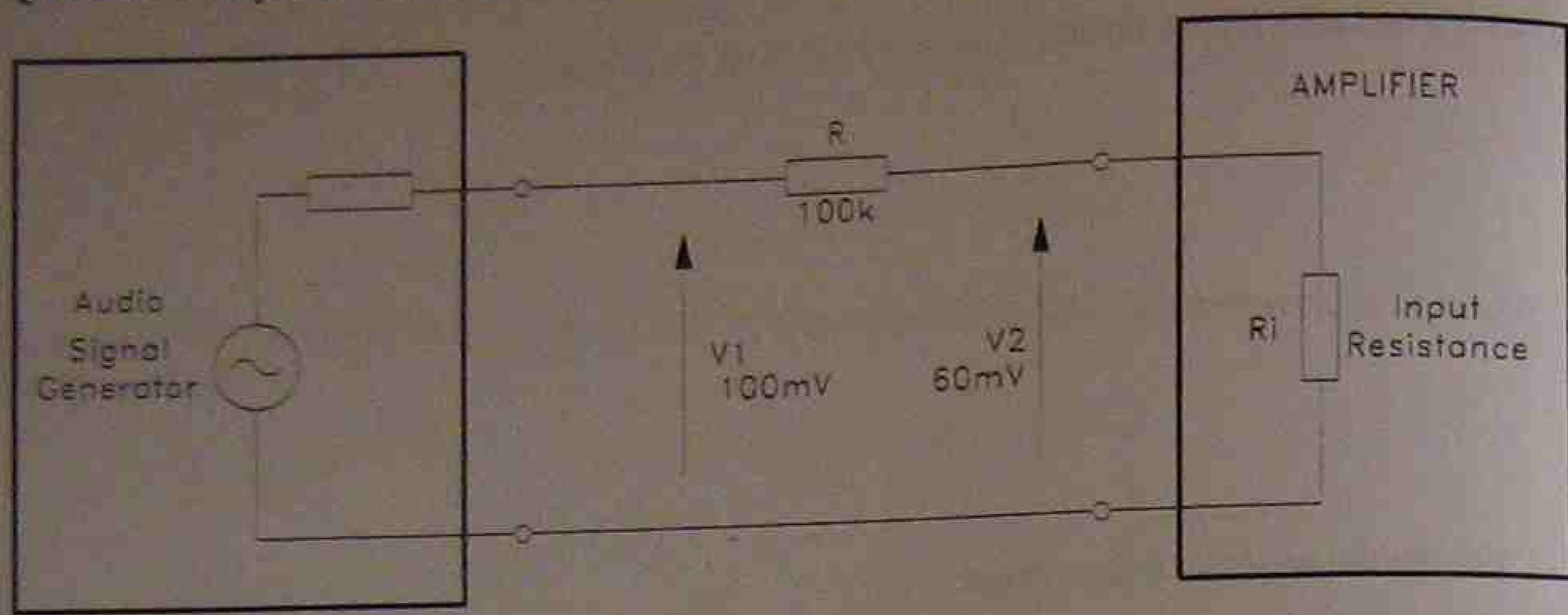
(i) Draw a equivalent input circuit for the source feeding the amplifier.

(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.



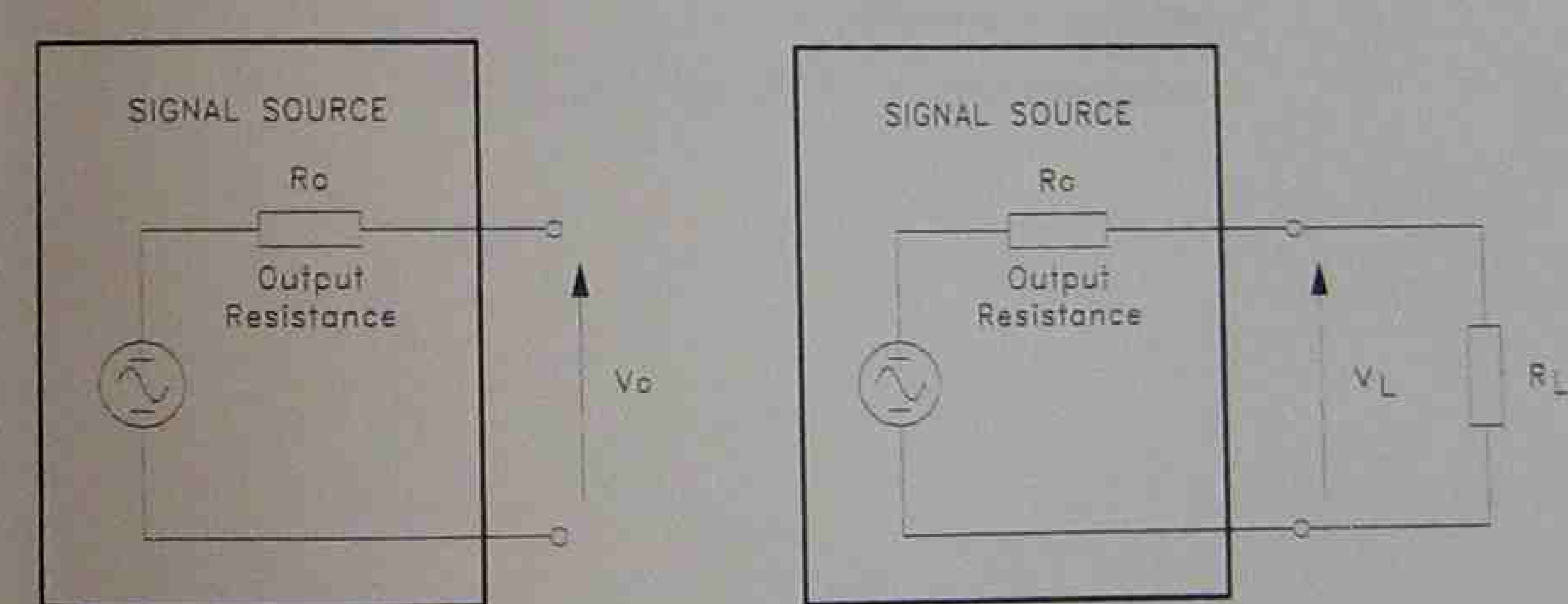
Q13. For the circuit diagram in Q12, a decade resistance box is employed instead of R. State the process of input resistance measurement.

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.

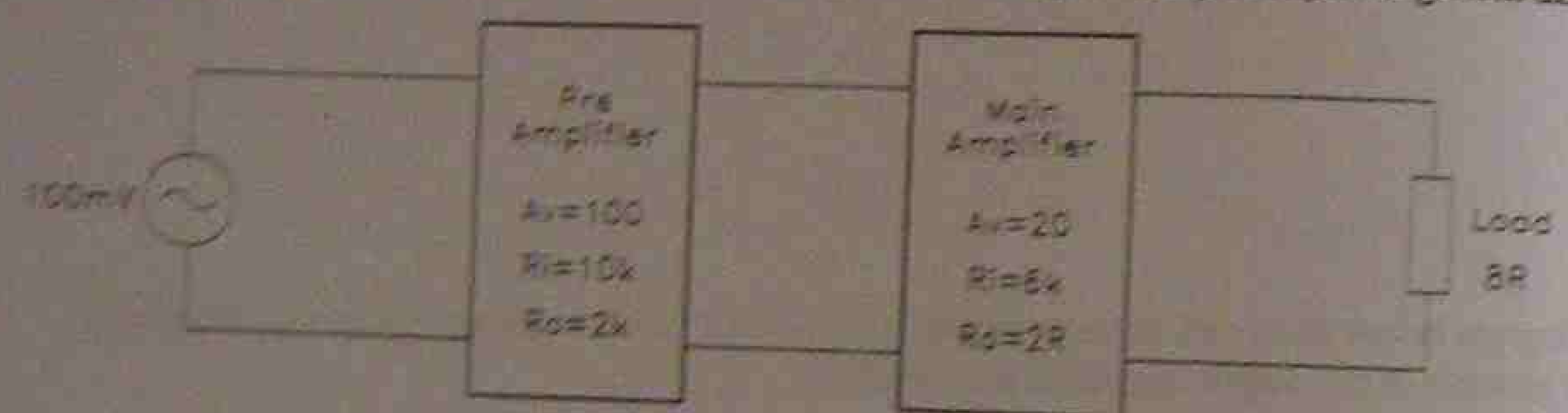
Q15. The diagram below shows an output resistance measurement method for any signal source. Briefly describe the process of the output resistance measurement.



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:

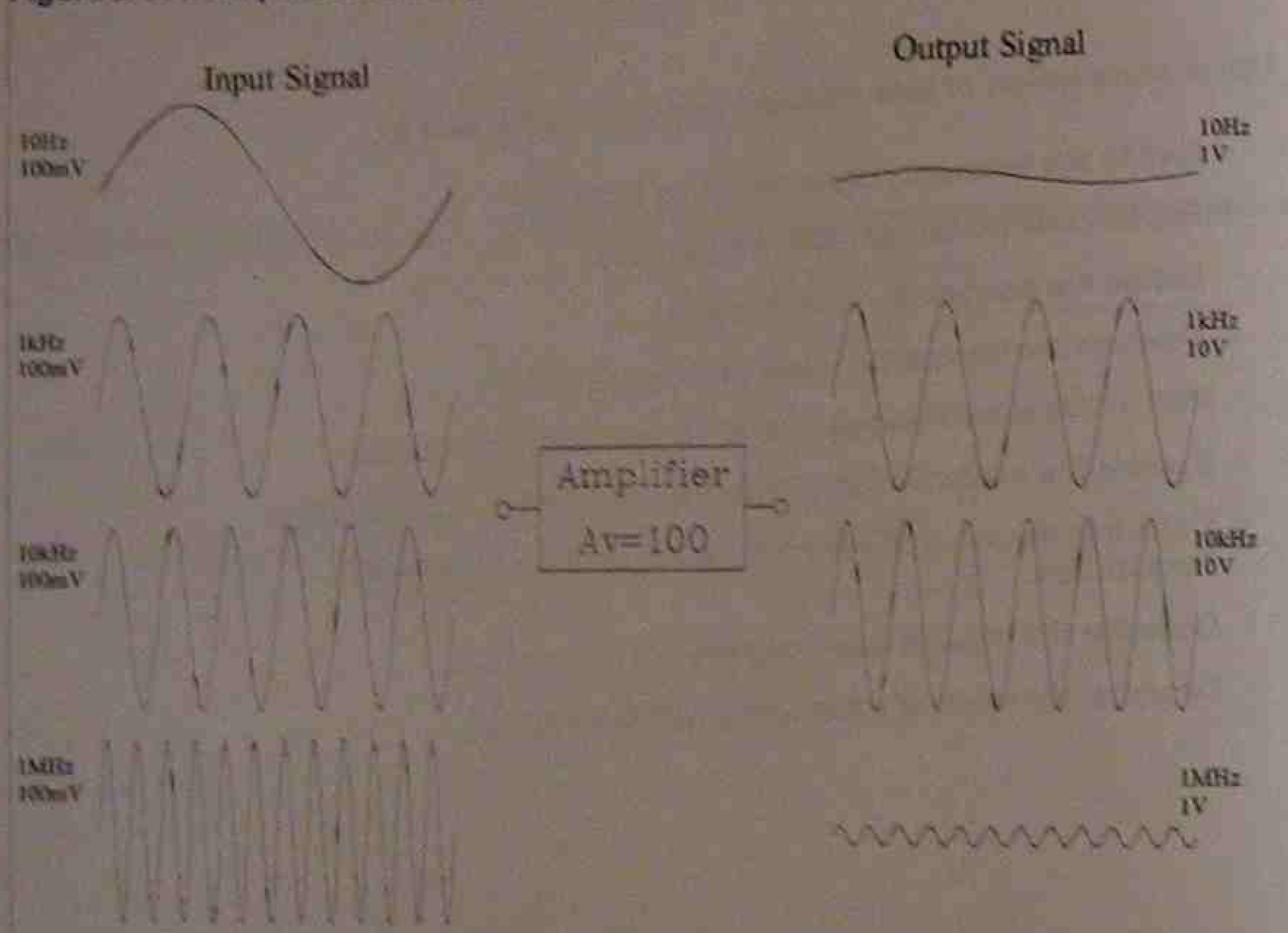
- Define the frequency response of an amplifier.
- Explain half-power (or -3dB) frequencies.
- Define the bandwidth of an amplifier.
- Measure the frequency response of an amplifier.
- Plot a frequency response curve on semi-log graph paper.
- Explain the major factors of frequency limitation of an amplifier.
- 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

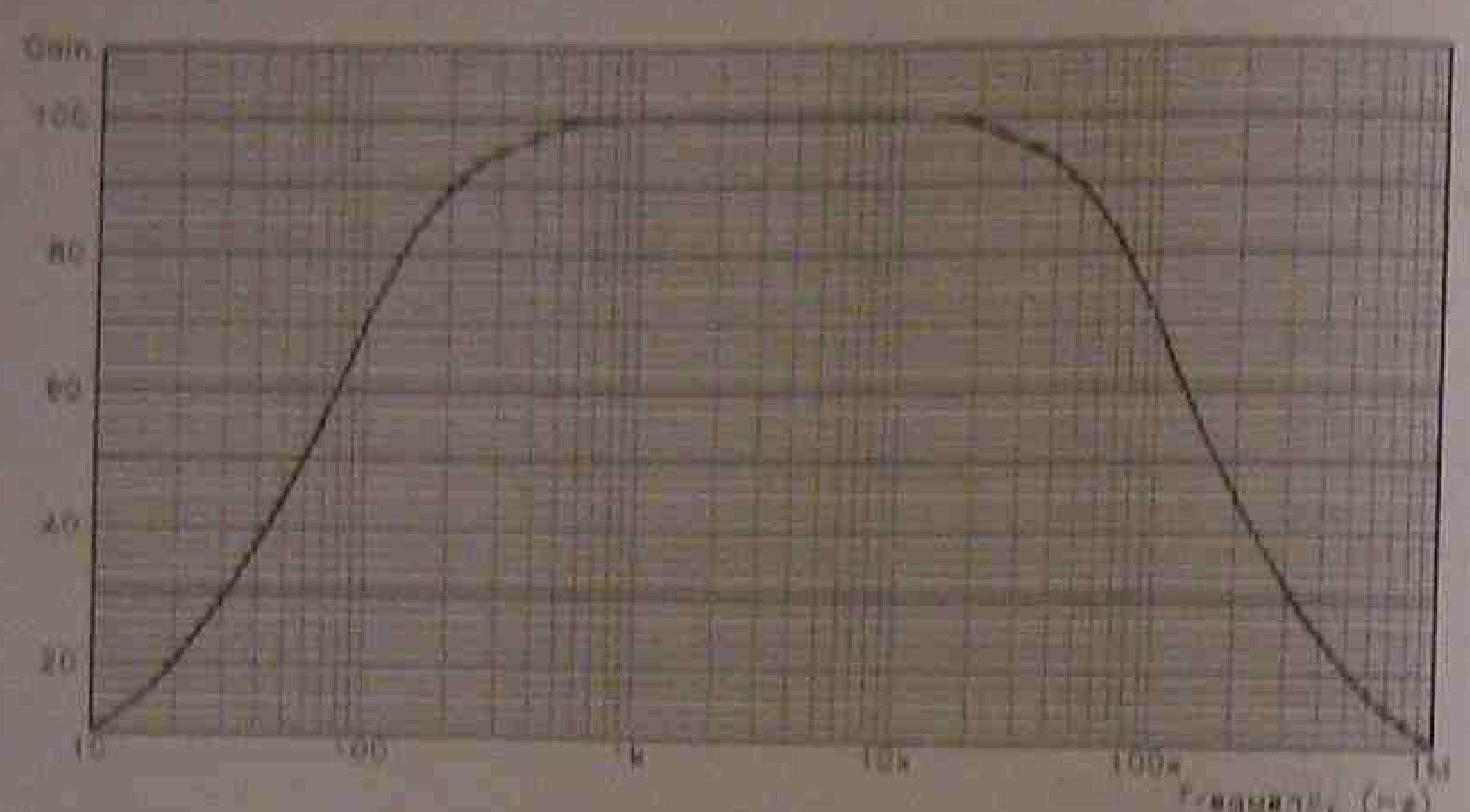
Frequency (Hz)	V_i (mV)	V_o (V)	Voltage Gain	Voltage Gain (dB)	10Ω load power(W)	power(dB) 1mW=0dB
10	100	1				
20	100	1.96				
40	100	3.71				
100	100	7.07				
200	100	8.94				
400	100	9.28				
1k	100	9.95				
2k	100	10				
4k	100	10				
10k	100	9.95				
20k	100	9.81				
40k	100	9.28				
100k	100	7.07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

Drill Question 1

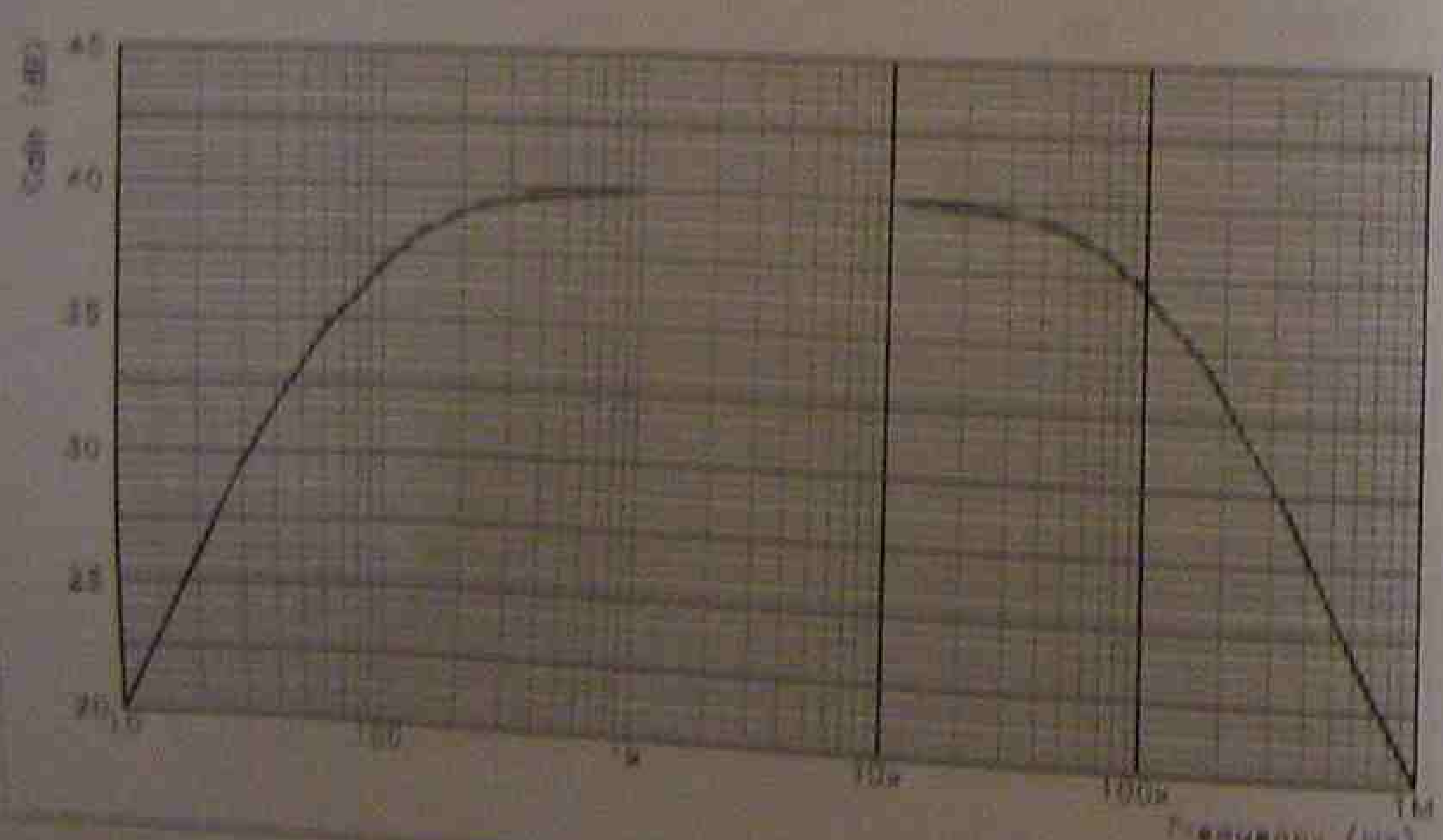
For Table 2.1, calculate voltage gain in direct ratio and in dB for the all measurements. Compare your calculation with frequency response plot in Figure 2.2 . Also, calculate output power to the 10Ω load and show this output power in dB (1mW=0dB) on the graph in (b).

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**.

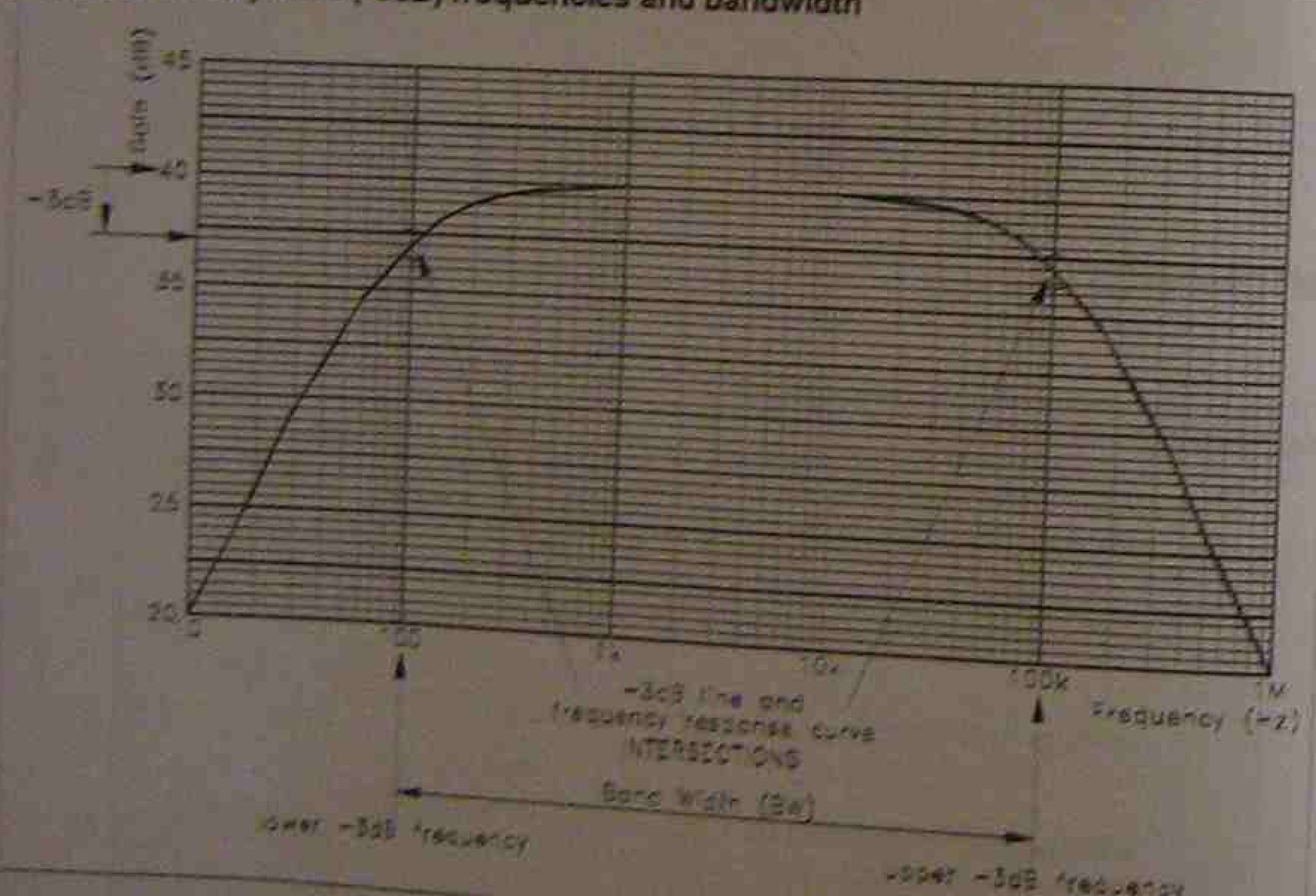
You also noticed that, regardless of voltage gain or power gain, there is a 3dB gain difference between mid-band frequencies and half-power frequencies. Because of this 3dB difference, half-power frequencies are sometimes called **-3dB frequencies**.

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

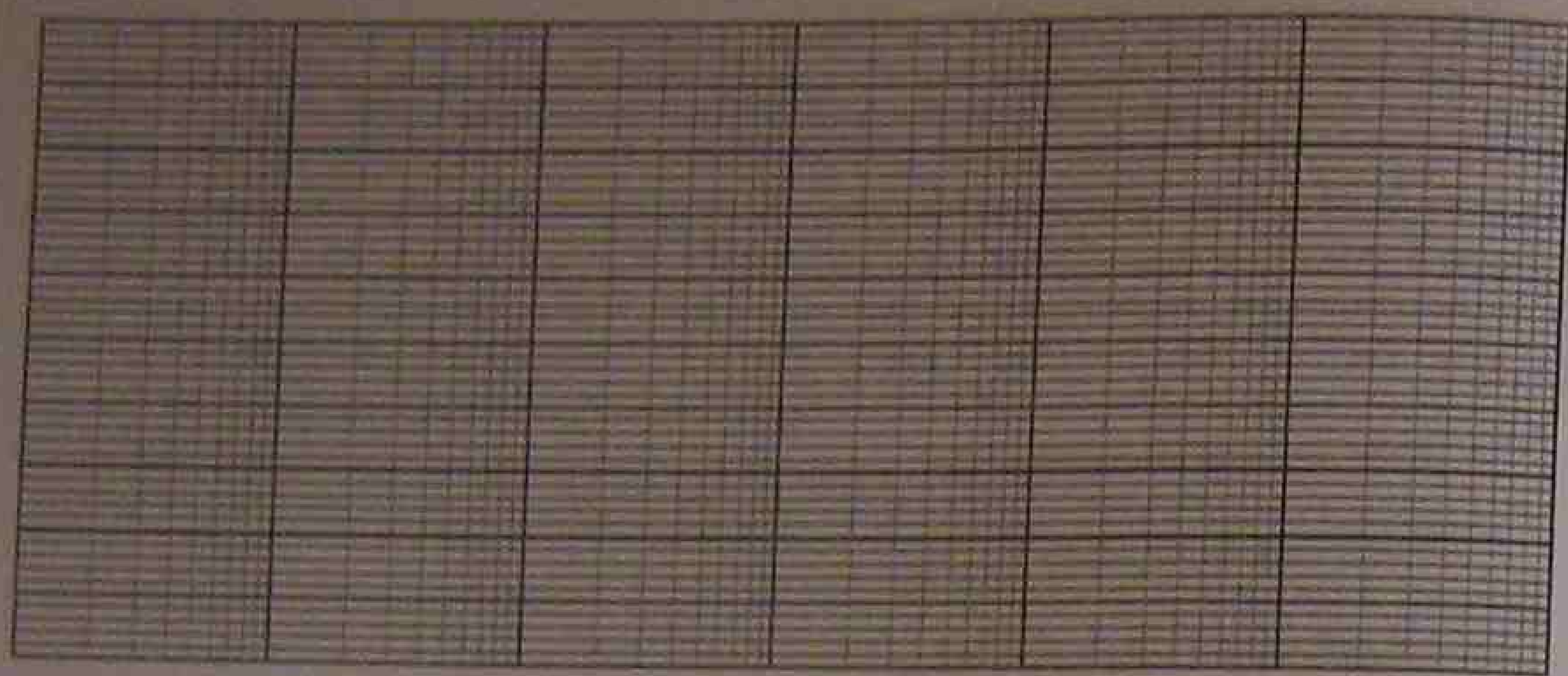
For a frequency response measurement result shown below:

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.



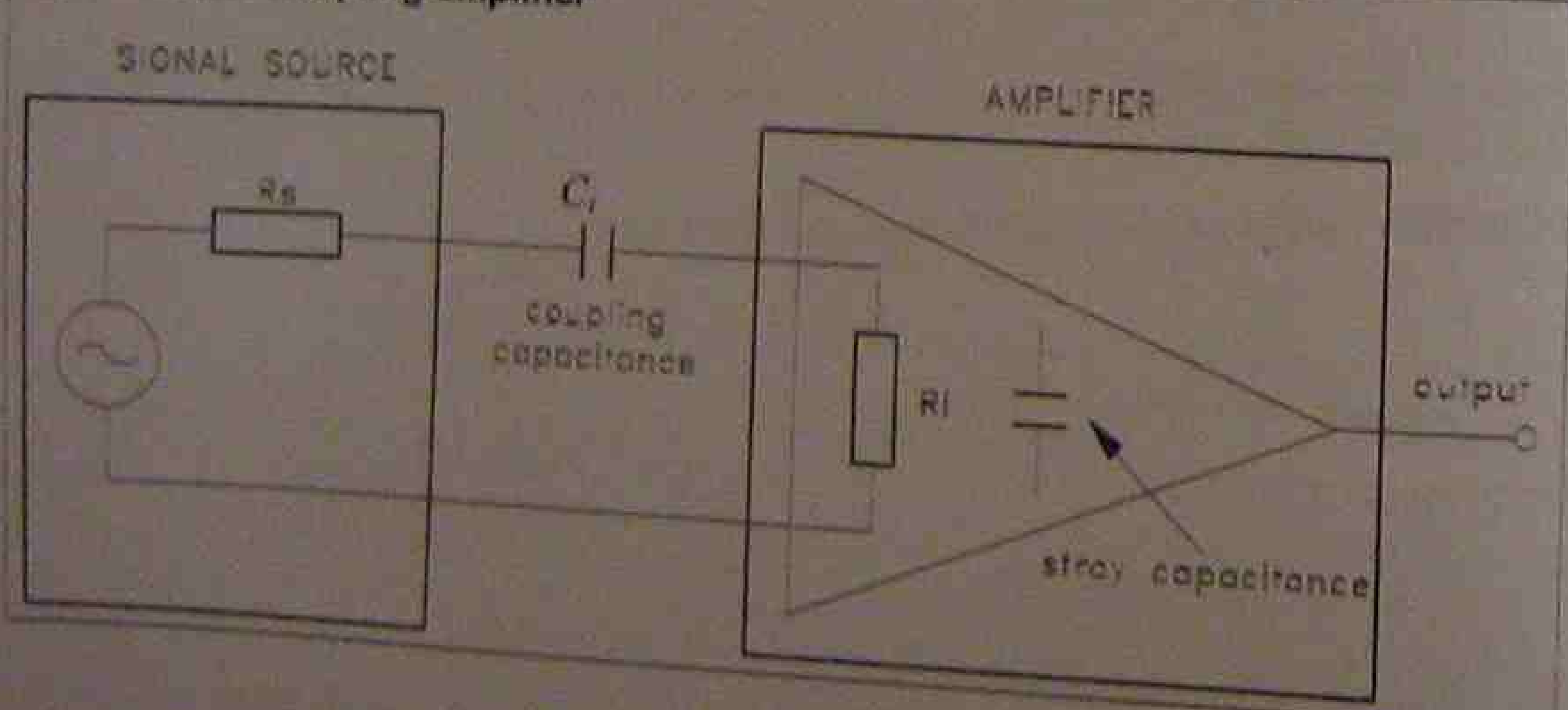
(C) Determine the half-power frequencies

(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

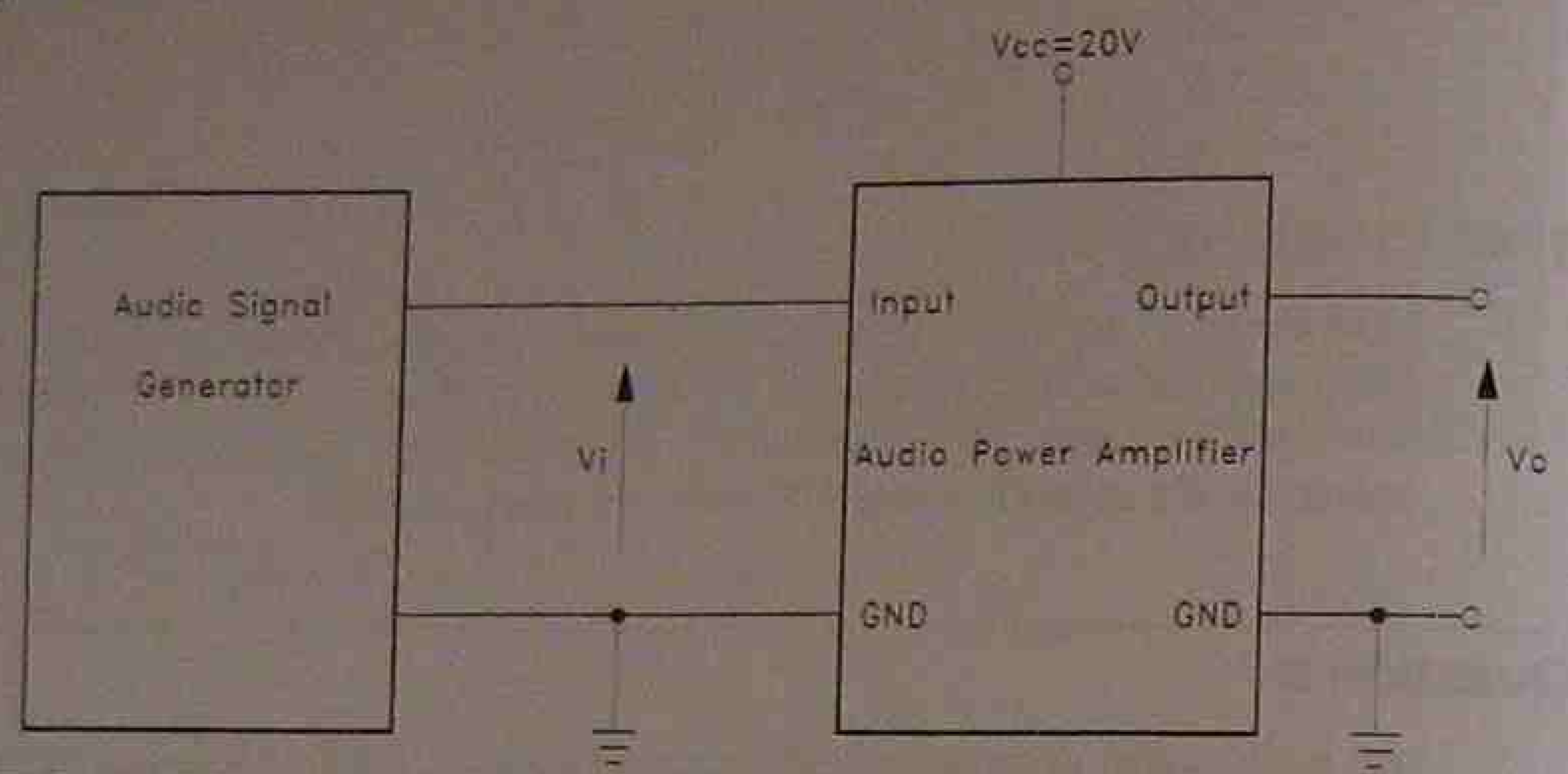
An audio signal generator whose source resistance is 600Ω is connected to an amplifier which has a 3kΩ input resistance. A 1μF coupling capacitor is employed between the signal source and the amplifier. Determine the lower half-power frequency

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 =

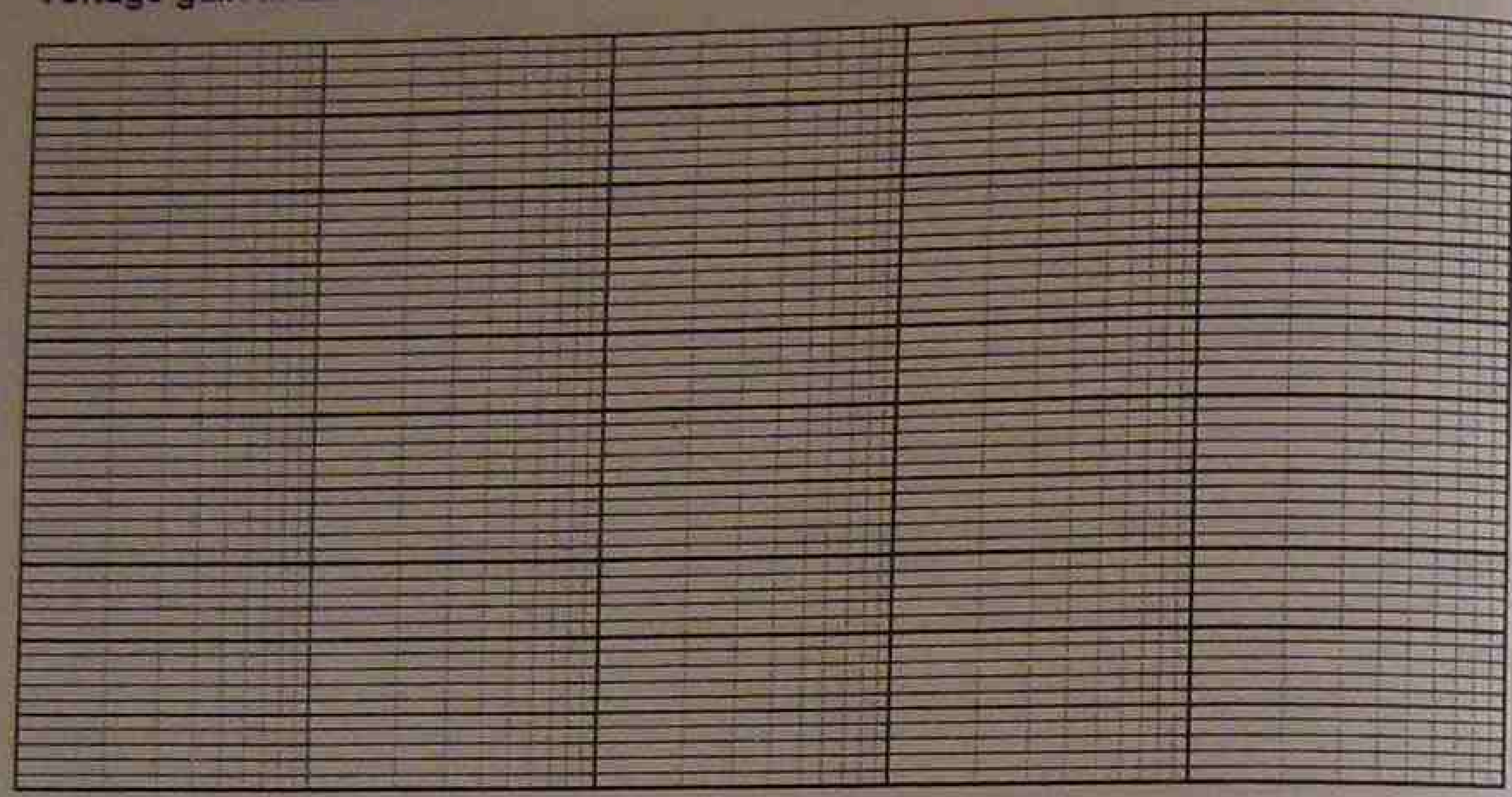
Upper half-power frequency =

Table 2.2

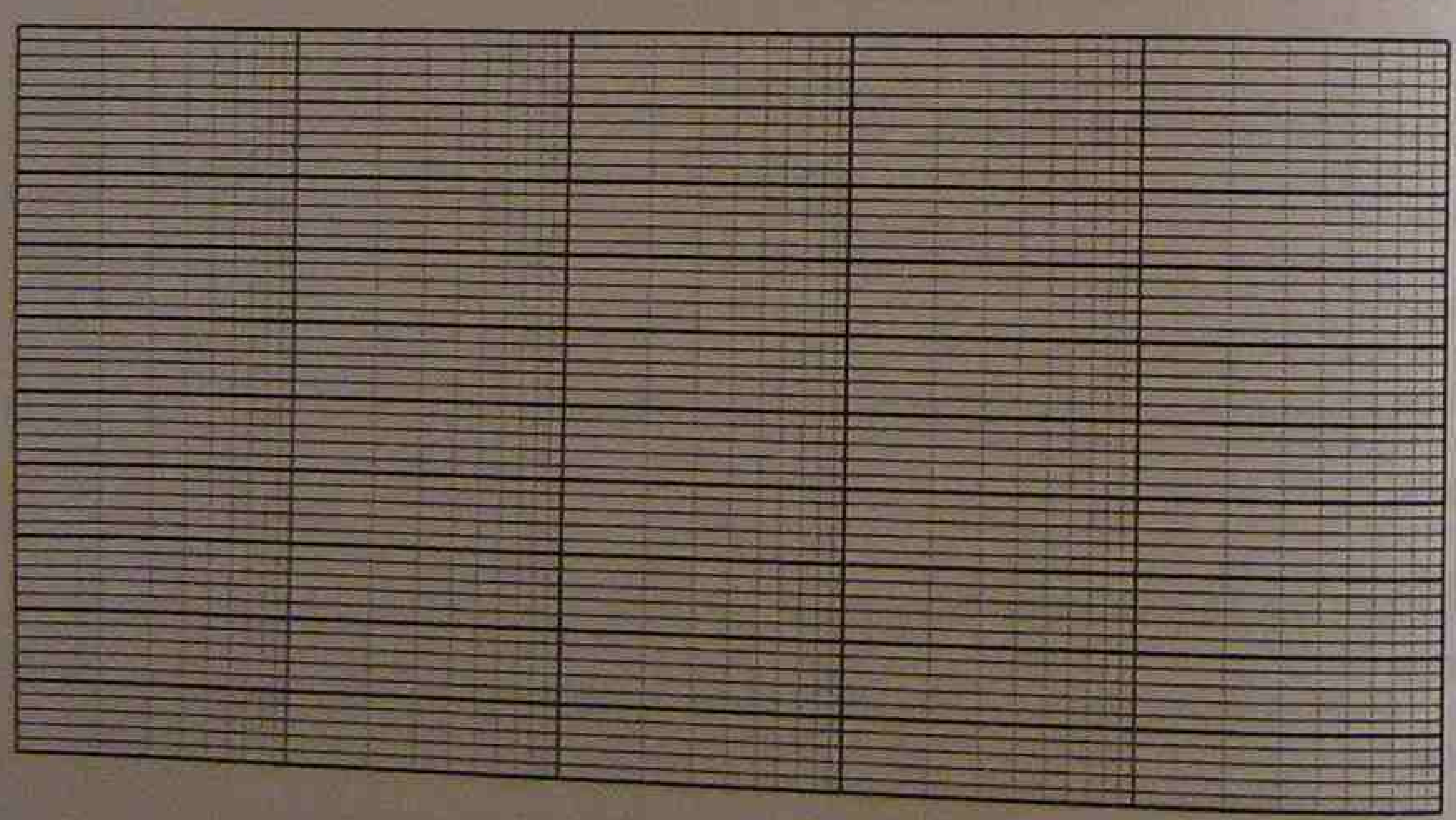
Frequency (Hz)	Figure 2.5				Figure 2.6			
	Input voltage (mv)	Output voltage (V)	Voltage Gain		Input voltage (mv)	Output voltage (V)	Voltage Gain	
			ratio	dB			ratio	dB
10								
20								
40								
80								
100								
200								
400								
800								
1k								
2k								
4k								
8k		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



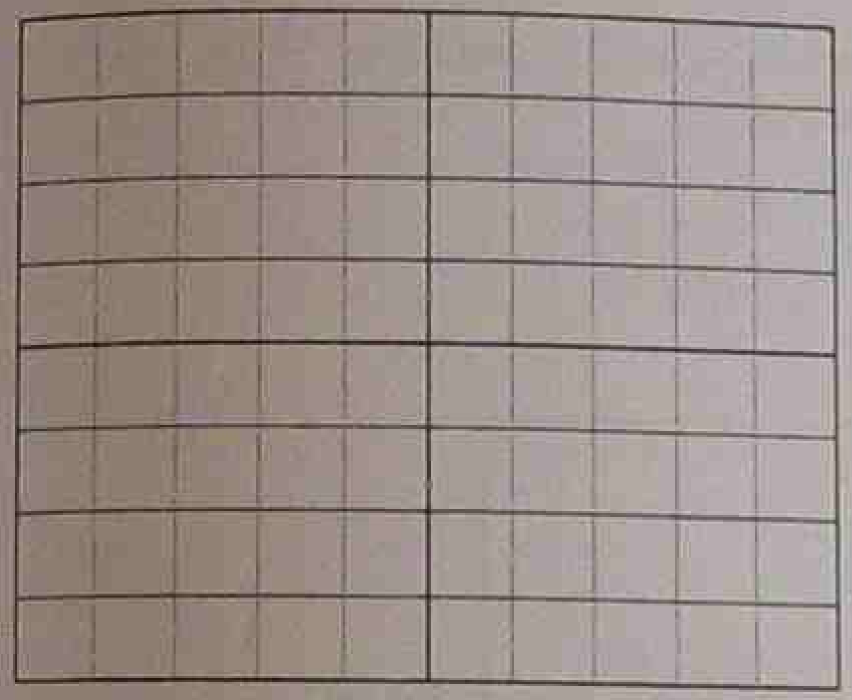
9. Determine:

(A) Mid-band voltage gain.

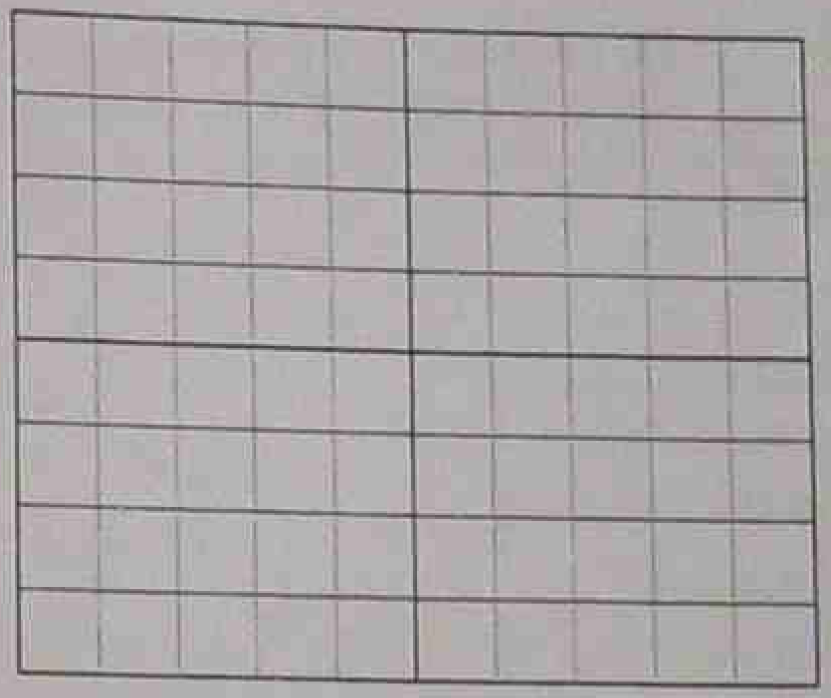
(B) Bandwidth of the amplifier.

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

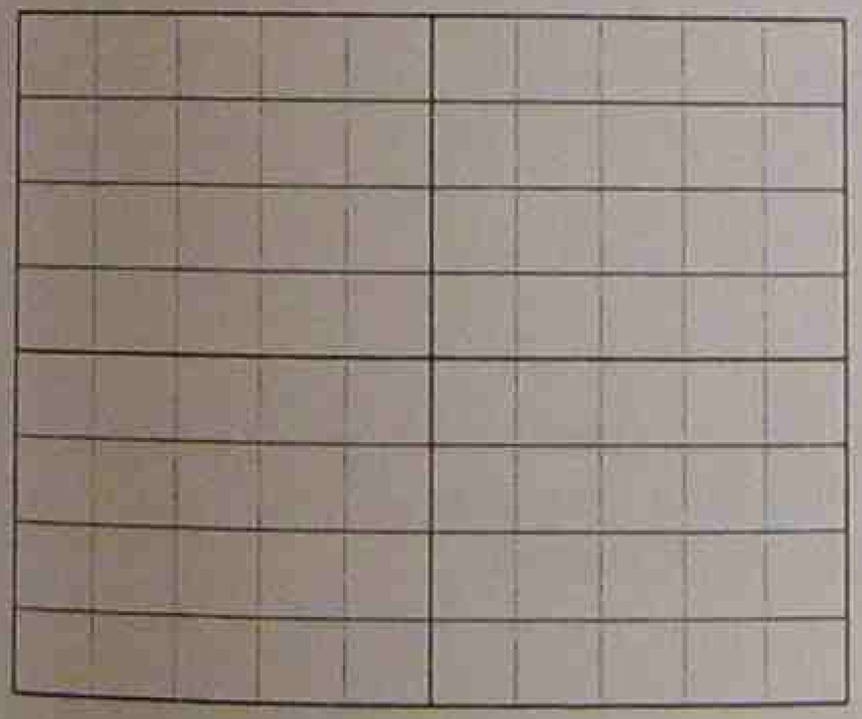
100Hz



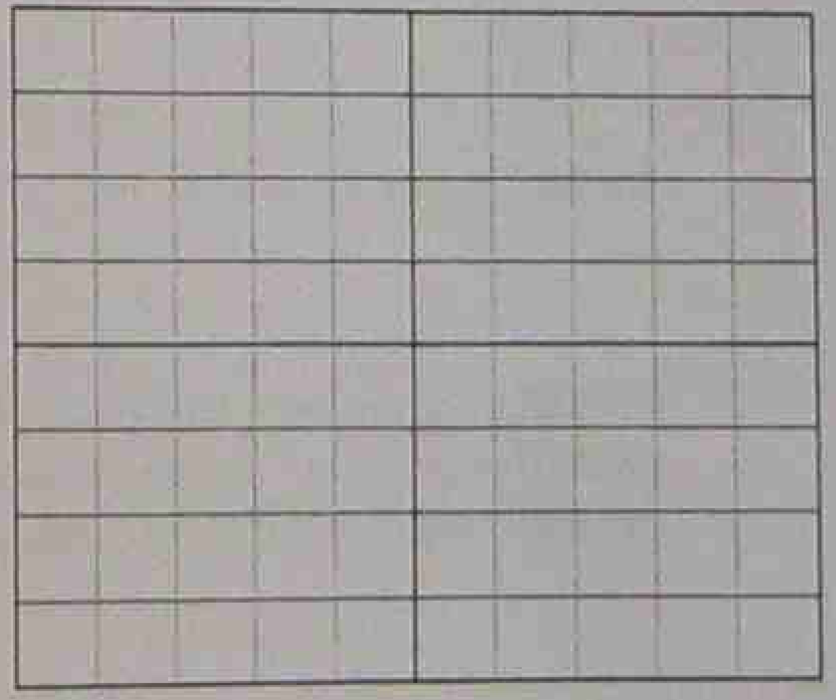
1kHz



10kHz



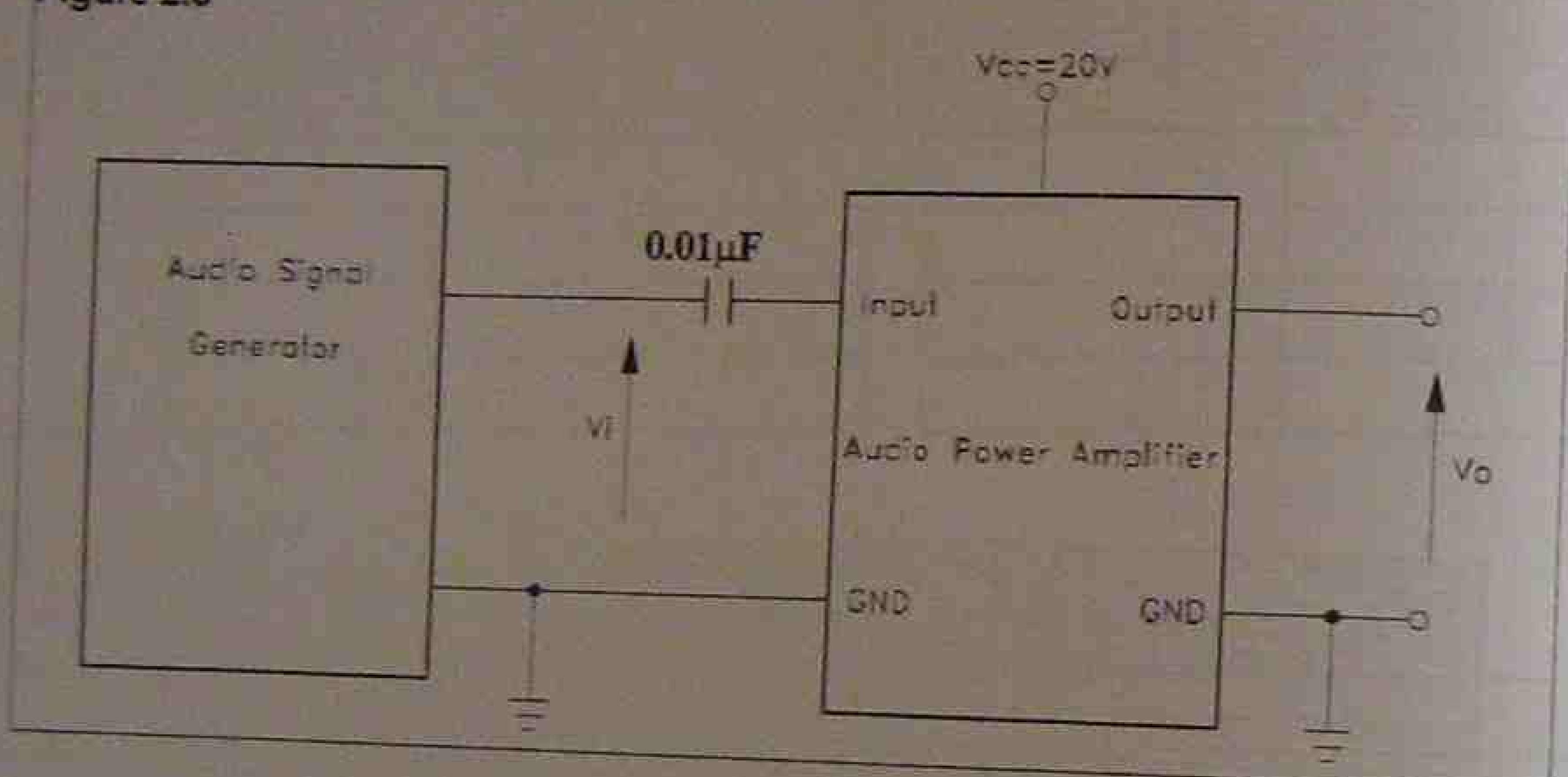
100kHz



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



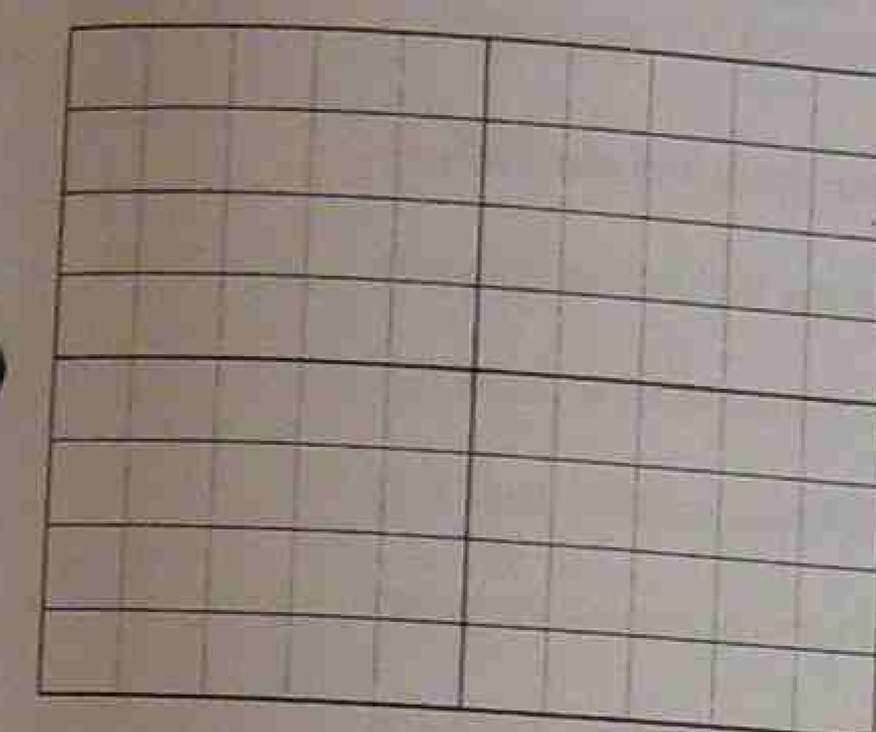
(A) half power frequencies are:

(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.

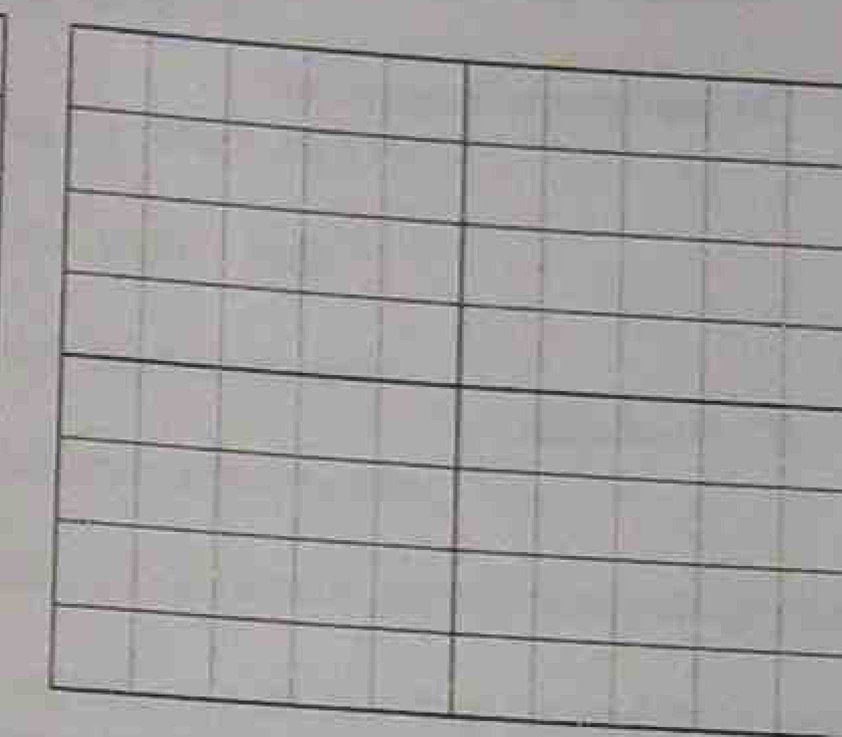
(C) Determine the bandwidth.

(D) Square wave response.

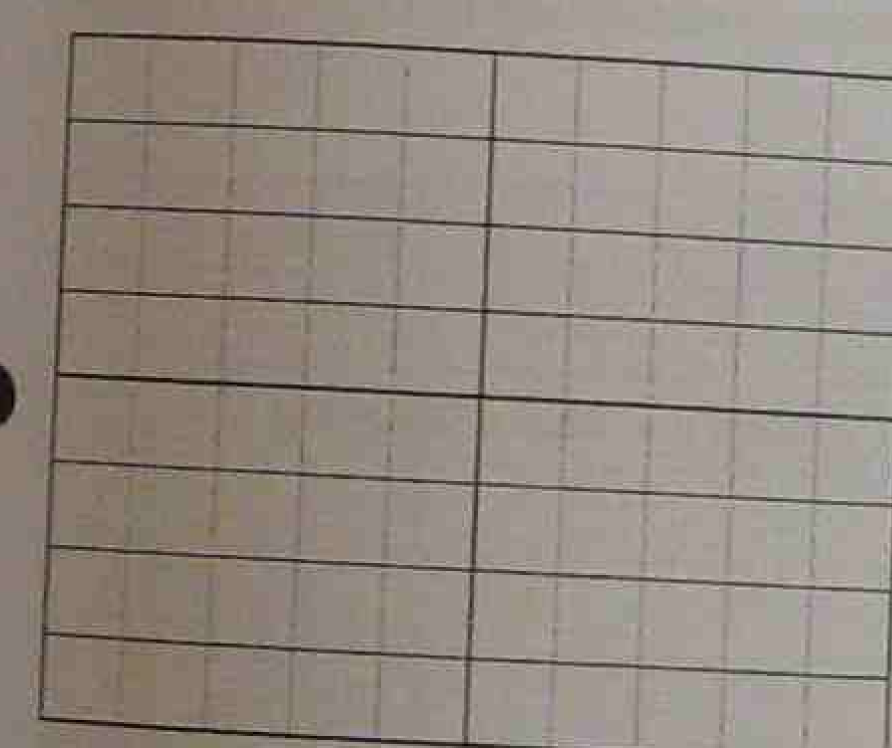
100Hz



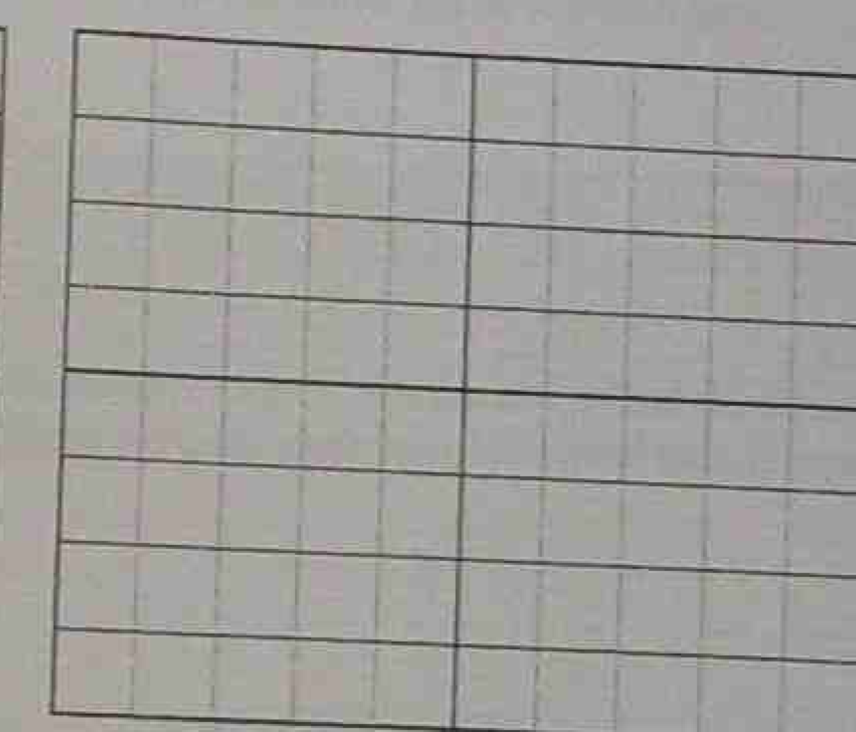
1kHz



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

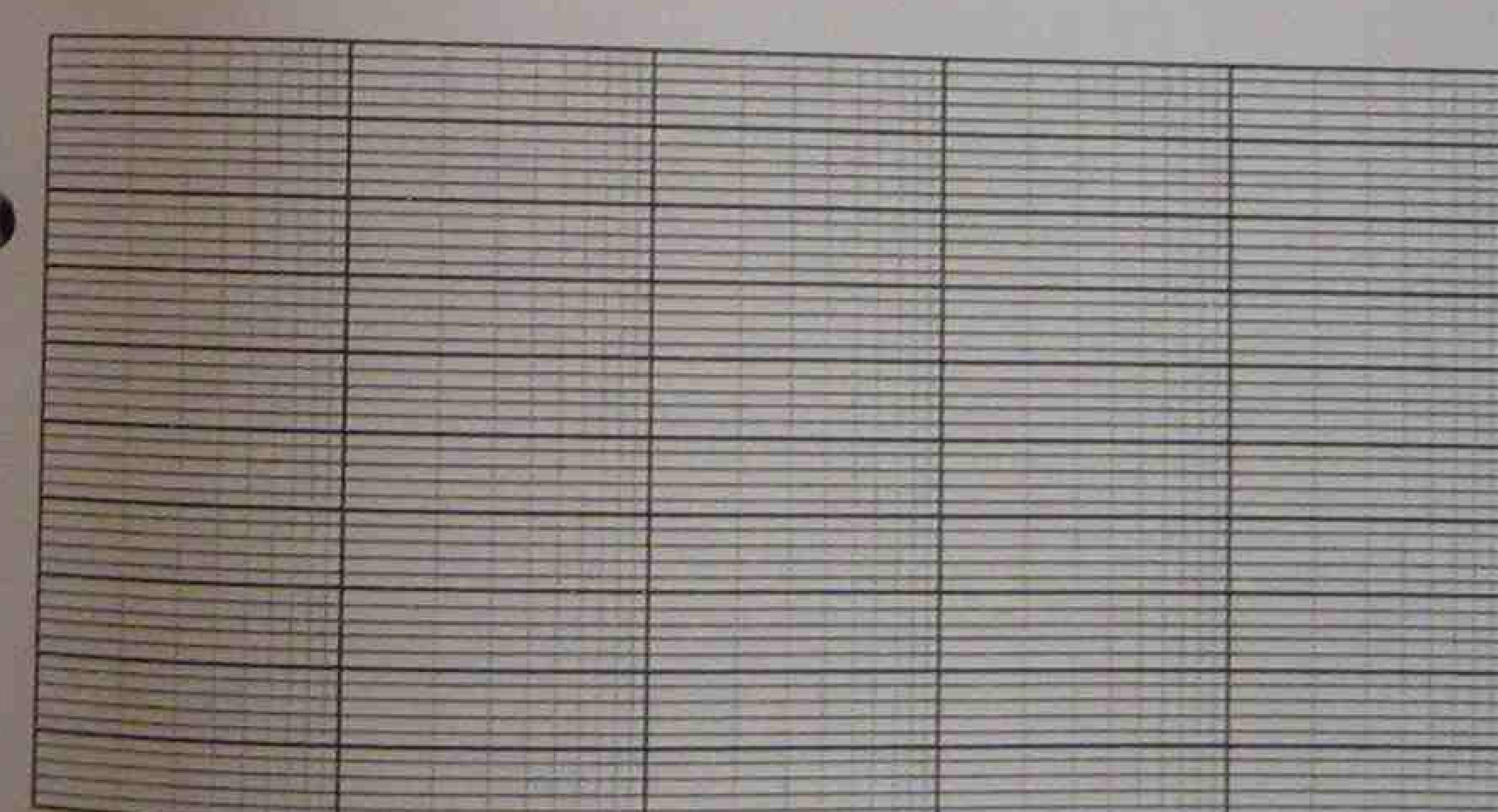
(C) Mark voltage gain for mid-band frequencies.

(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

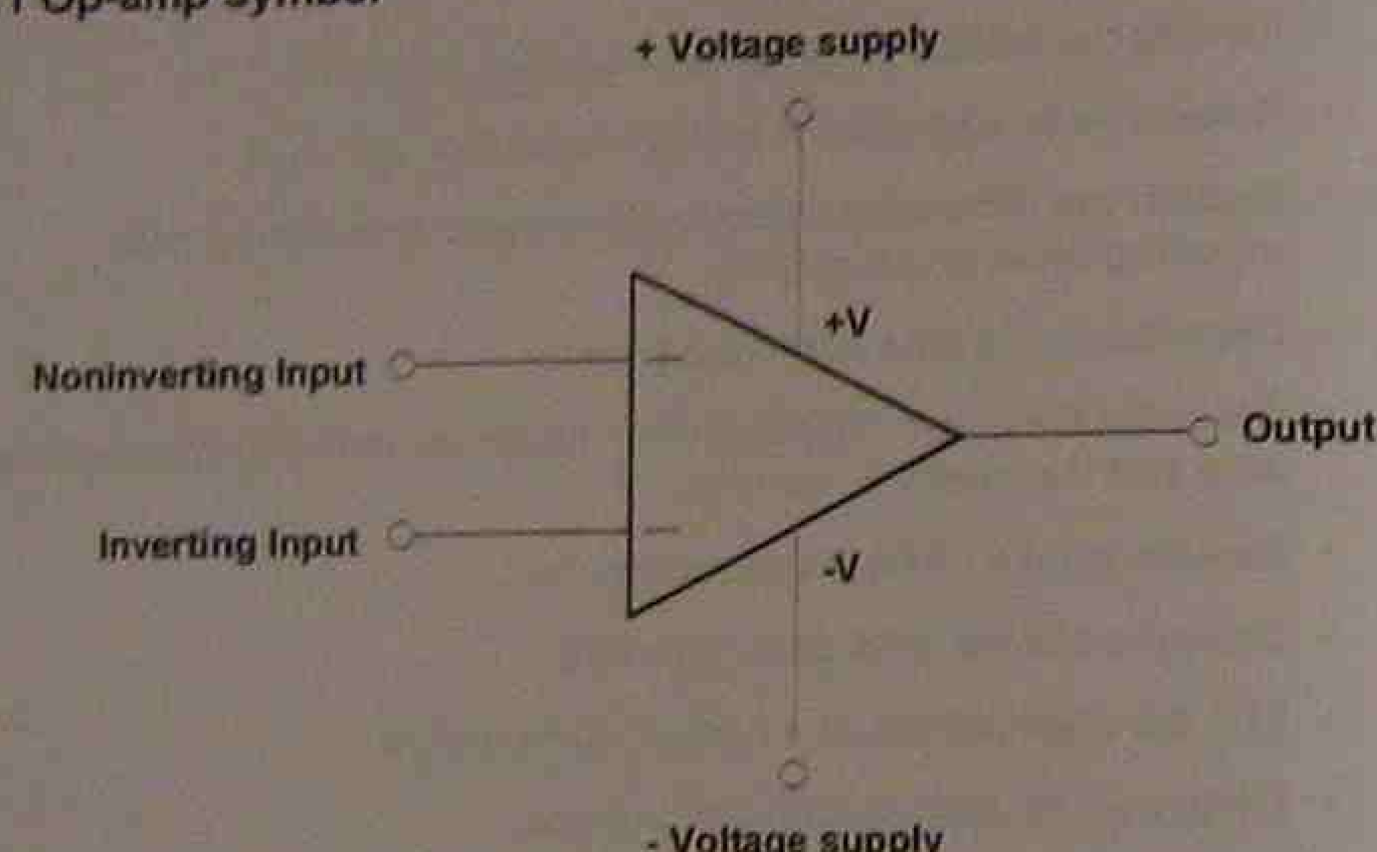
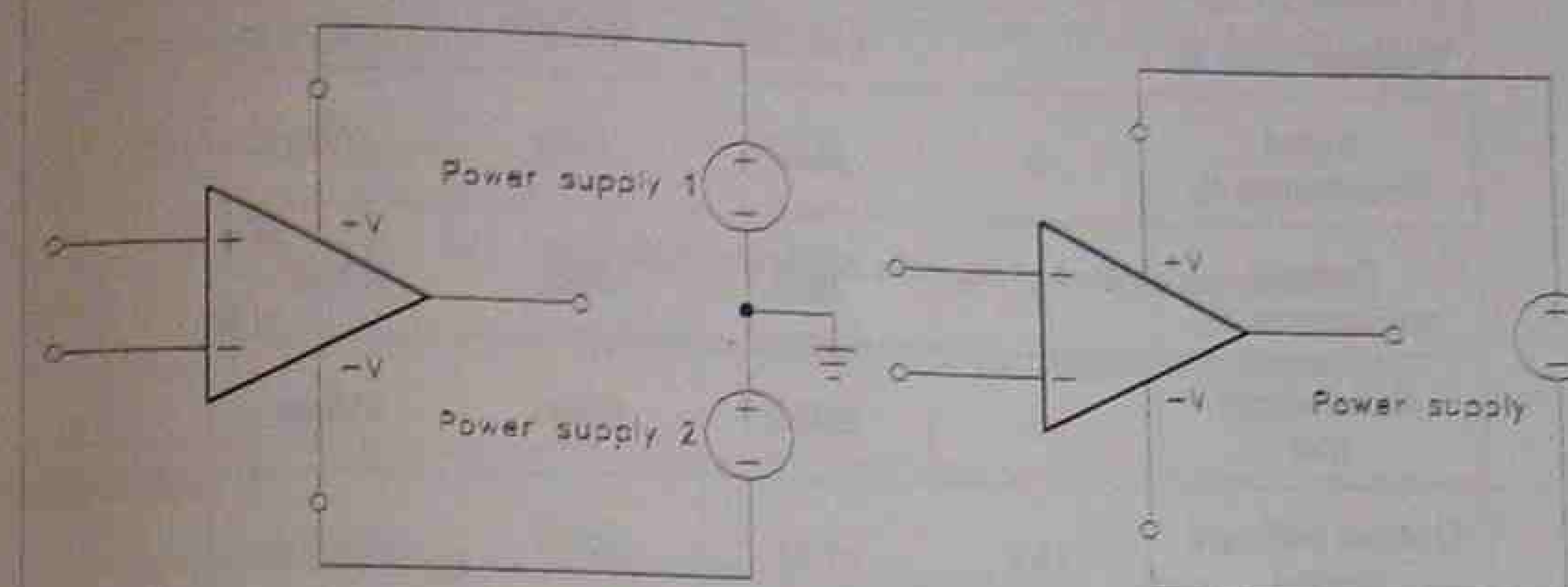


Figure 3.1 shows the standard op-amp symbol. Firstly, it has two input terminals, called the noninverting (+) input and the inverting (-) input. It amplifies the voltage difference between noninverting input and inverting input. The input names are from the polarity (phase) difference between input terminal 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.

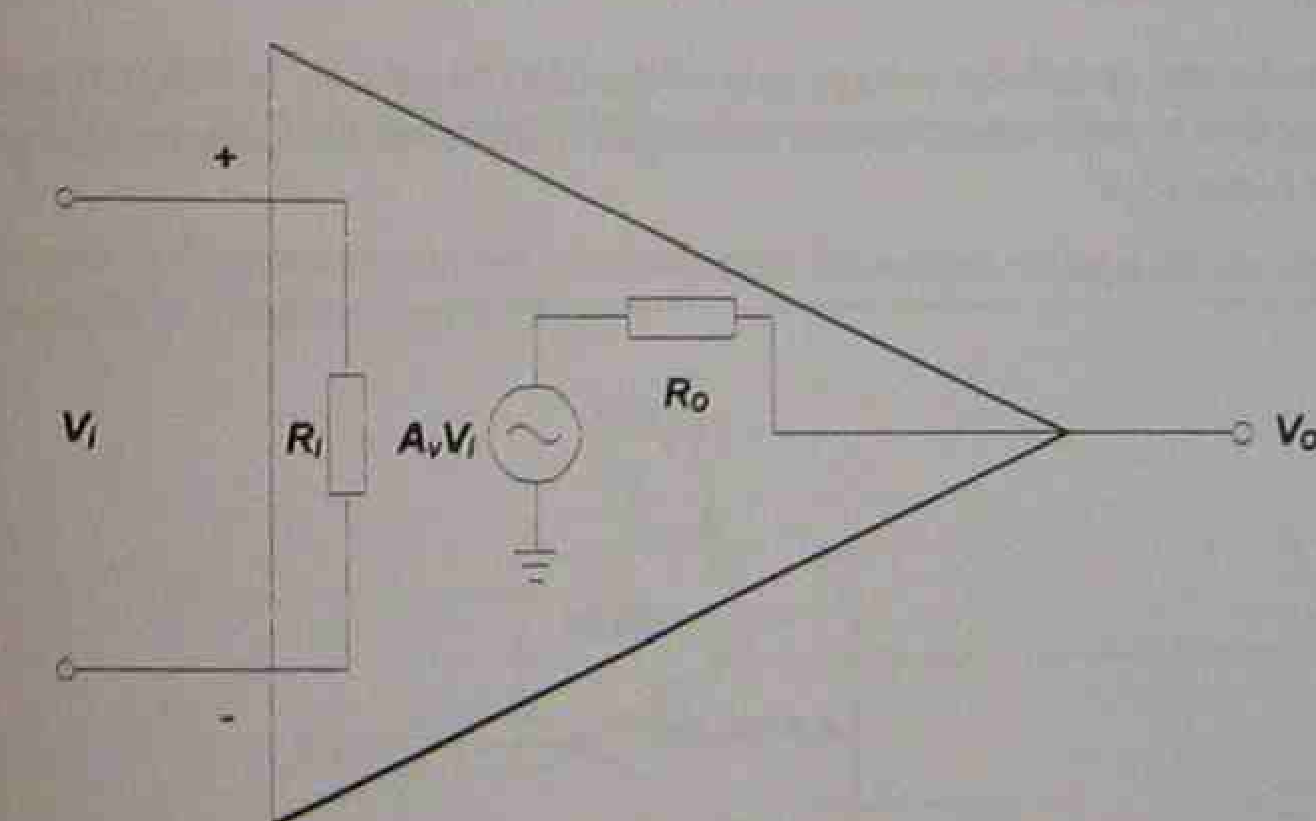


(a) CORRECT power supply wiring (b) WRONG power supply wiring

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)

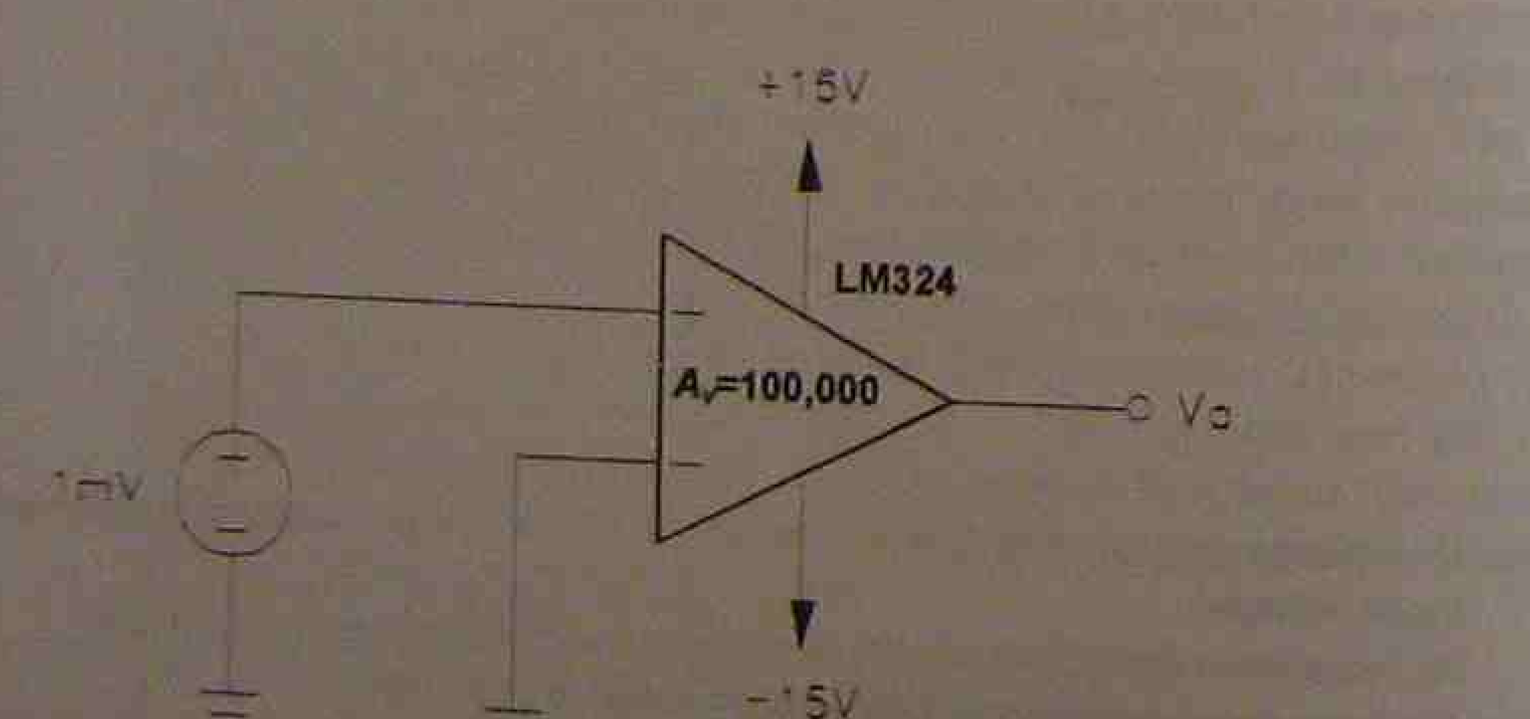
Characteristics	Ideal op-amp	Practical op-amp		
		LM741	LM 324	LF351
Voltage gain A_v	∞	200,000	100,000	100,000
Input resistance R_I	∞	2MΩ	1MΩ	1TΩ
Output resistance R_O	0	75Ω	75Ω	270Ω
Bandwidth BW	∞	2MHz	1MHz	4MHz
Output voltage swing	±15V	±13V	±13V	±13.5V

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:

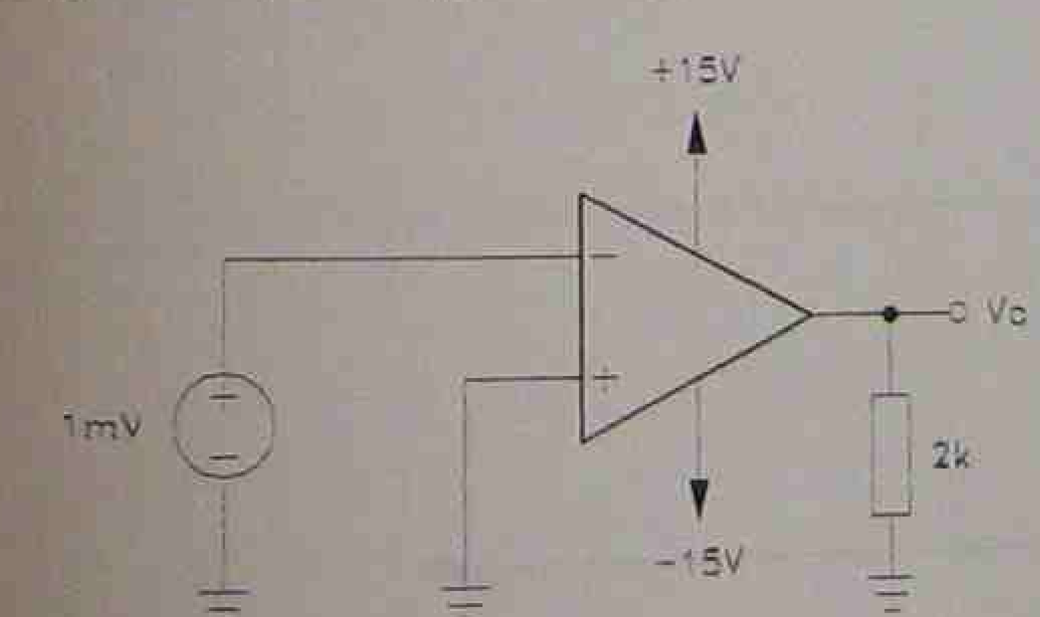
- If $V_{I+} > V_{I-}$, V_o = positive supplied voltage (positive saturation)
- If $V_{I+} < V_{I-}$, V_o = negative supplied voltage (negative saturation)

Drill Question 1

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

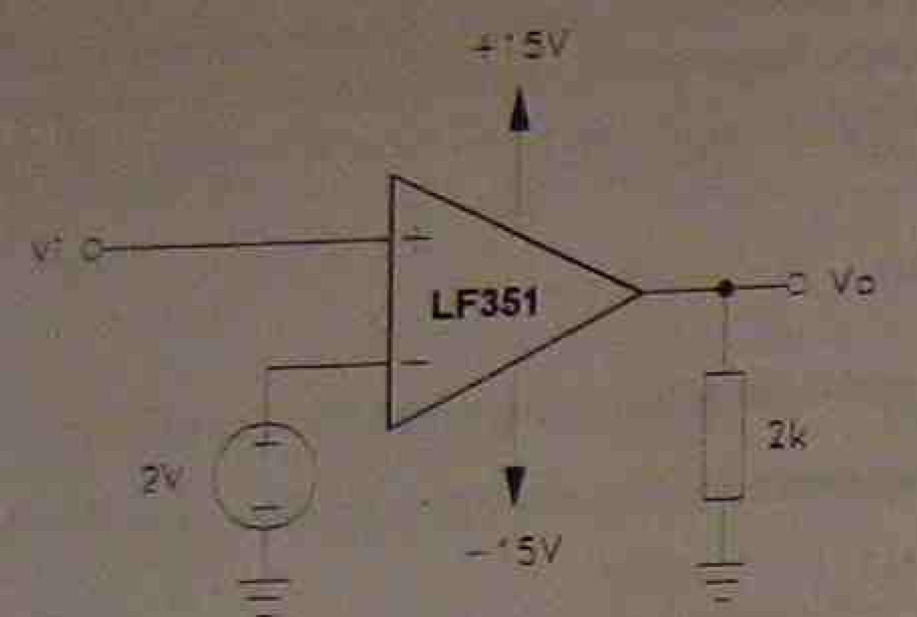
Drill Question 2

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



Drill Question 3

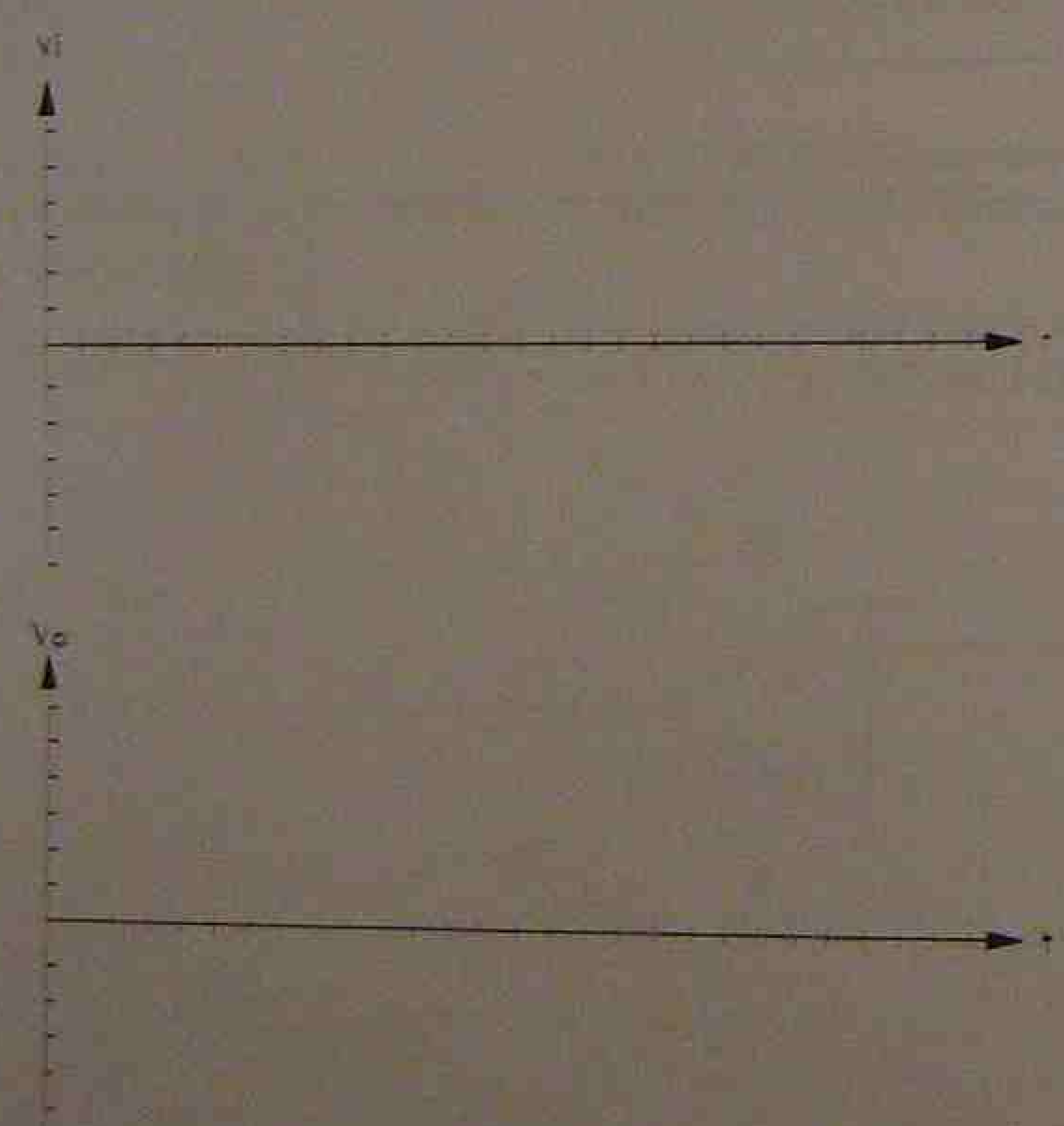
For the circuit diagram shown below, determine the output voltages for the input voltages shown in table below.



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

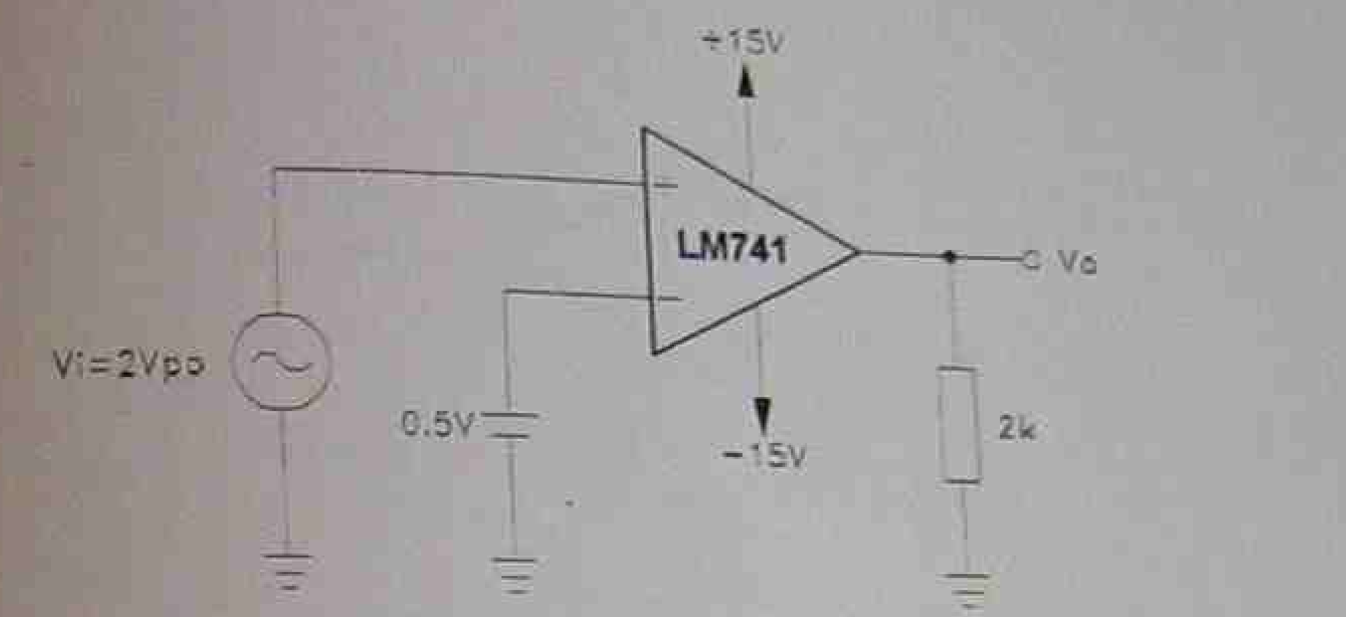
Drill Question 4

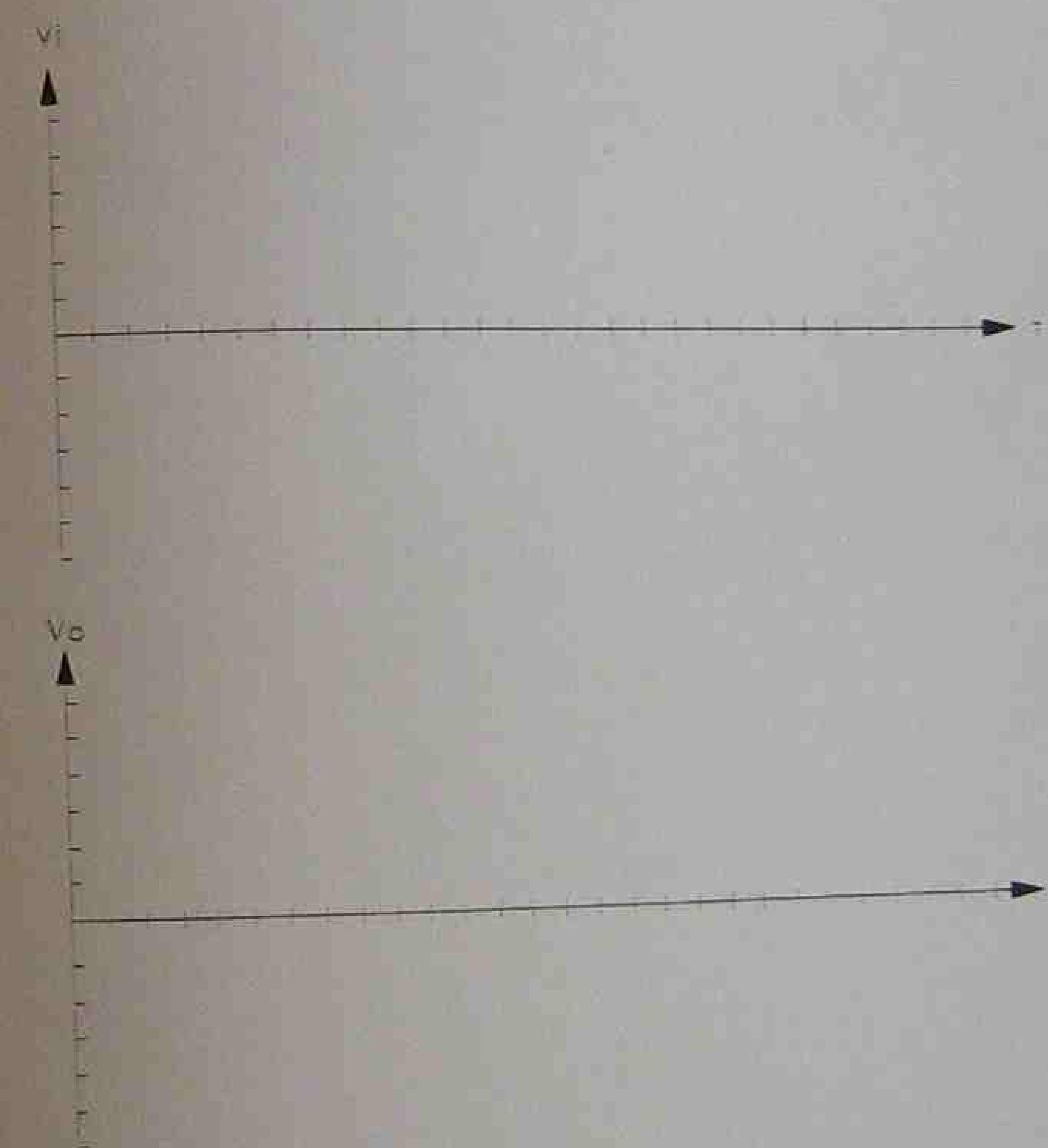
For the circuit diagram in Figure 3.4, the 1mV dc voltage source is replaced with 2Vpp 1kHz ac signal. Draw both input and output waveforms.



Drill Question 5

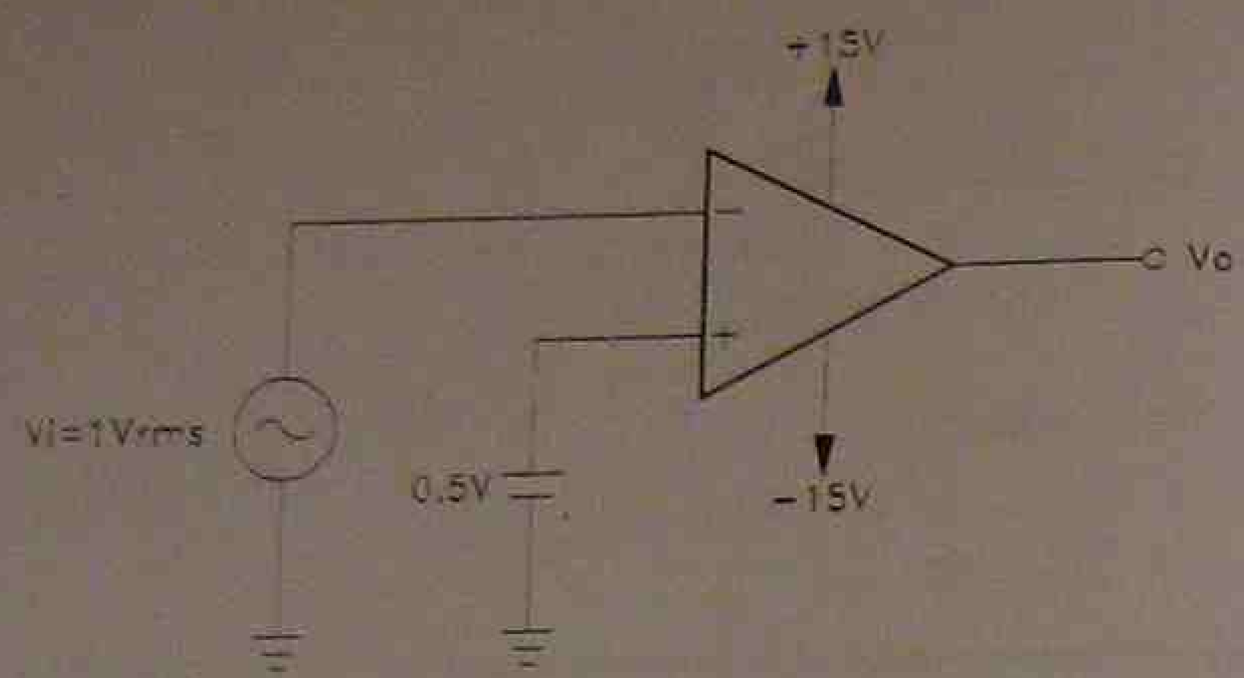
For the circuit diagram shown below, draw both input and output waveforms.

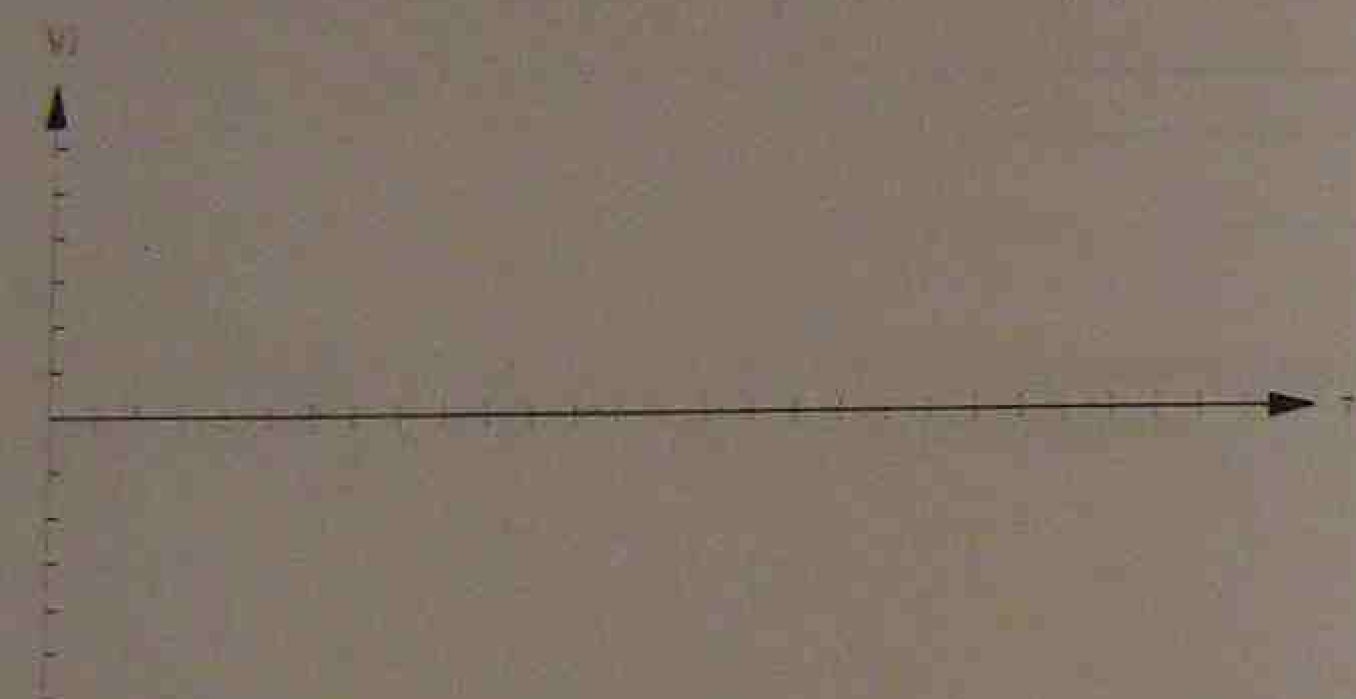


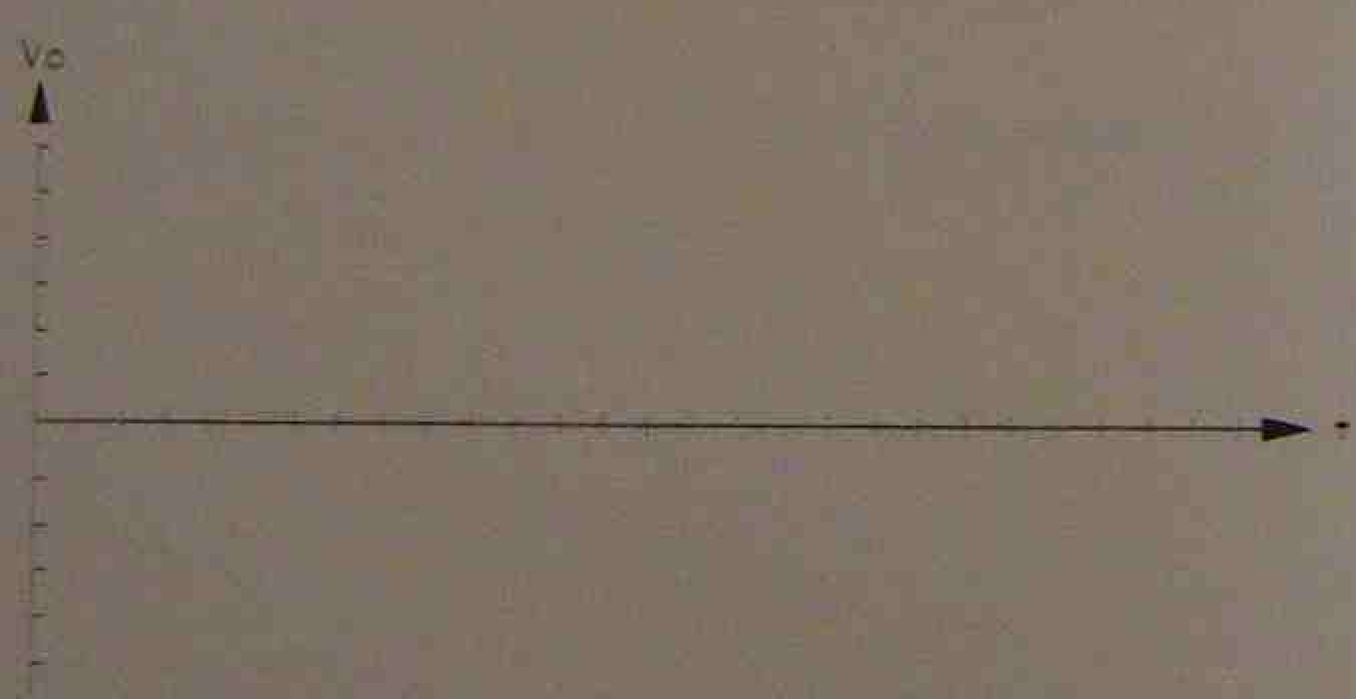


Drill Question 6

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

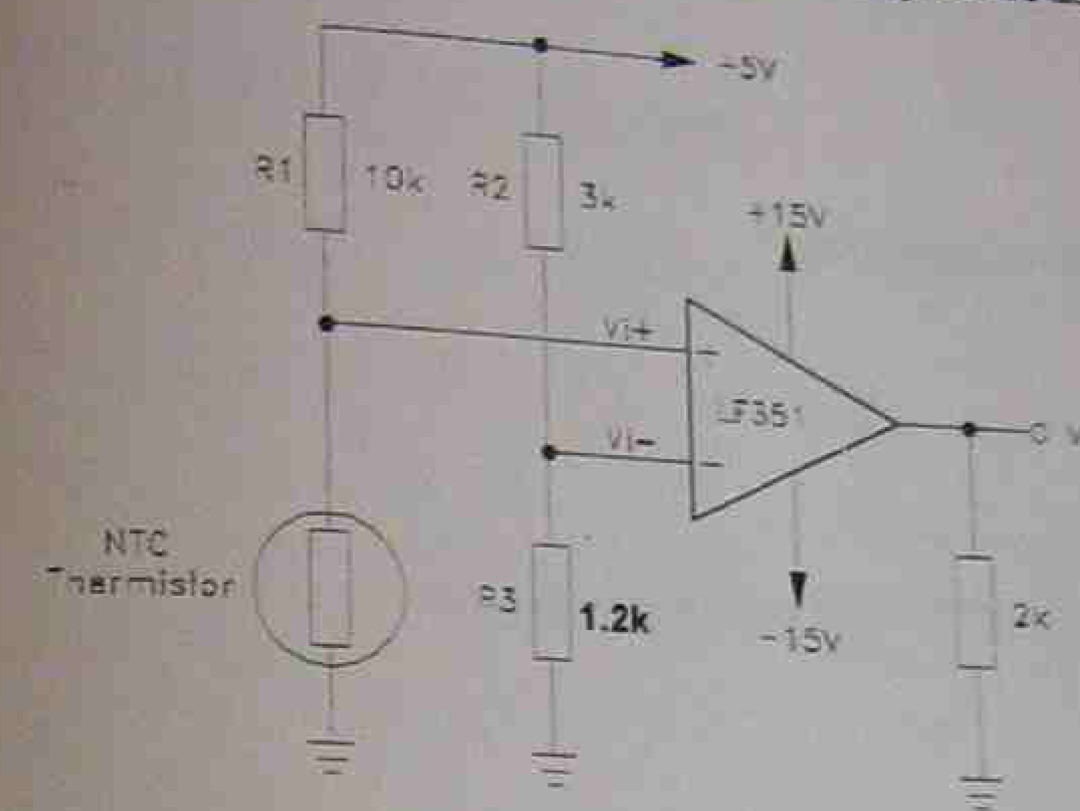






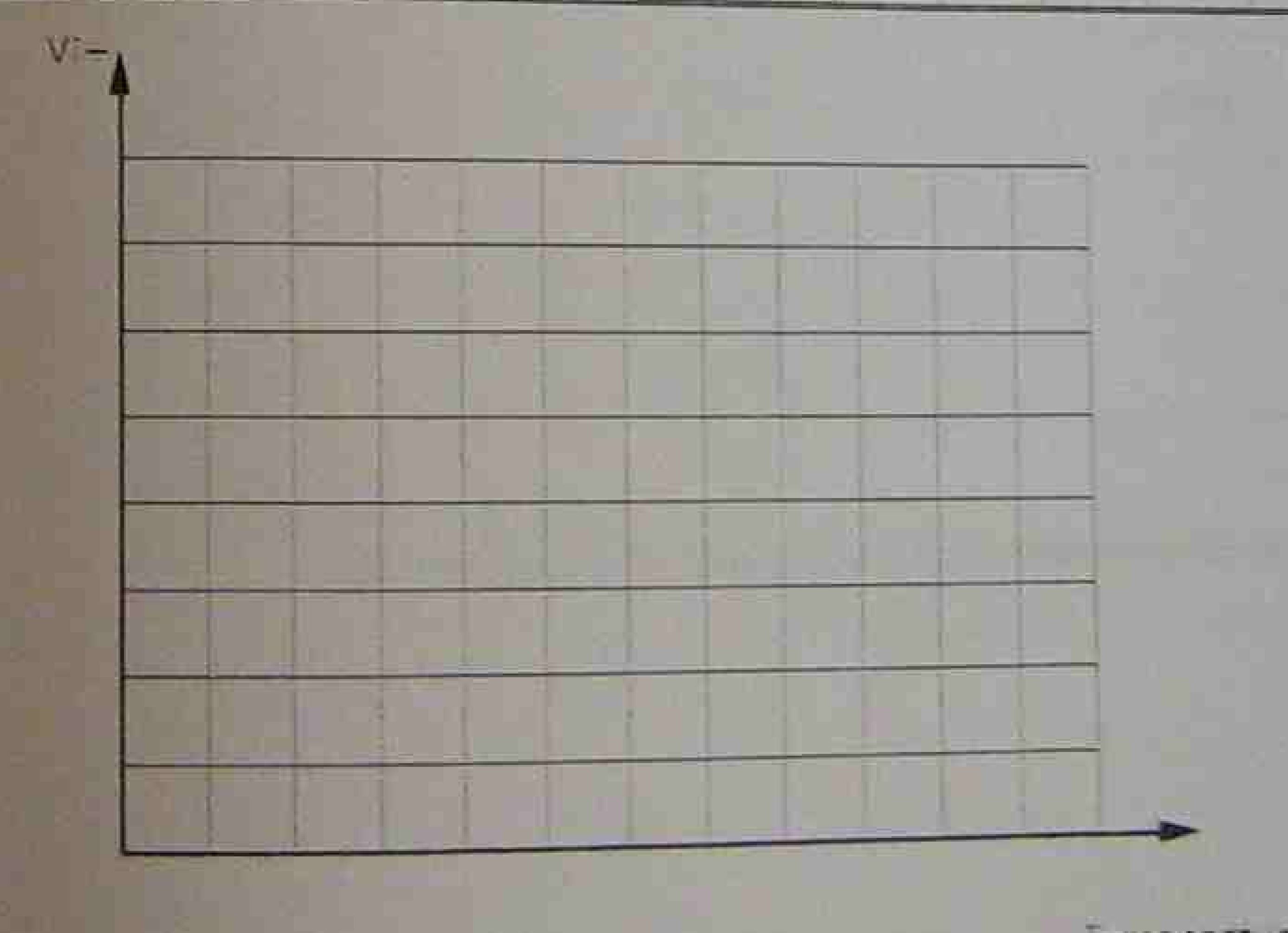
Drill Question 7

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.

Temperature (°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.

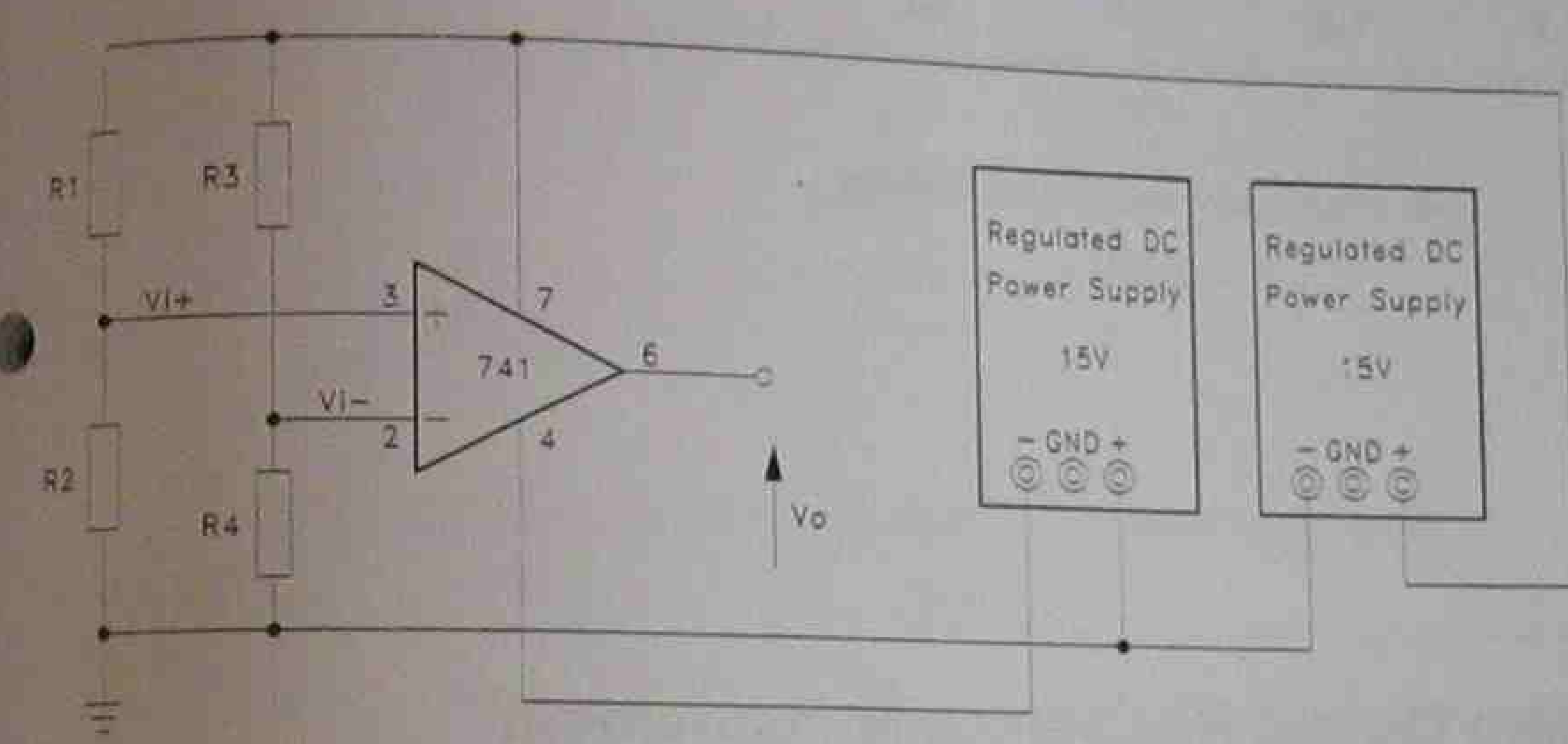
(e) (optional) Show a relay interface circuit to the output terminal to operate a 240V heating element.

(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 =

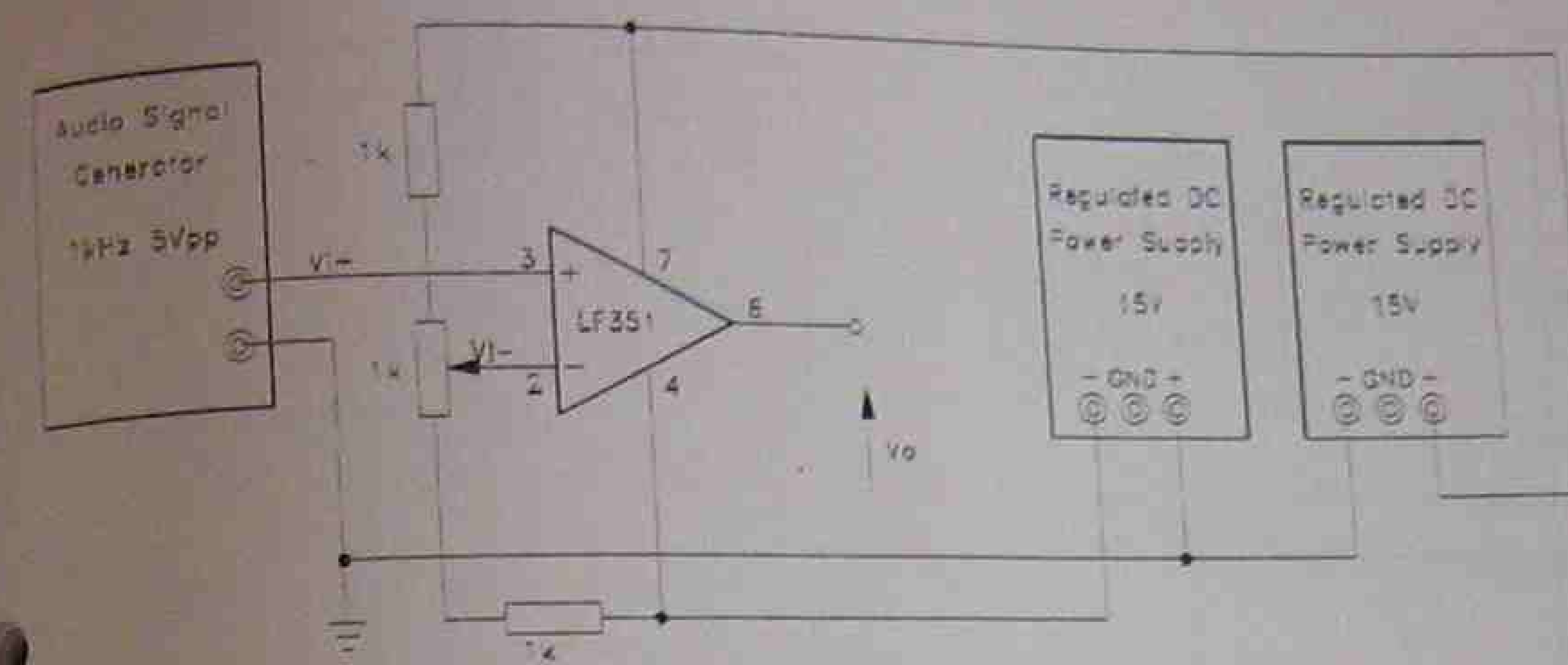
2. Connect 1.2k load between the output terminal (pin 6) and ground. Measure the output voltage (V_L) and calculate the output resistance using the equation in page 1-20.

Ro =

3. Compare with manufacturer's specification.

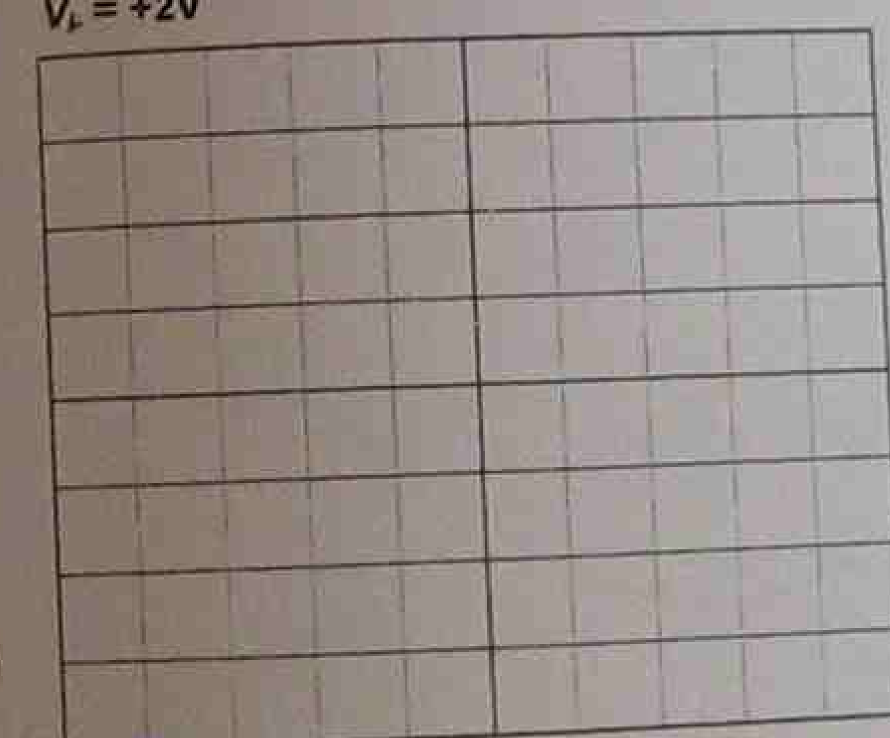
Pulse Width Modulation

1. Construct the circuit shown below.

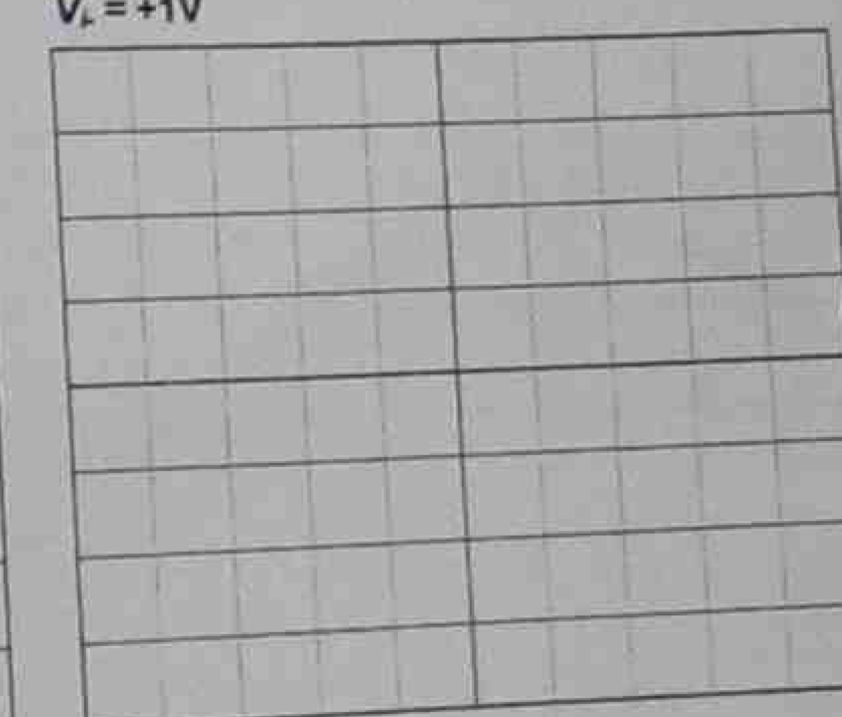


2. Sketch the V_{i+} and V_O waveforms for each of the V_{i-} shown below.

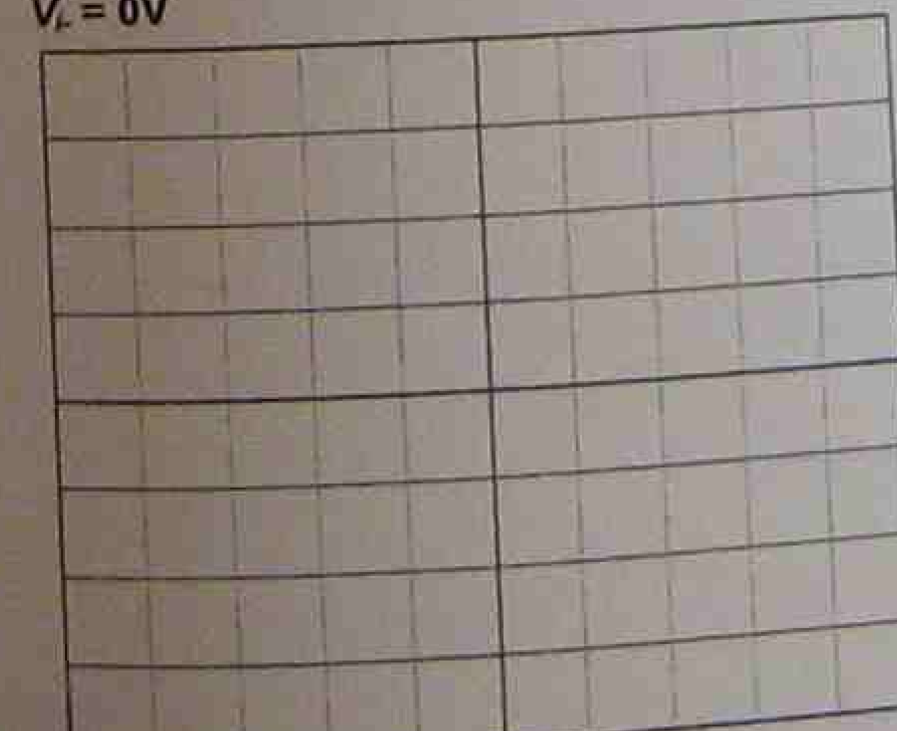
V_{i-} = +2V



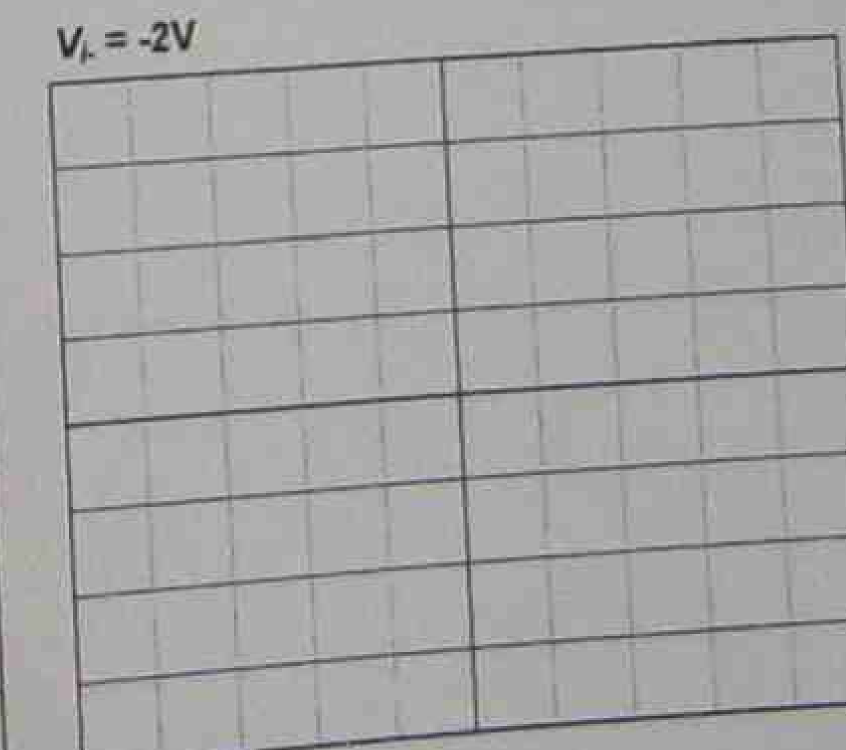
V_{i-} = +1V



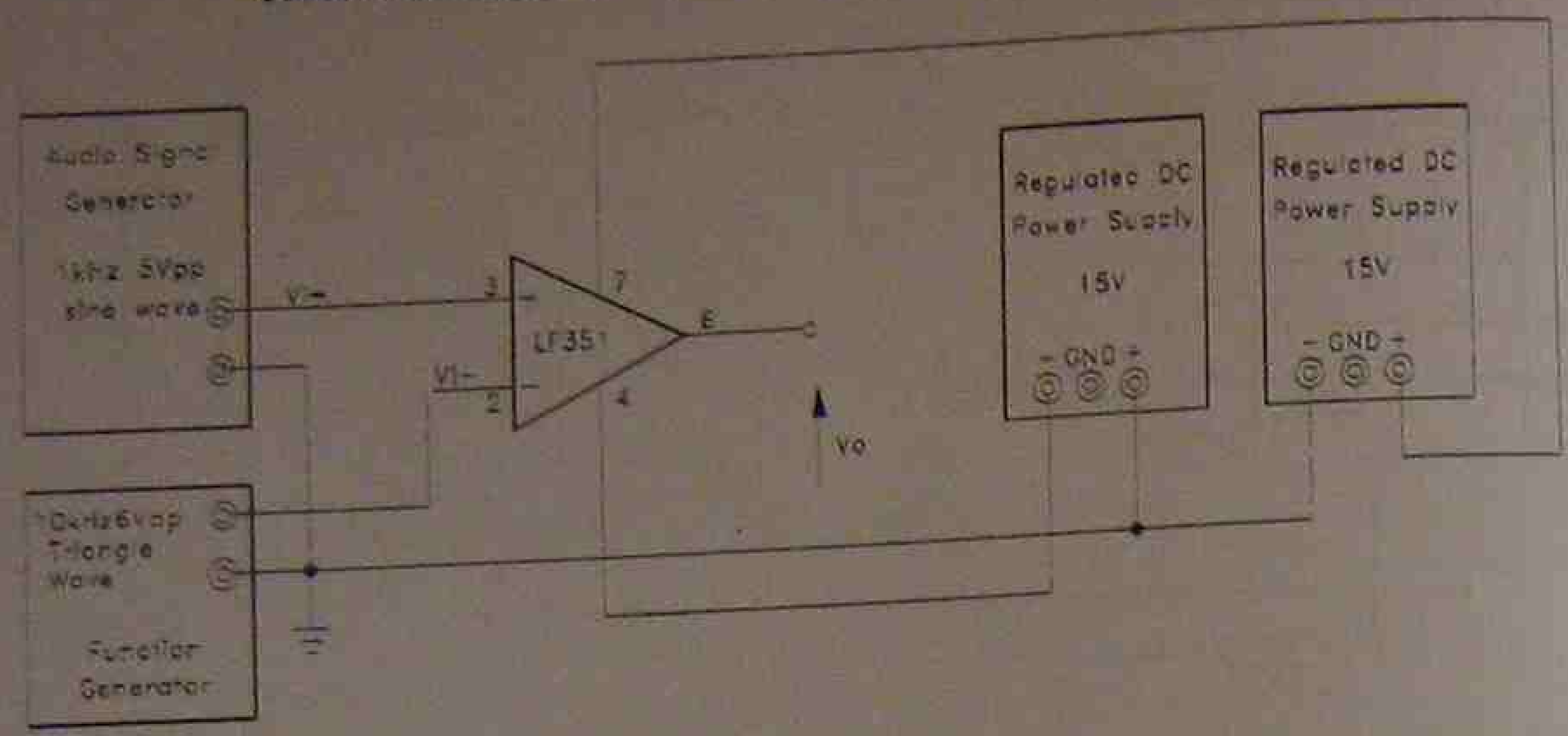
V_{i-} = 0V



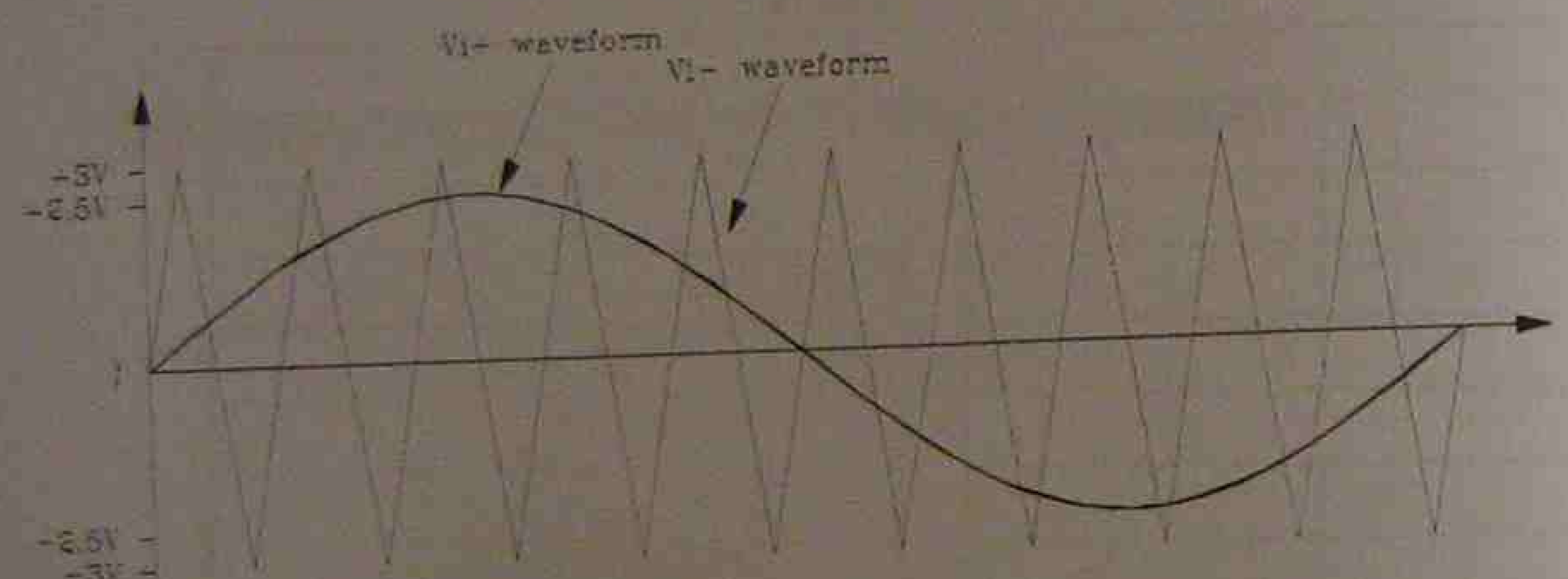
V_{i-} = -2V



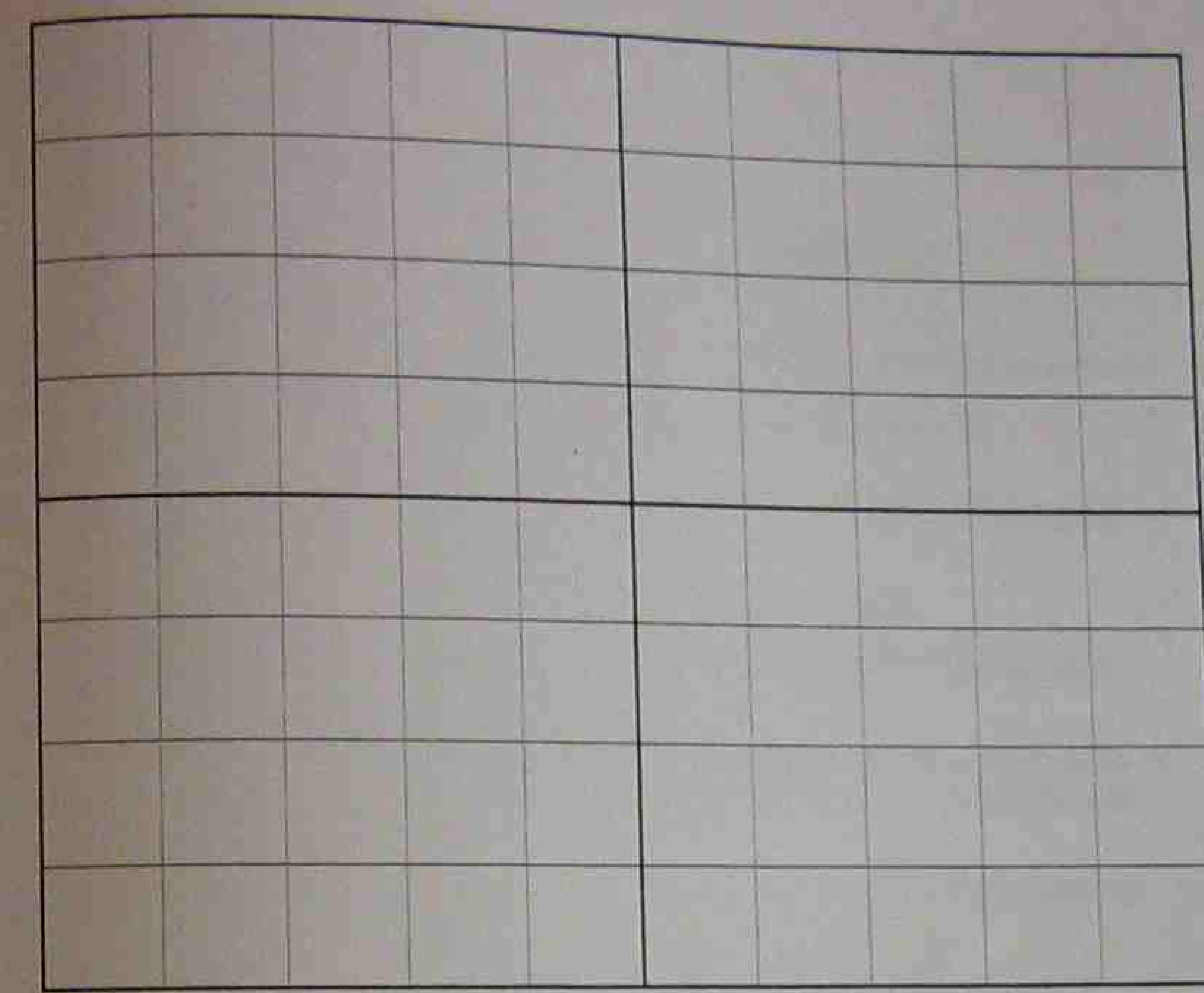
3. Replace voltage divider for V_{i-} with a function generator shown below then observe the output waveform.



(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

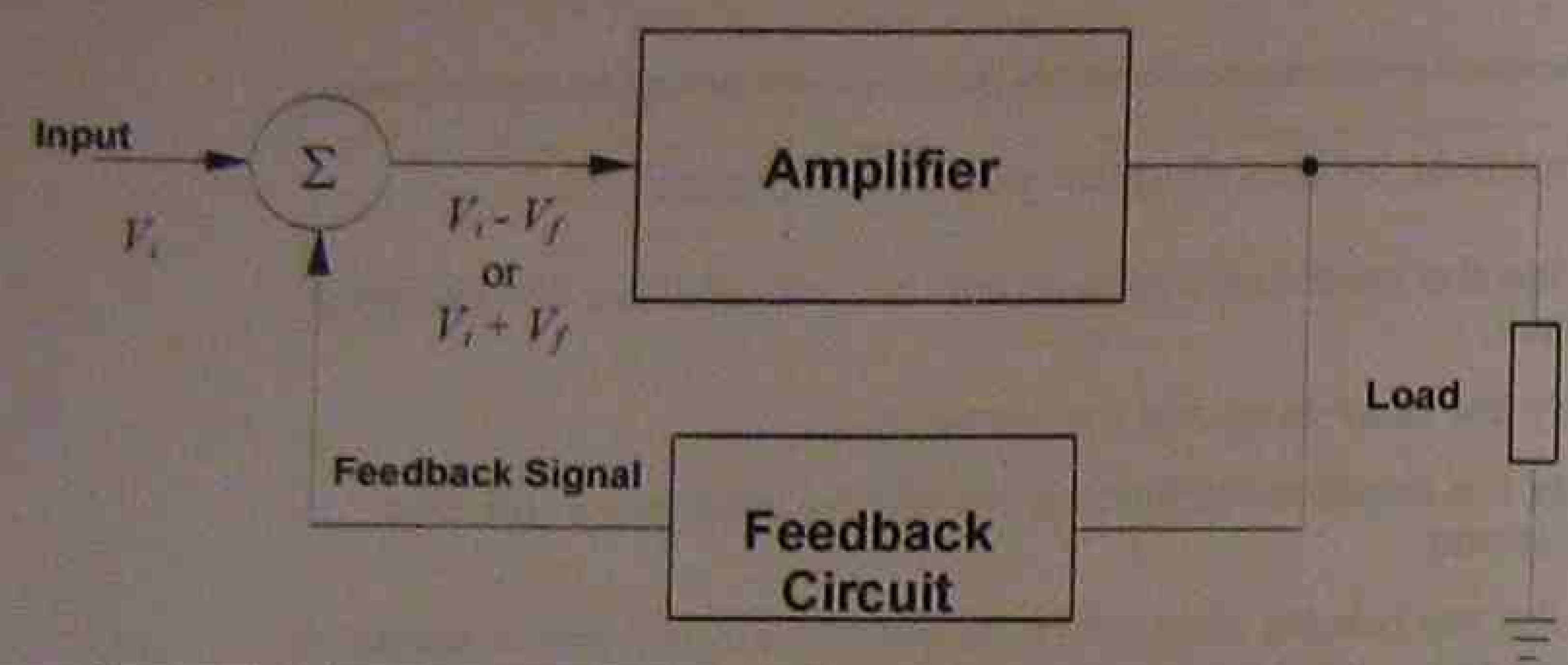
Upon completion of this chapter, you should be able to:

- 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

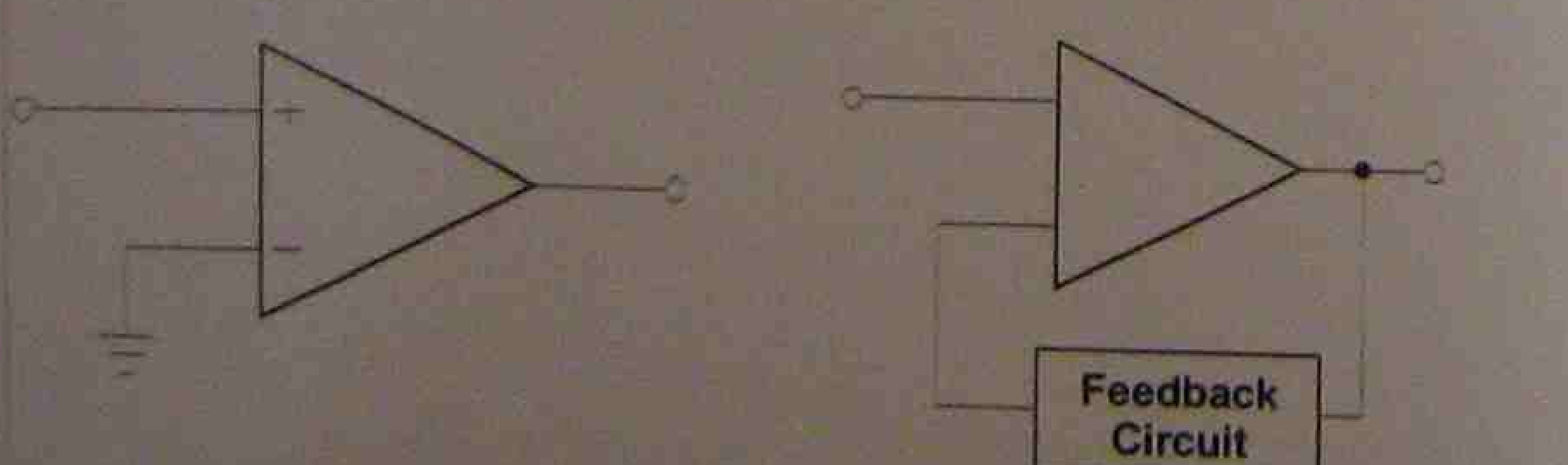


Open-loop and Closed-loop

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



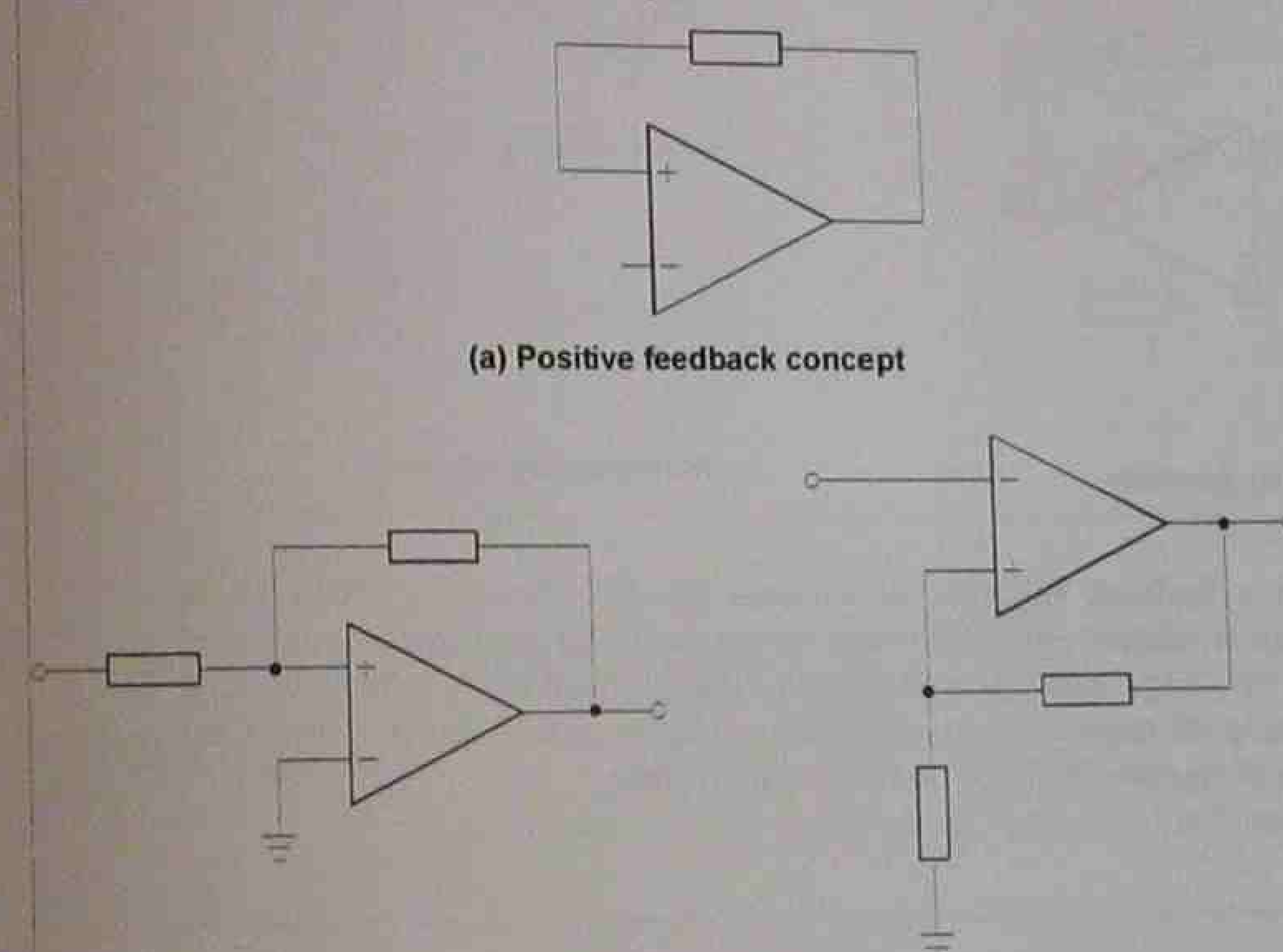
(a) Open-loop circuit

(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

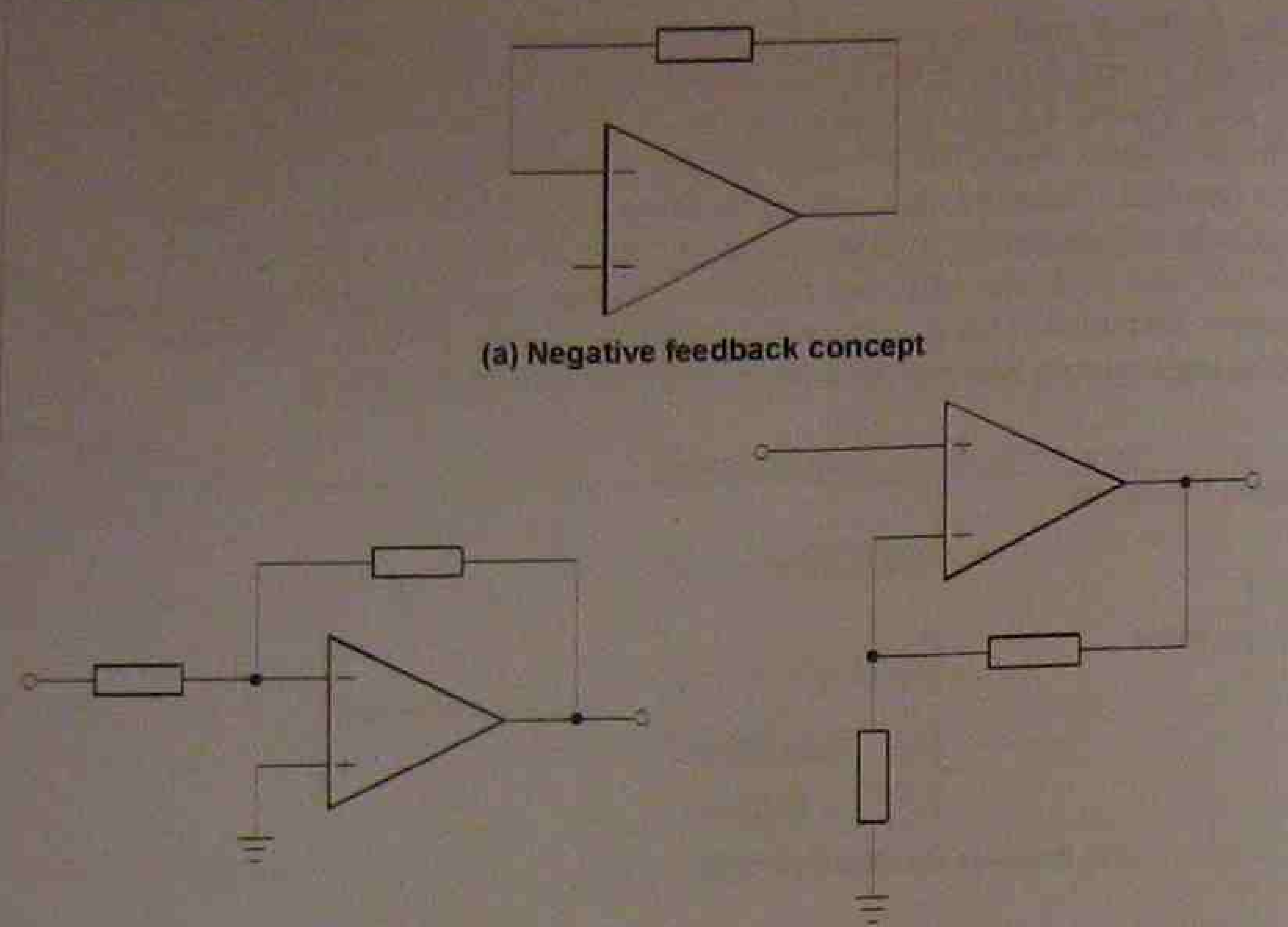
(b) Non-inverting Schmitt rigger

(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

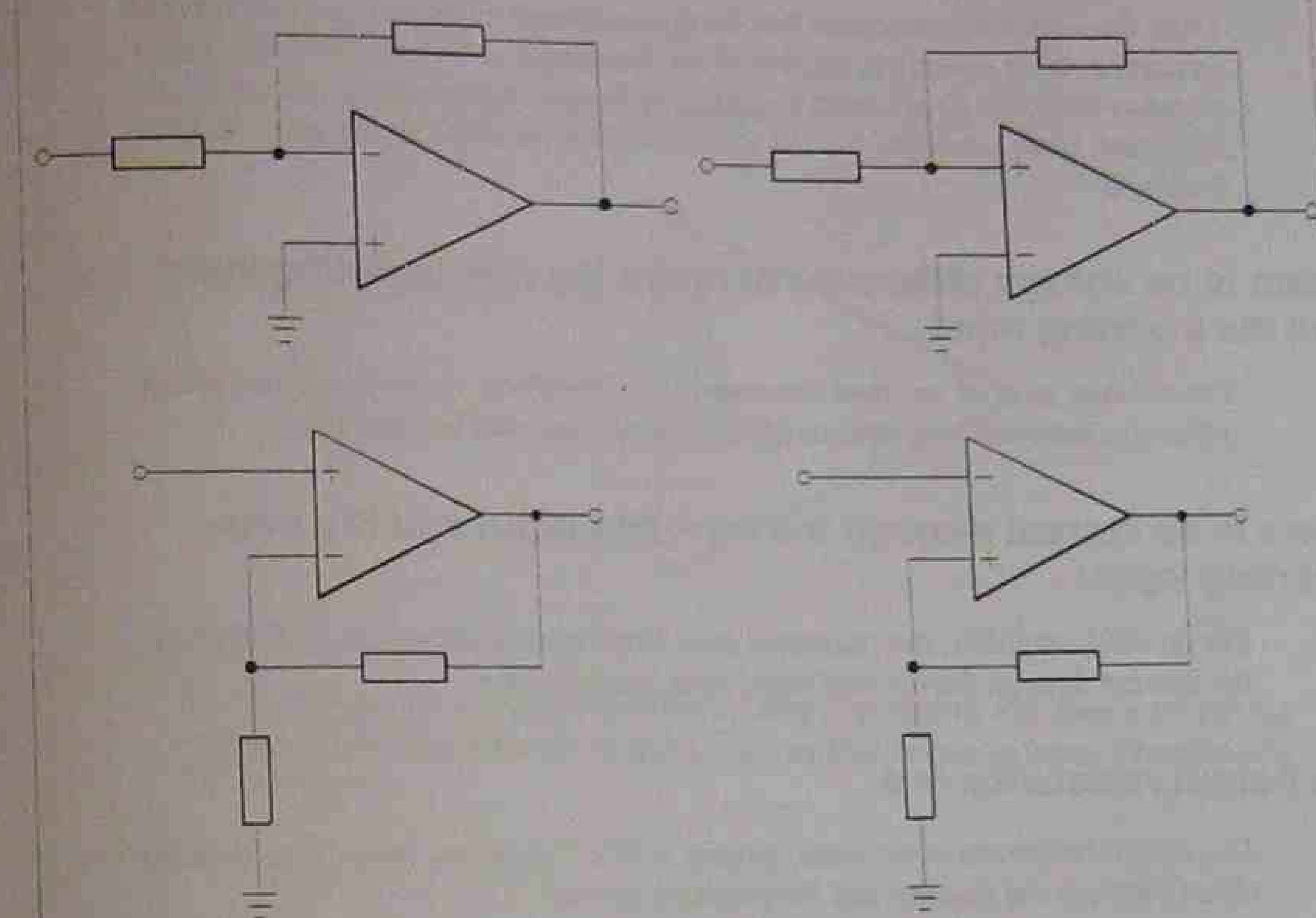
Negative feedback enhances all the desirable properties of a circuit at the expense of reduced gain. That is why the general purpose op amps are designed with a very large open-loop voltage gain. Negative feedback is extremely useful for nearly all control circuits and many signal circuits. Table 4.1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

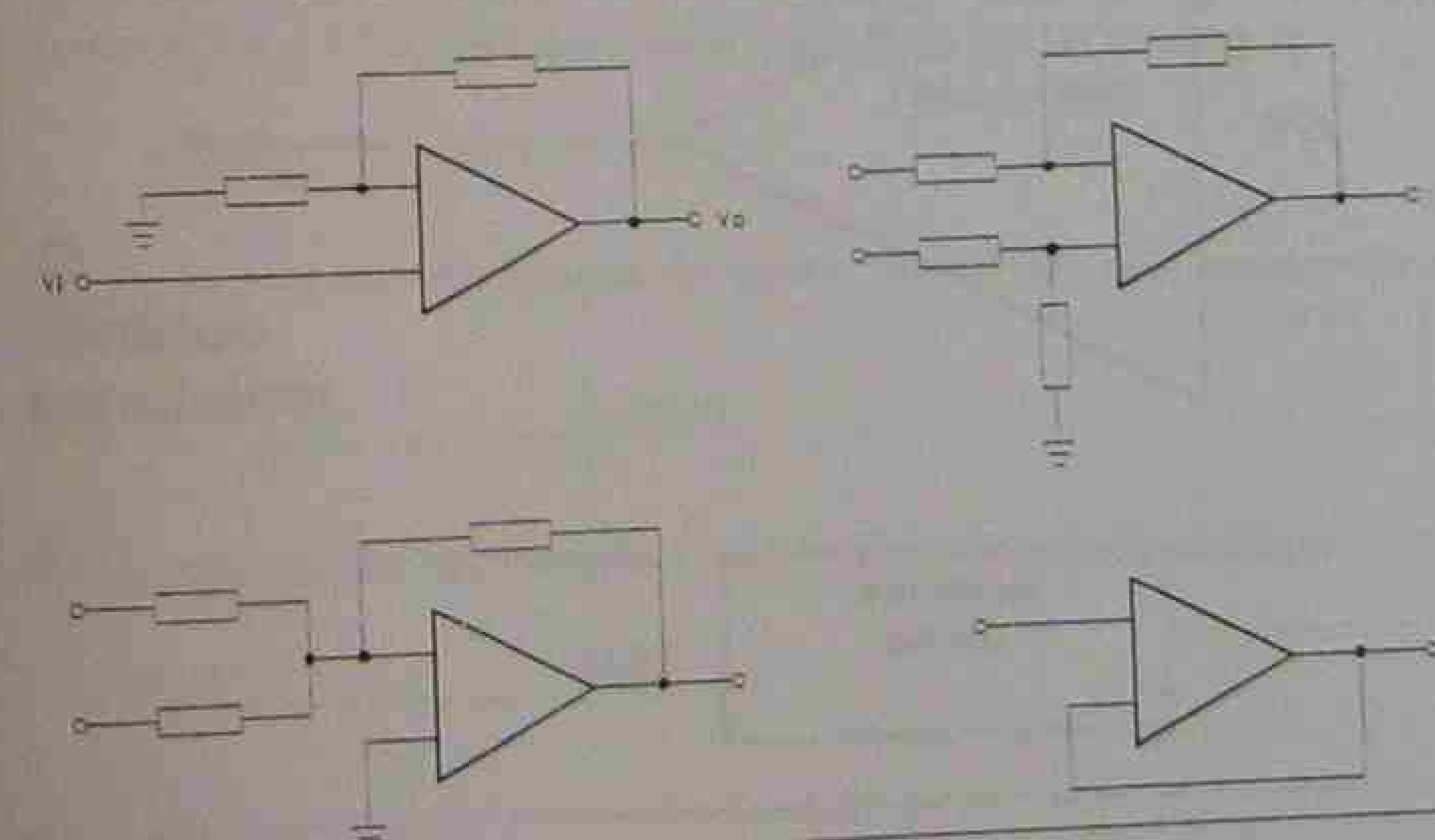
Drill Question 1

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.

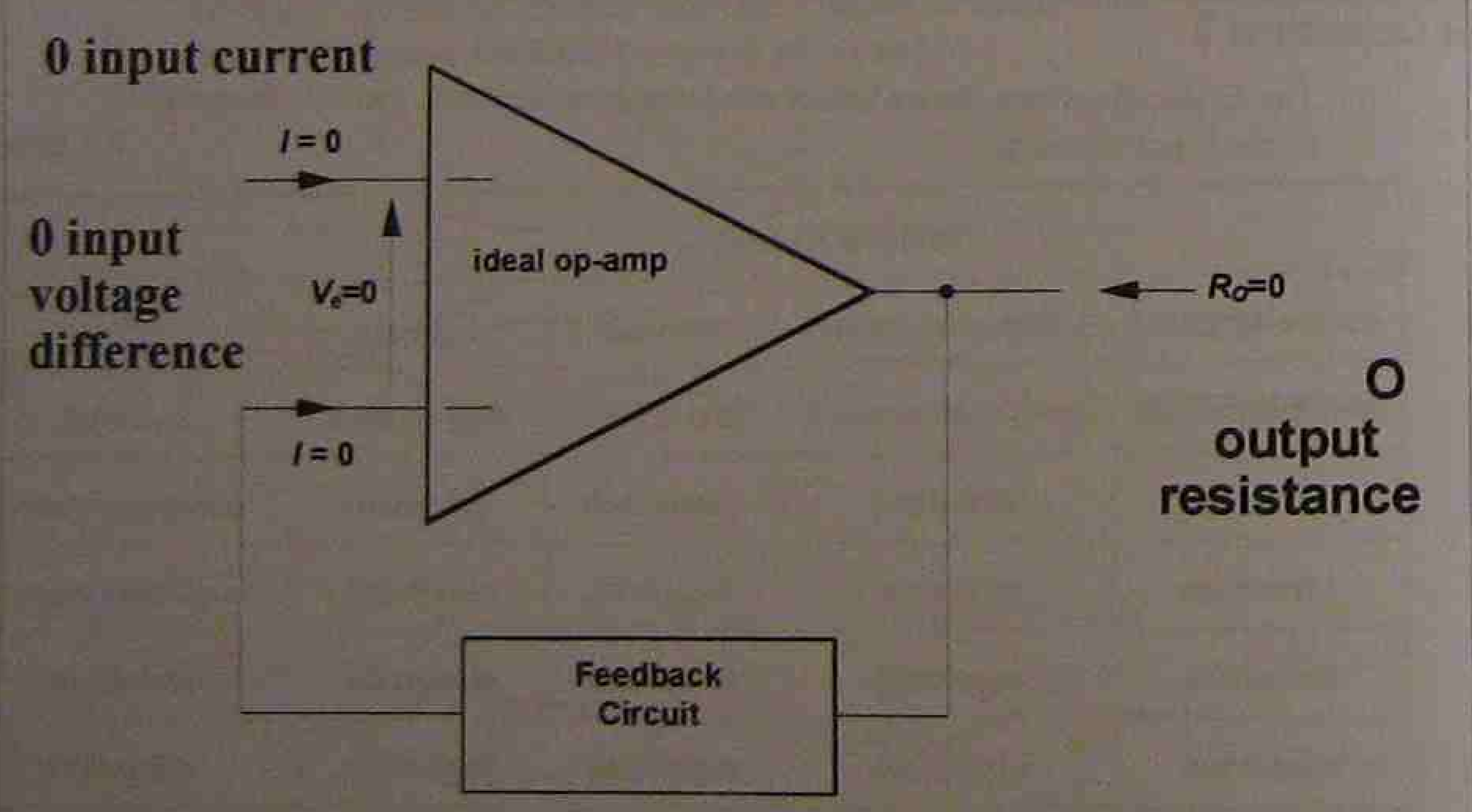
There is no current through the inverting input and the non-inverting input

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.

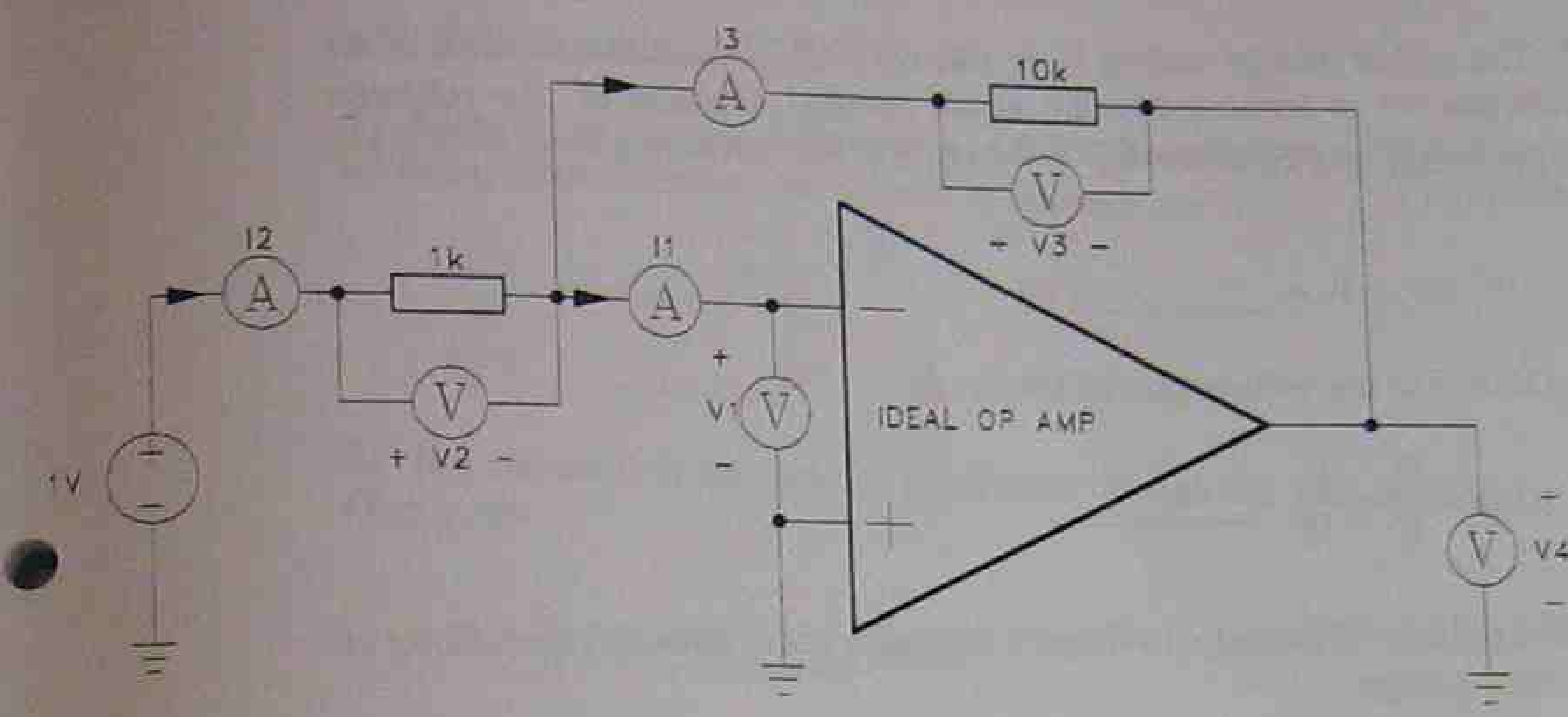
The output resistance of an ideal op-amp is 0Ω. Therefore, there is no loading effect between the op-amp and the feedback circuit.

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

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

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)

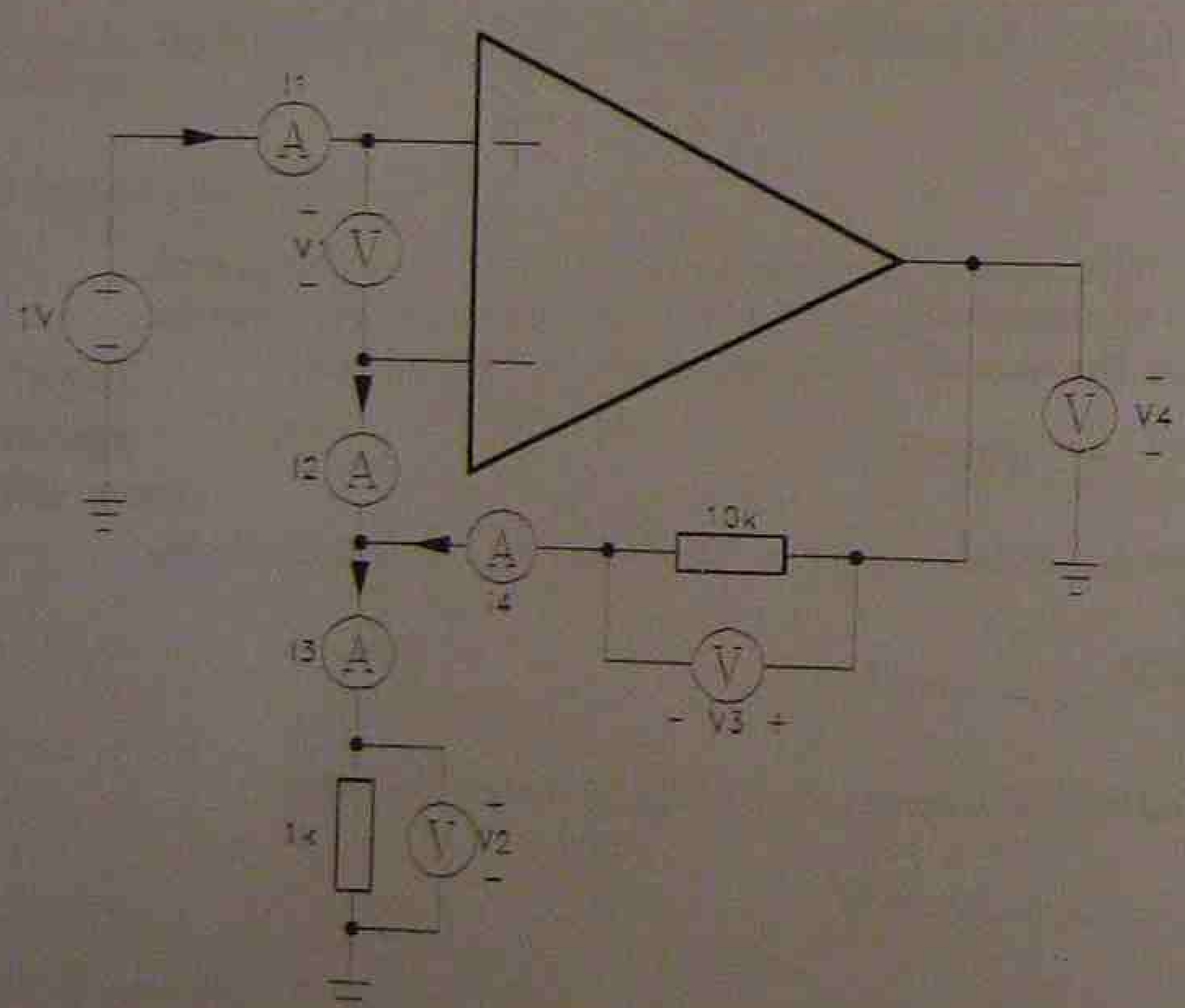
8. Determine the voltage gain of the circuit.

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

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

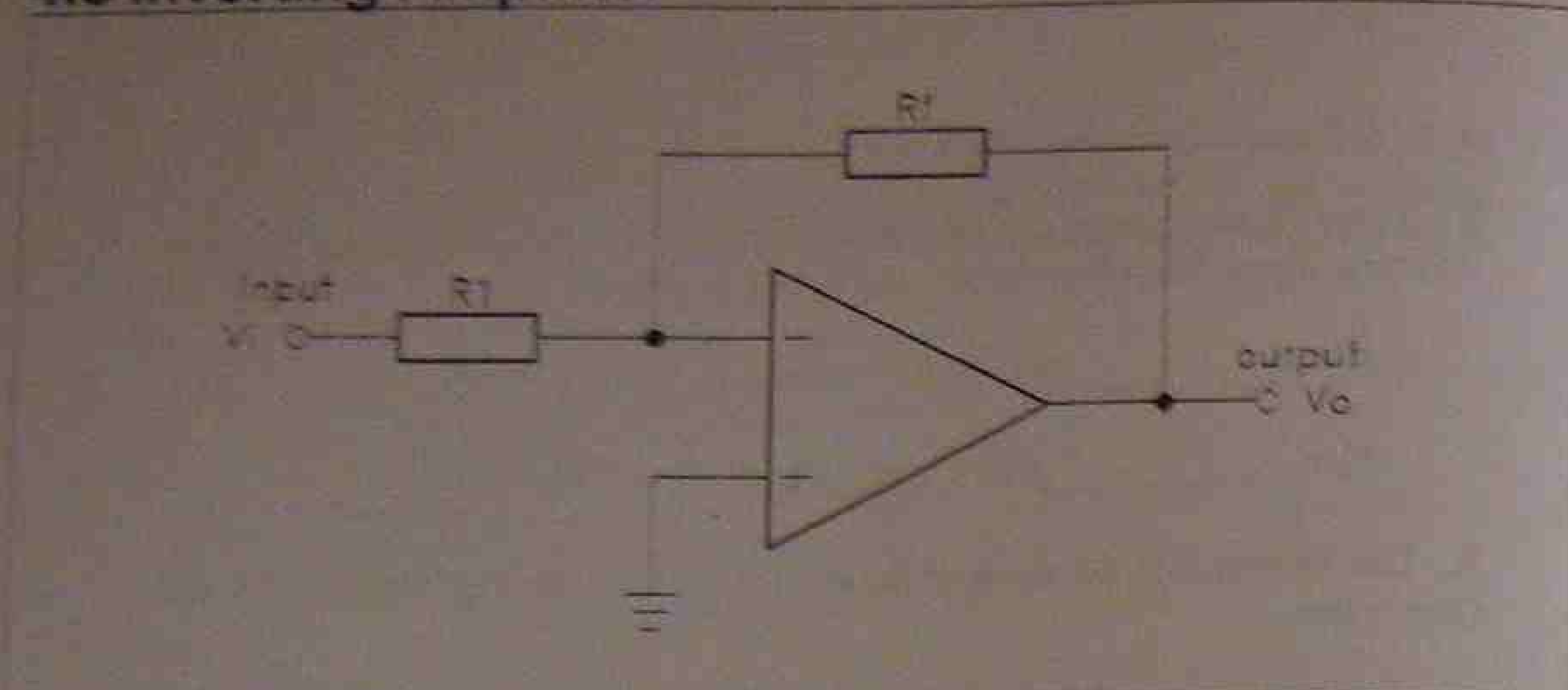


Figure 4.6 Inverting amplifier

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.

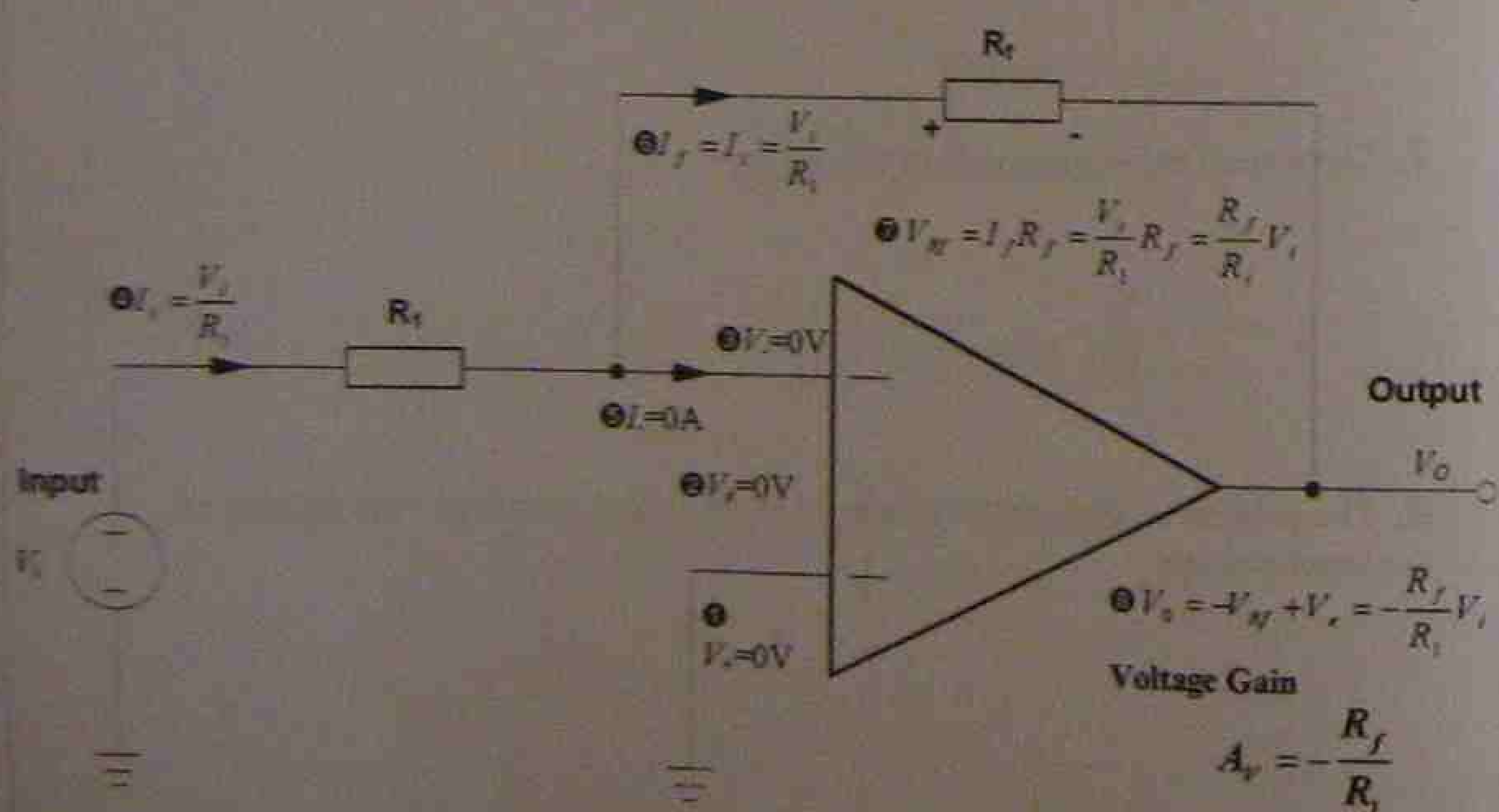


Figure 4.7 An analysis of an inverting amplifier

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

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

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

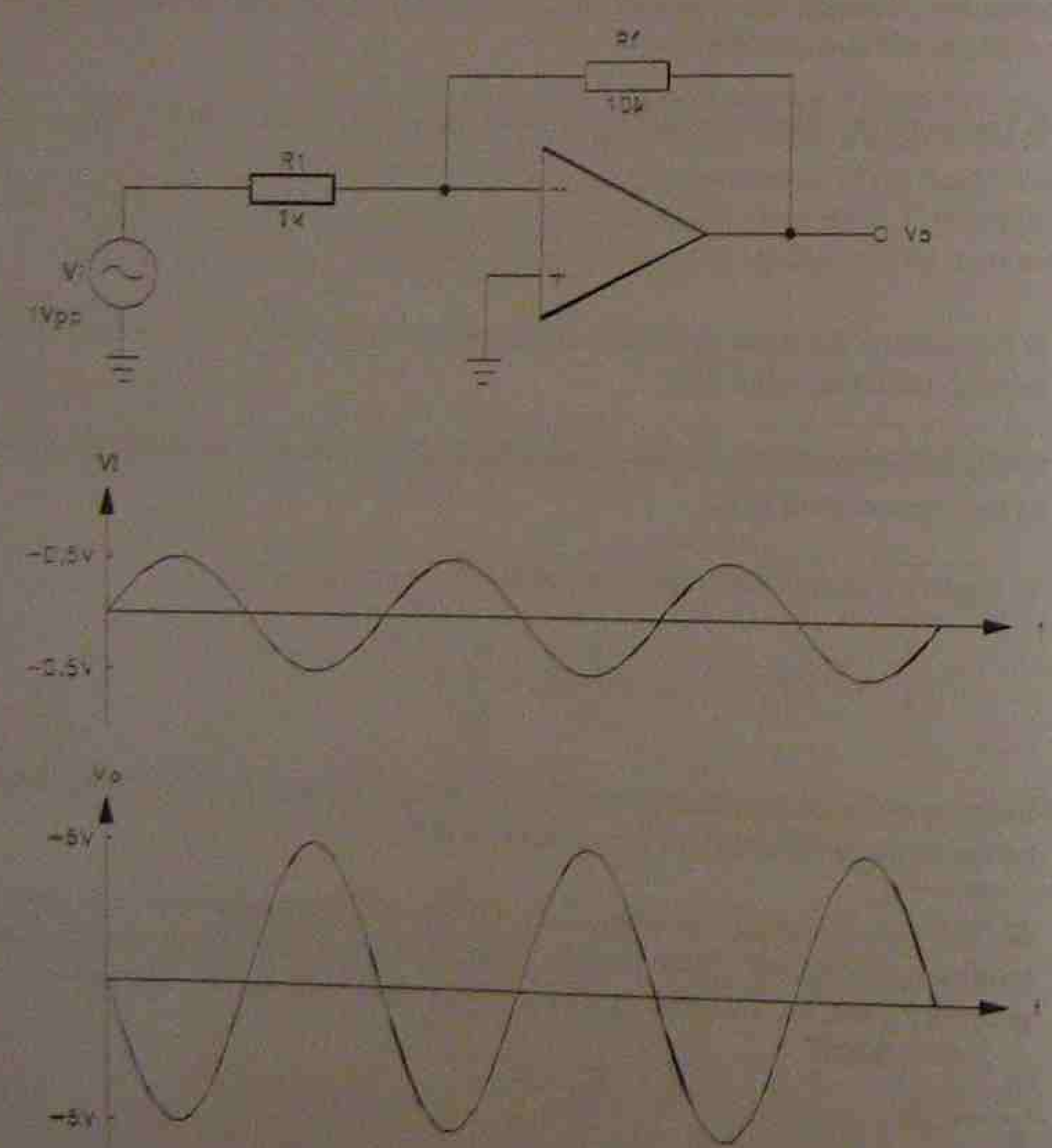
Now we can determine the voltage gain of the amplifier.

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

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:

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

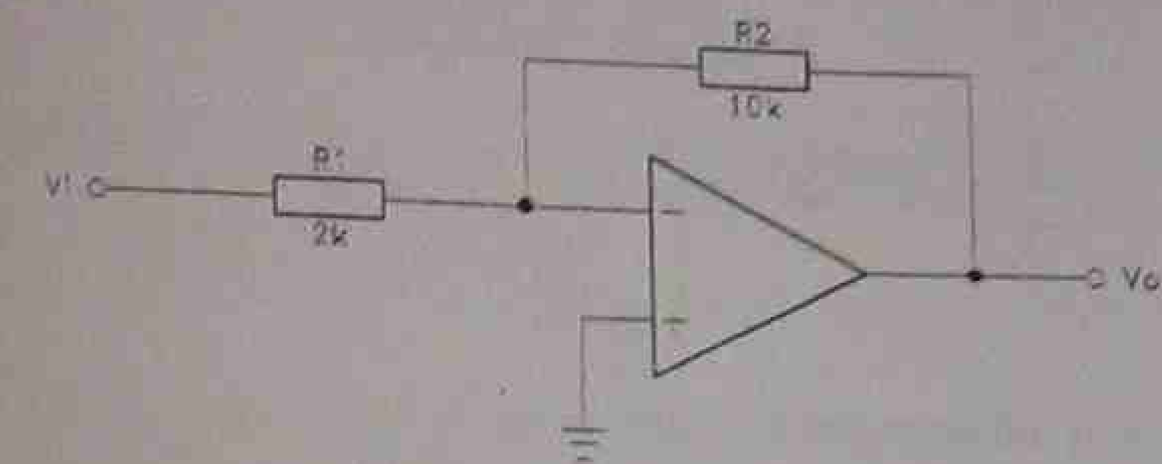
Figure 4.8 Reversed phase in an inverting amplifier



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.

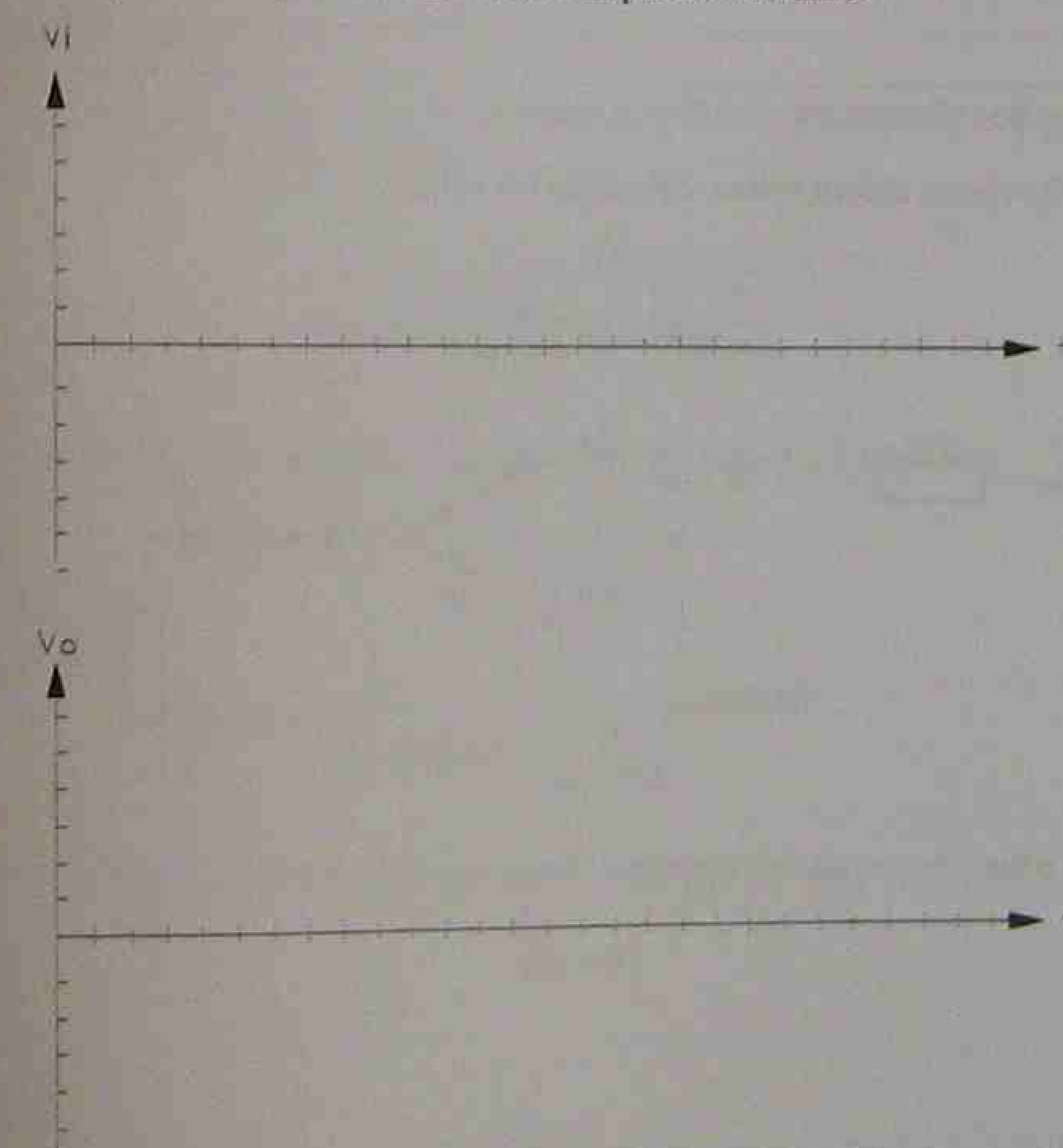
Drill Question 5

For an amplifier shown below:



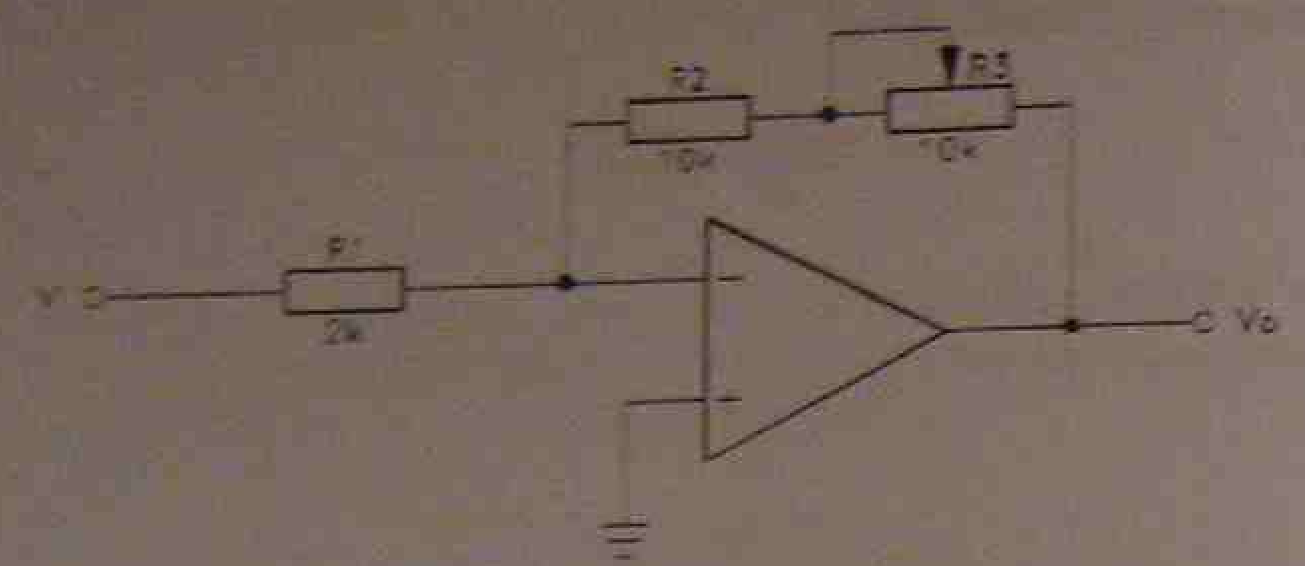
(a) Determine the voltage gain of the amplifier.

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



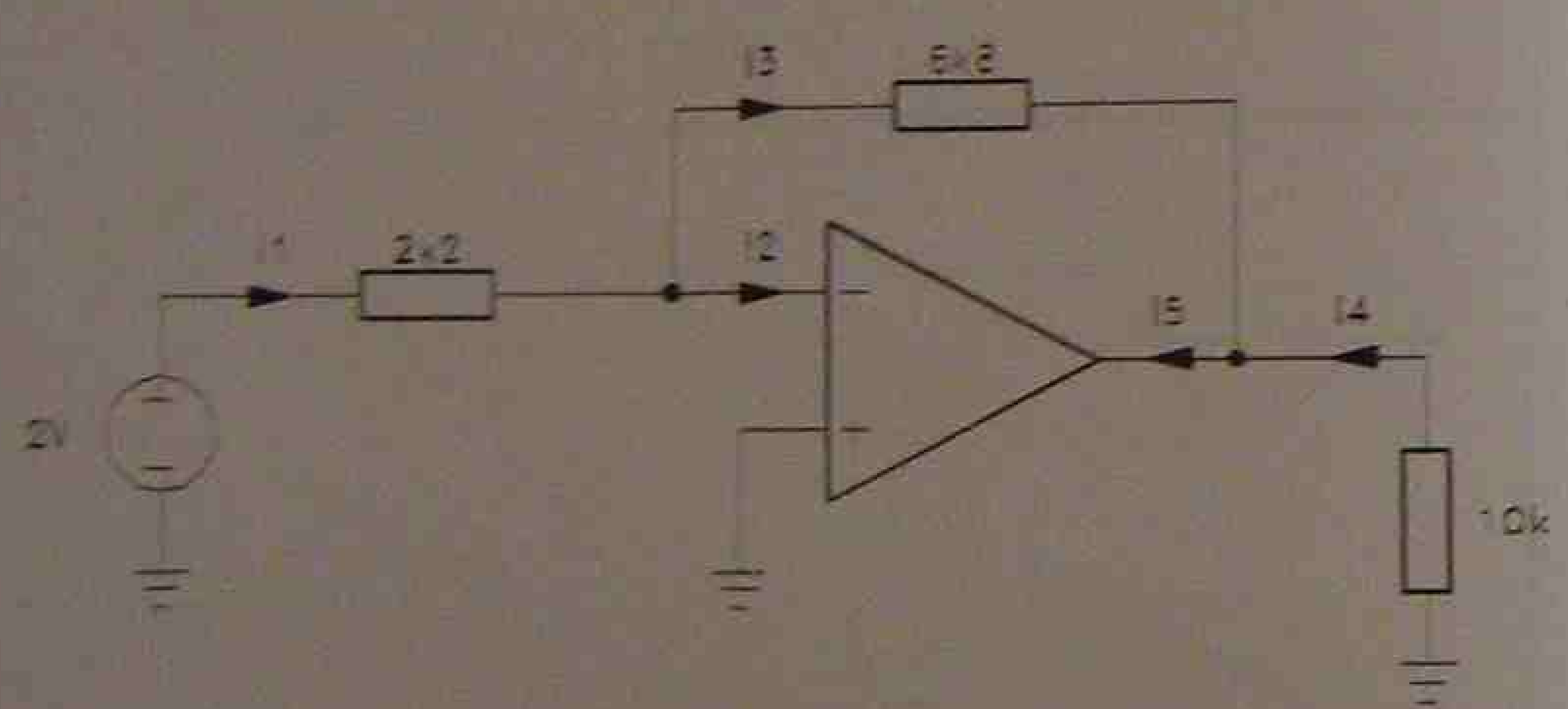
Drill Question 6

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

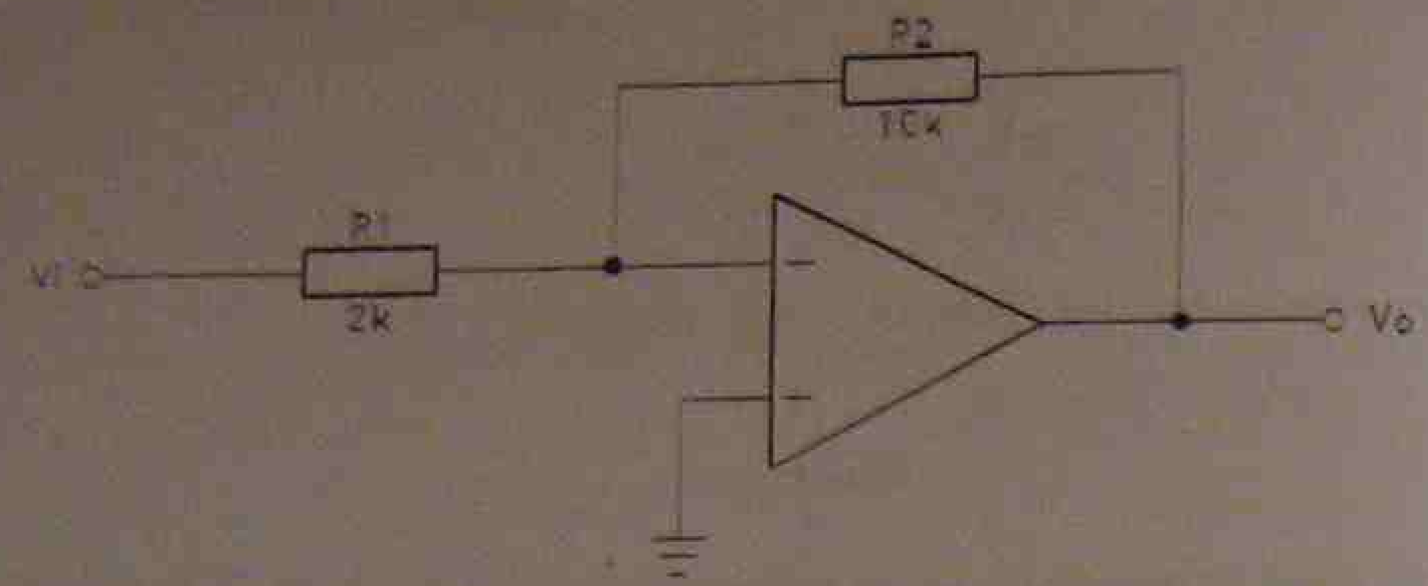
$$R_{O(closed)} = \frac{75}{1 + 200{,}000 \times \frac{1k}{1k + 10k}} \approx 0.004\Omega$$

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

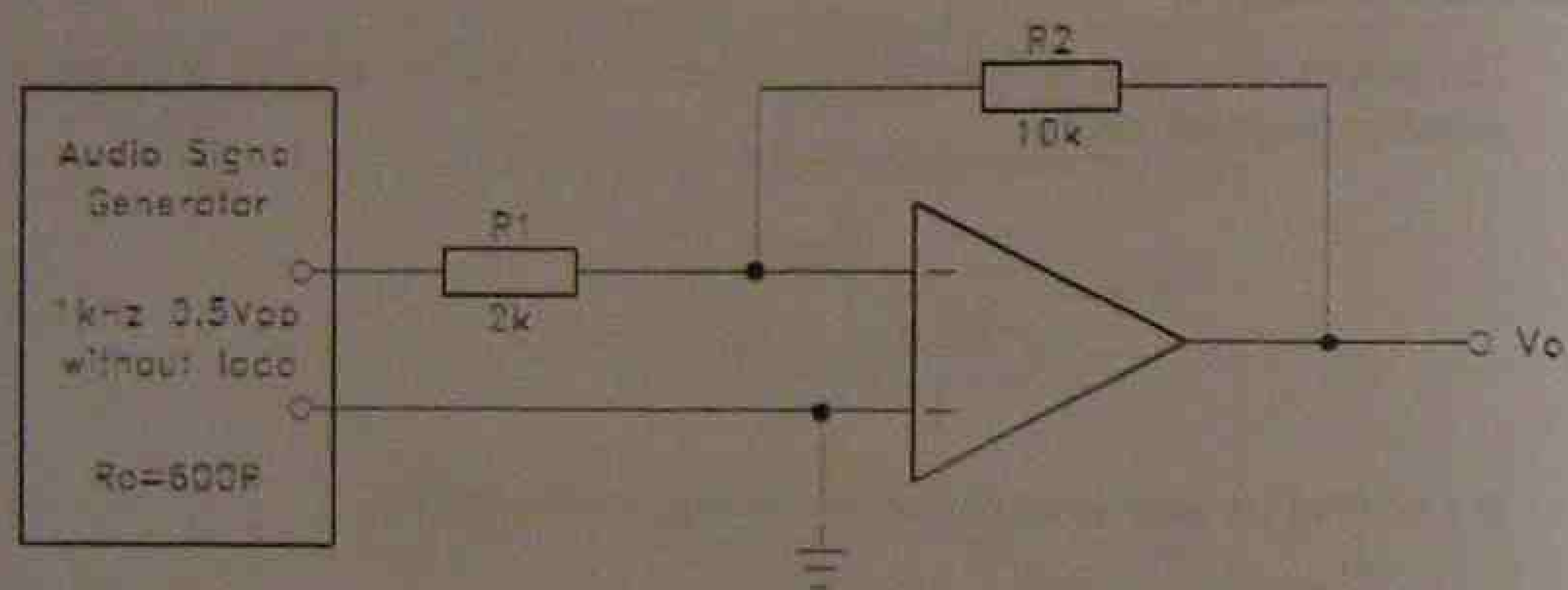
$$R_O = 0$$

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.

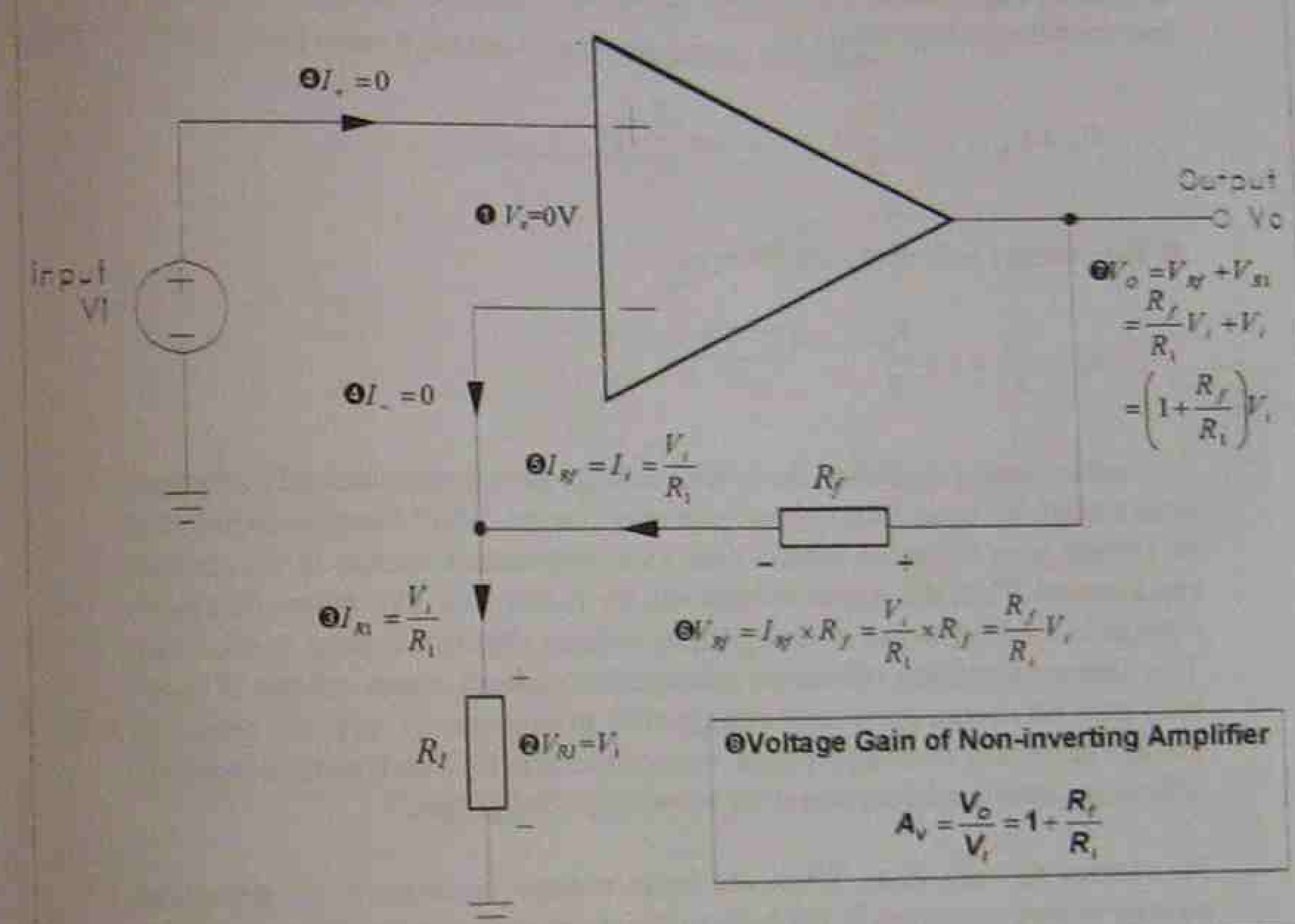


4.4 Non-inverting Amplifier

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_{Rf} 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.

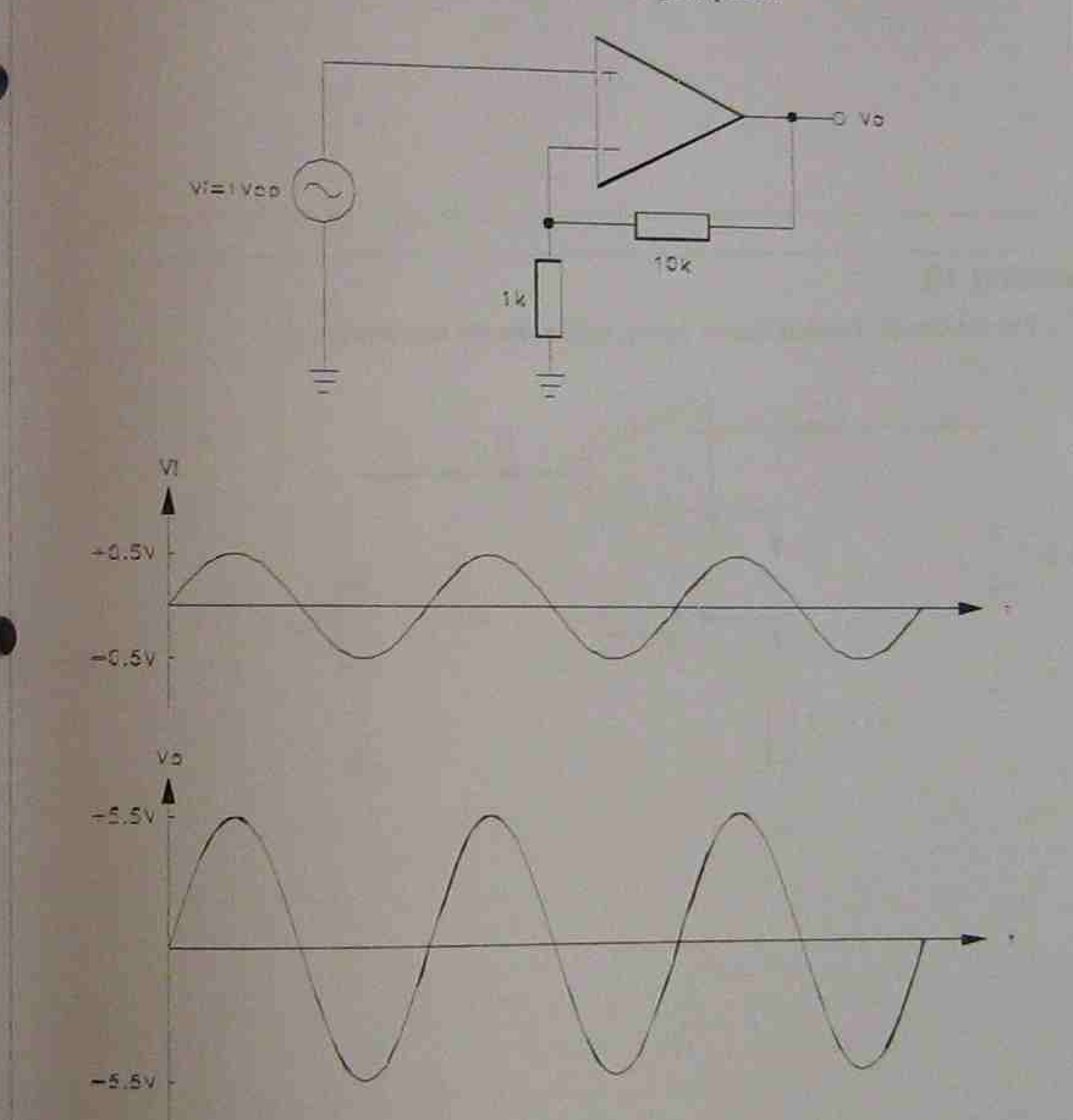
Input and Output Phase

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:

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

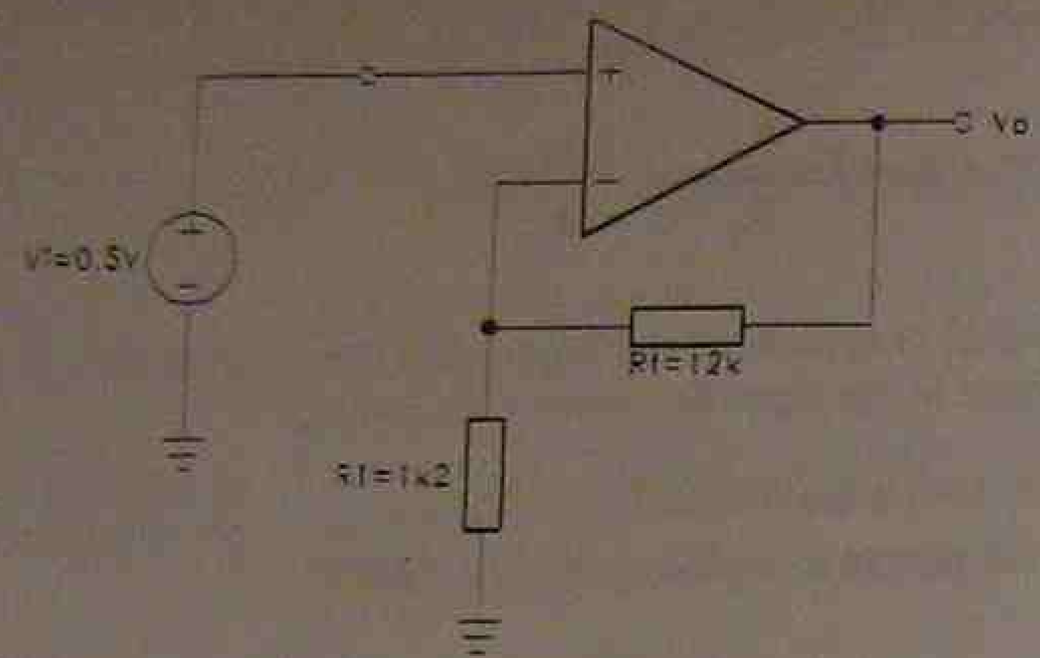
If a 1Vpp ac signal is applied to the input, the output signal will be 11Vpp in phase with respect to the input as shown in Figure 4.8.

Figure 4.10 Input and output phase in non-inverting amplifier



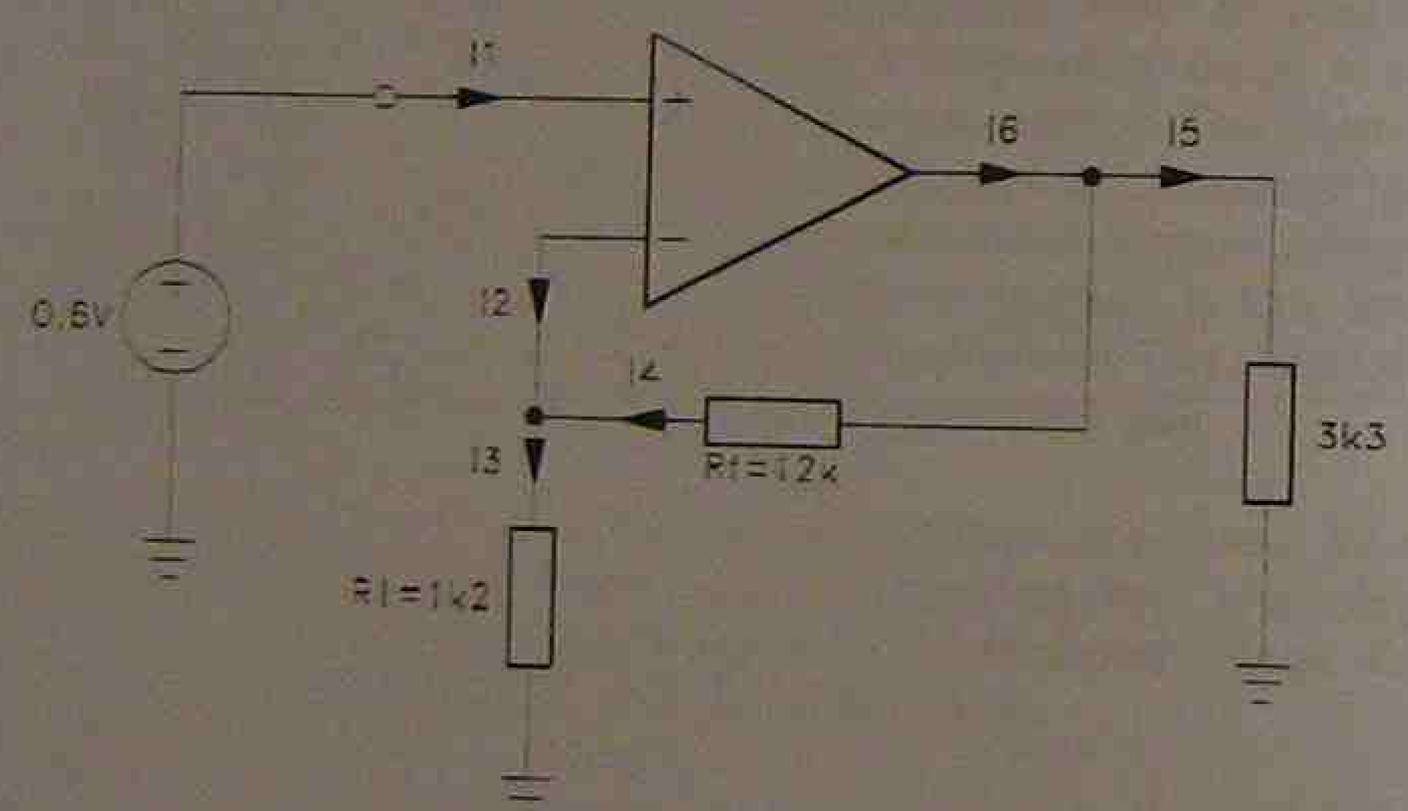
Drill Question 9

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



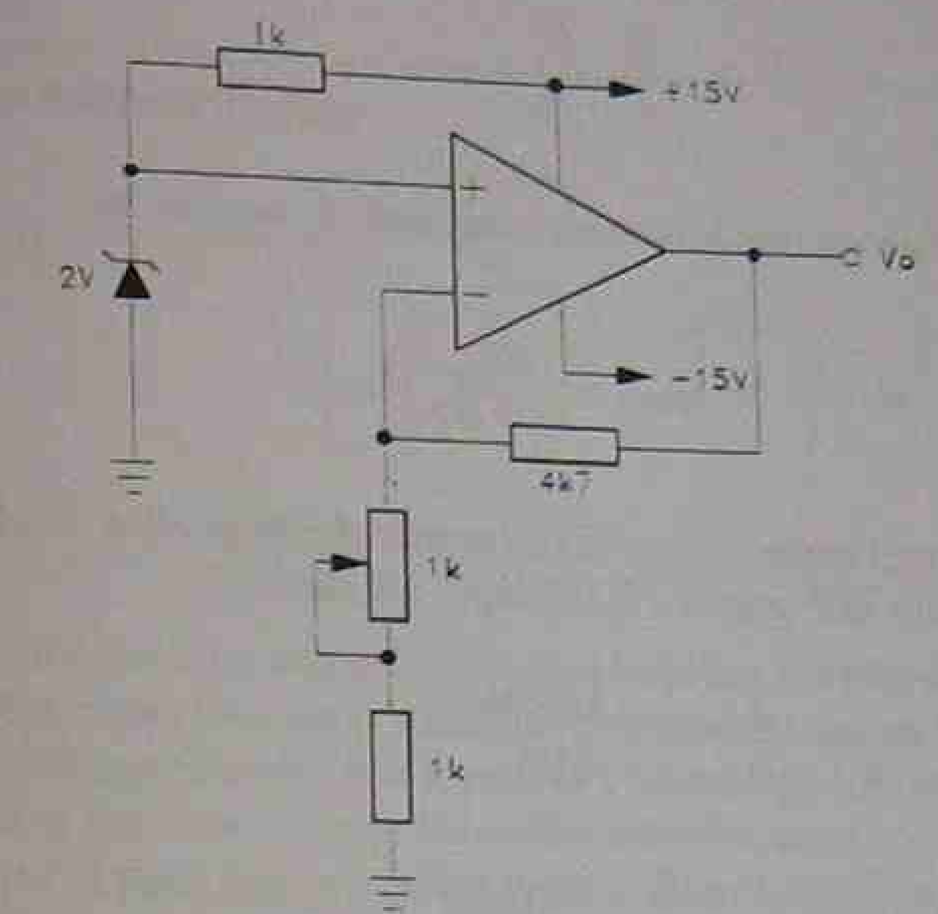
Drill Question 10

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

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

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

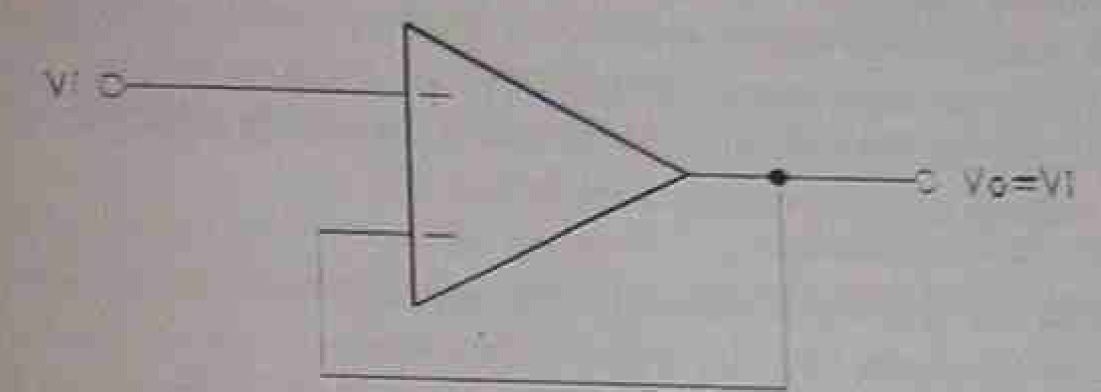
Table 4.2 Characteristic comparison of inverting amplifier and non-inverting amplifier

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

4.5 Voltage Follower (Buffer)

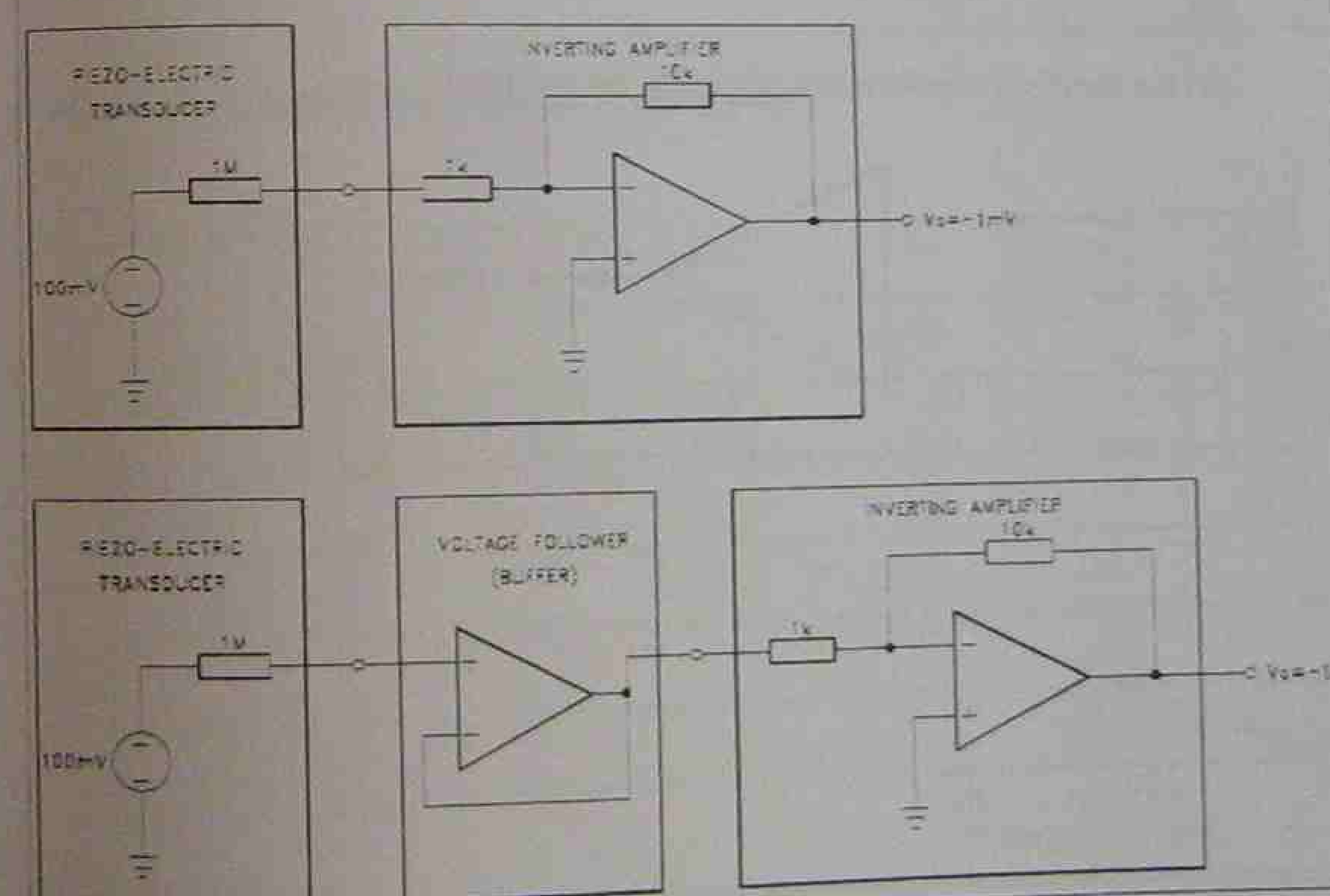
The voltage follower is a non-inverting amplifier whose voltage gain is 1. Figure 4.11 shows a voltage follower using an op amp.

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

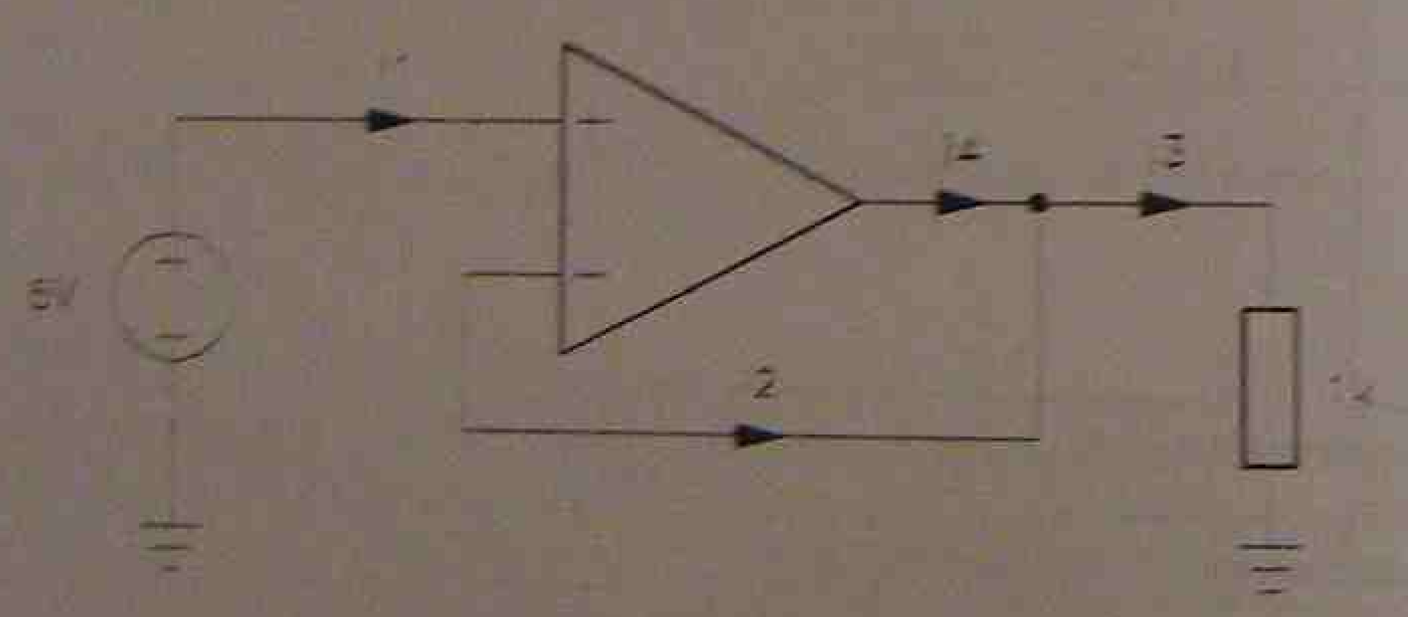
(a) Describe the characteristics of a voltage follower in terms of:

(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:

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

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

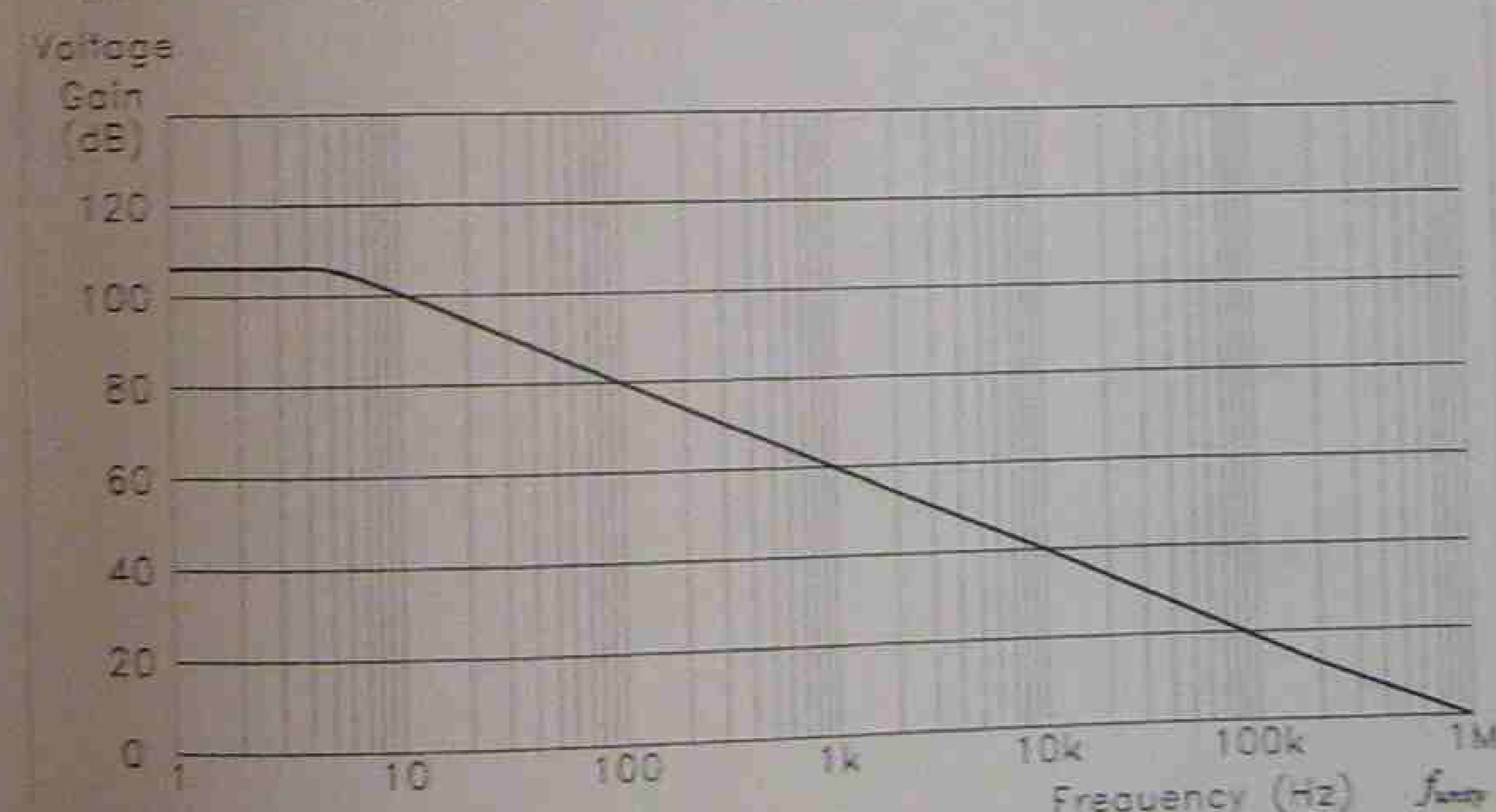
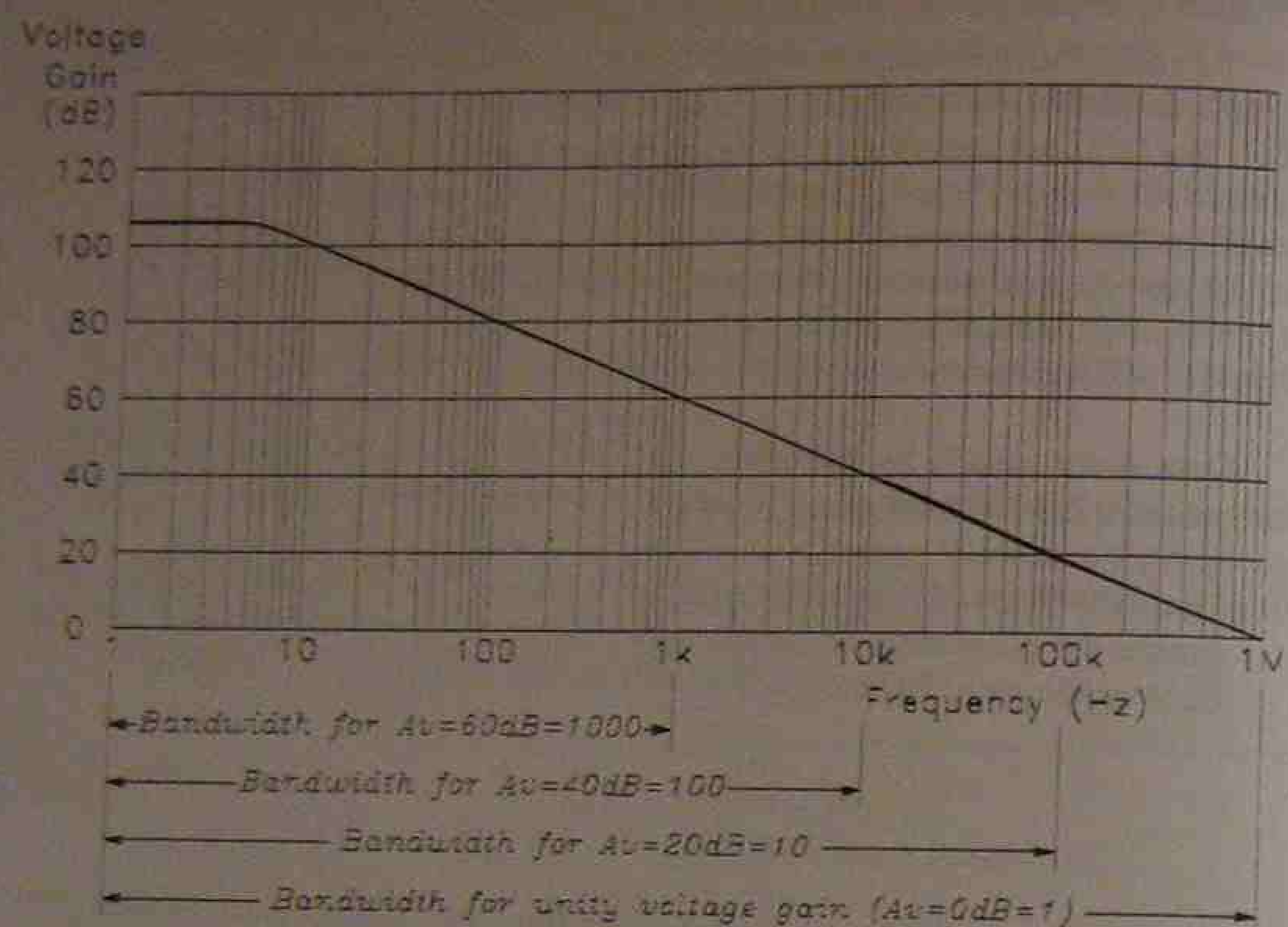


Figure 4.14 Bandwidth increases when closed-loop voltage gain decreases.



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) $A_V = 1000$

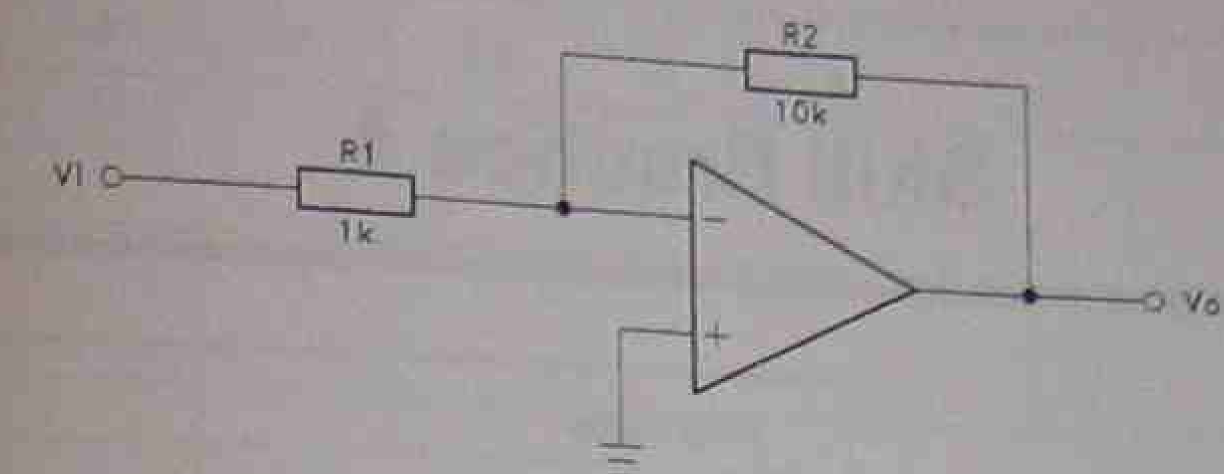
(ii) $A_V = 50$

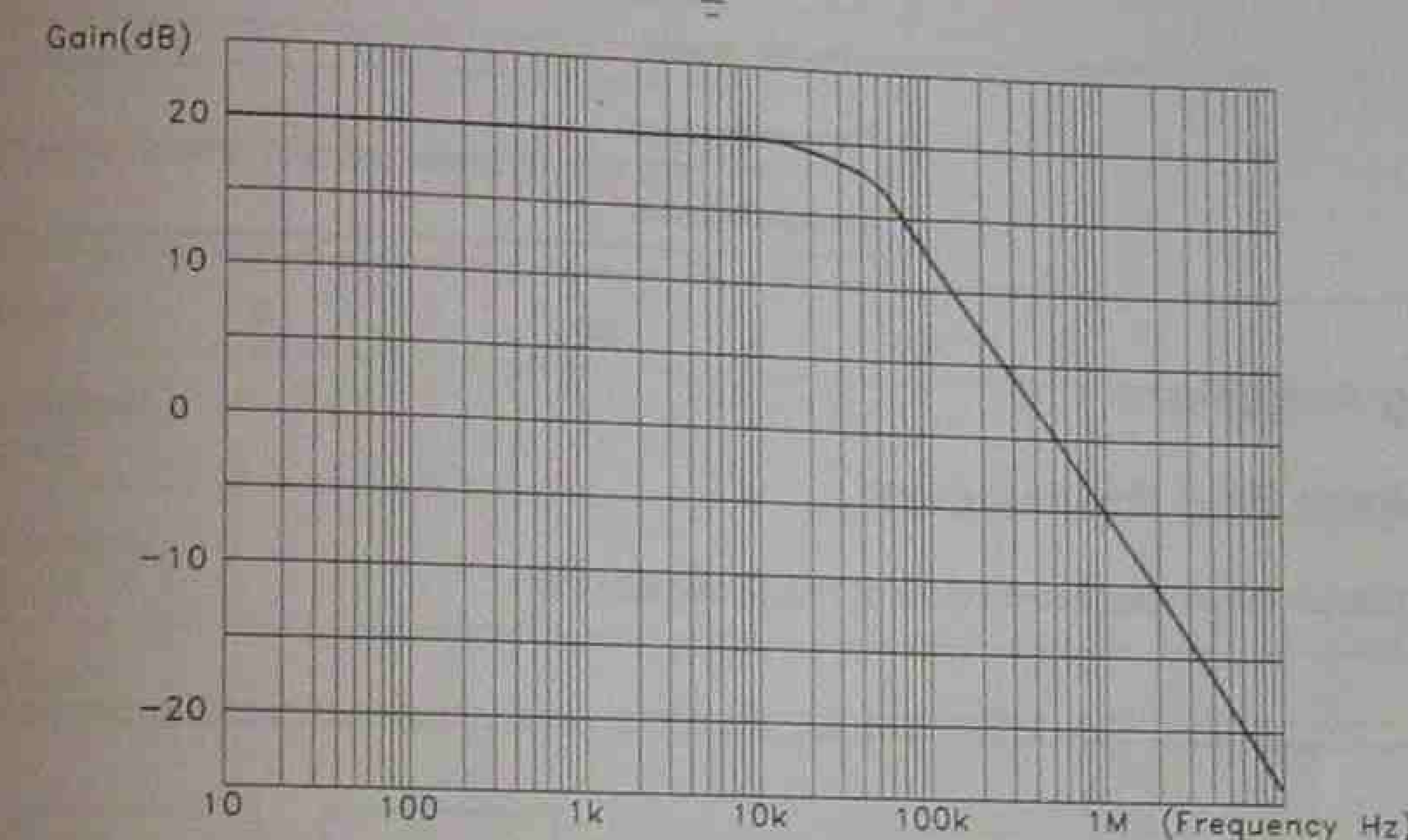
(iii) $A_V = 1$

(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

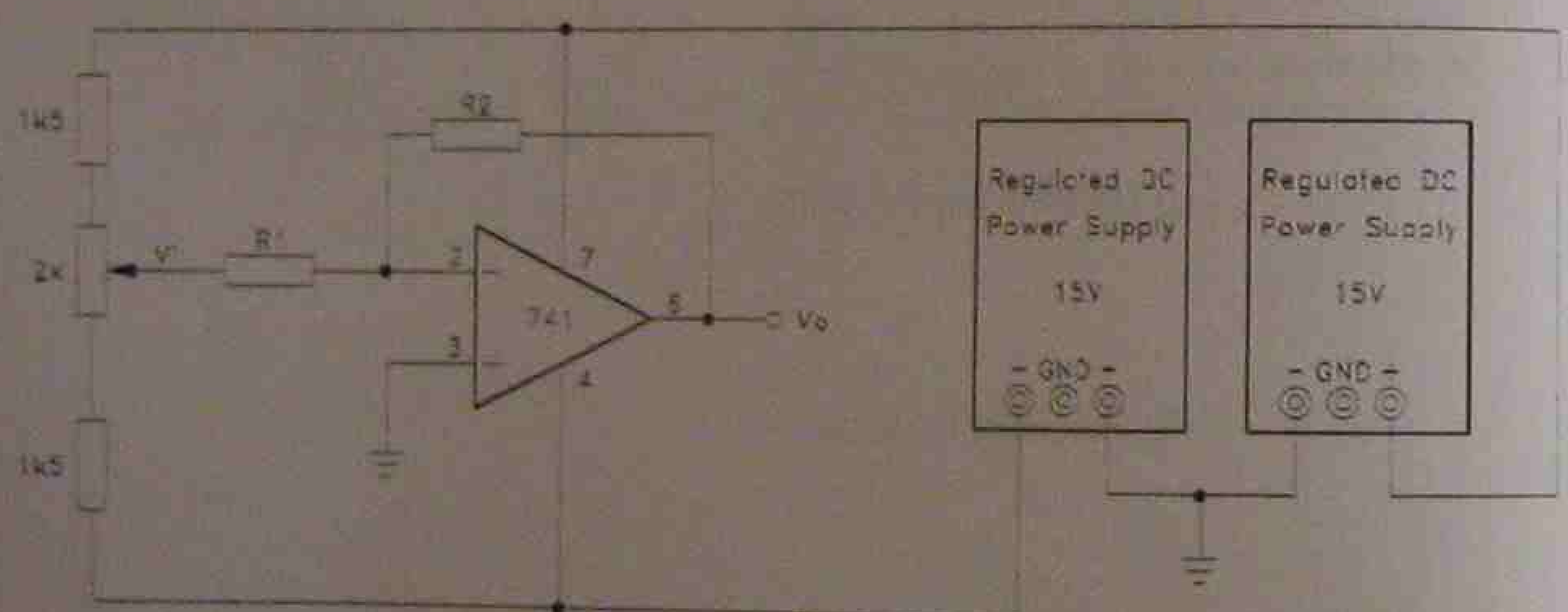
Skill Practice 4

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.

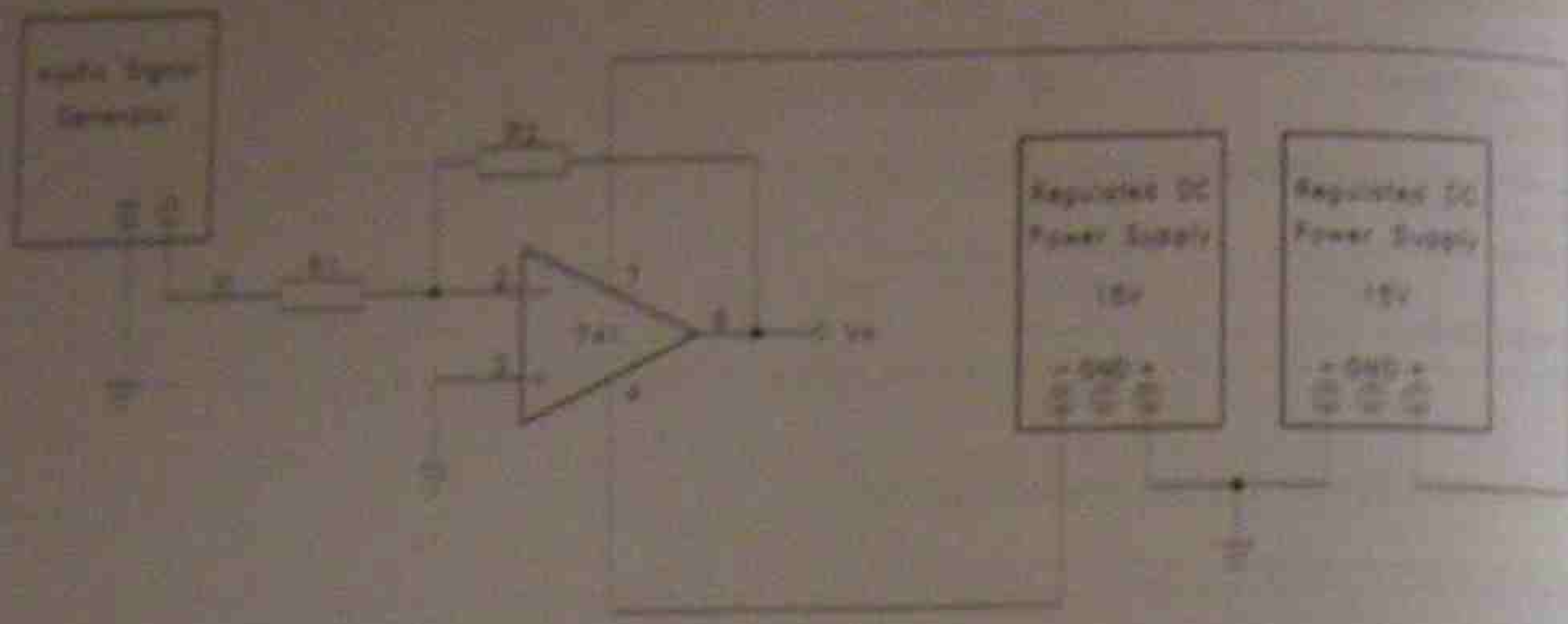
(iii) Negative saturation output voltage.

(iv) DC output offset voltages.

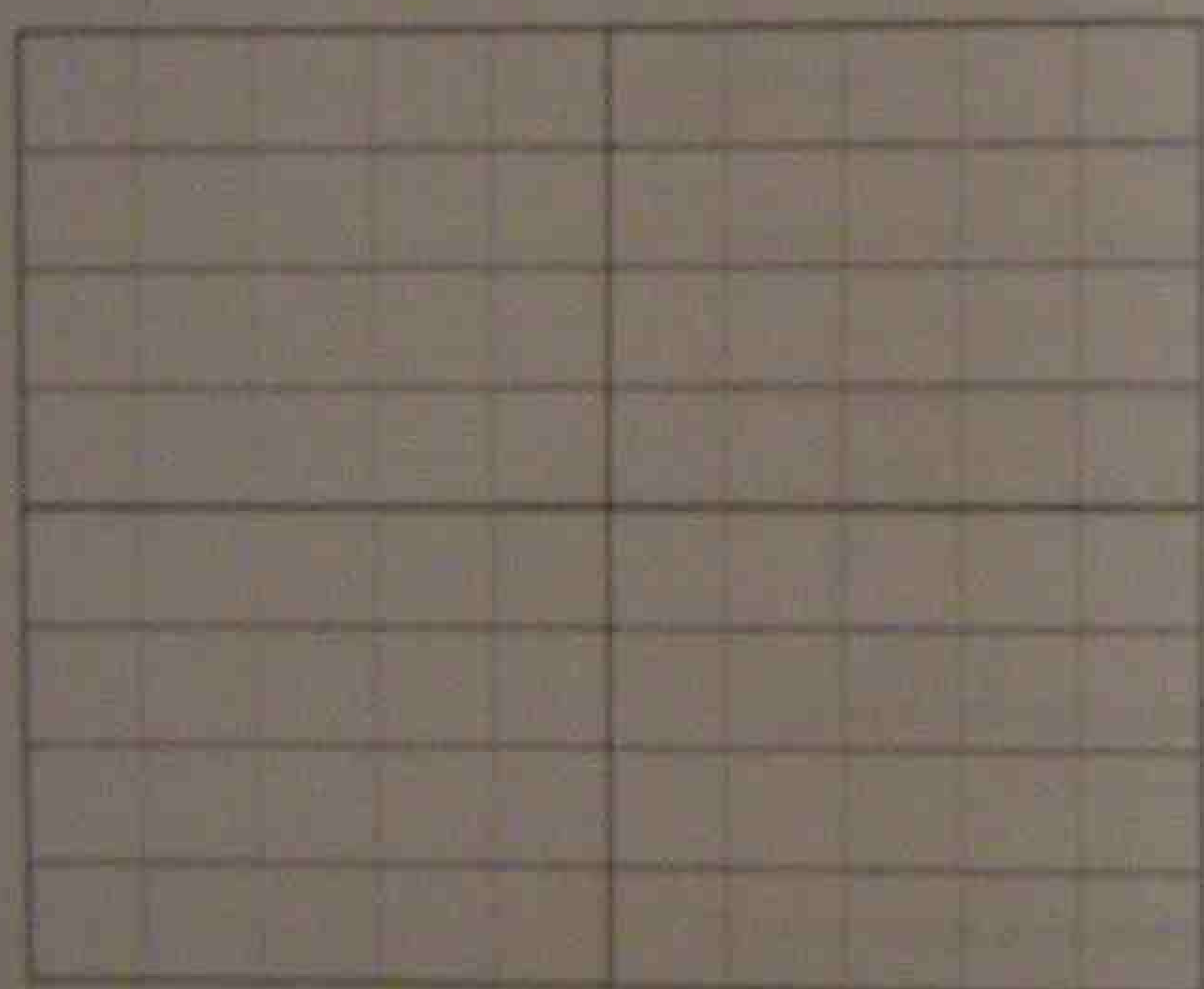
(f) Explain why it is an inverting amplifier.

AC Voltage Gain and Waveform Observations

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

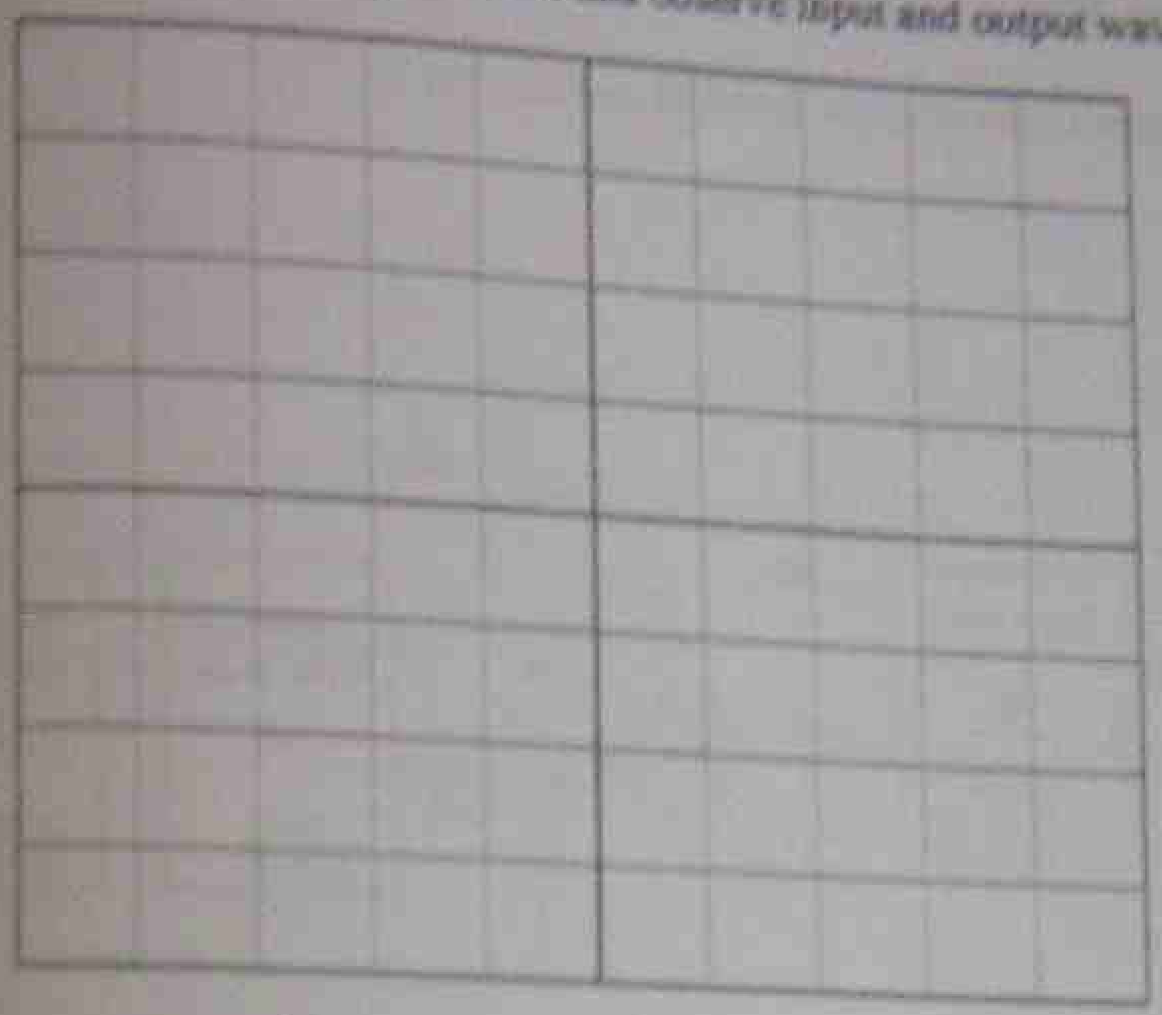


(b) Set Vi for 1kHz 1Vpp sine wave and observe both input and output voltage waveforms.
Determine the voltage gain of the amplifier form the observation.



Confirm the inverting amplification from the observation.

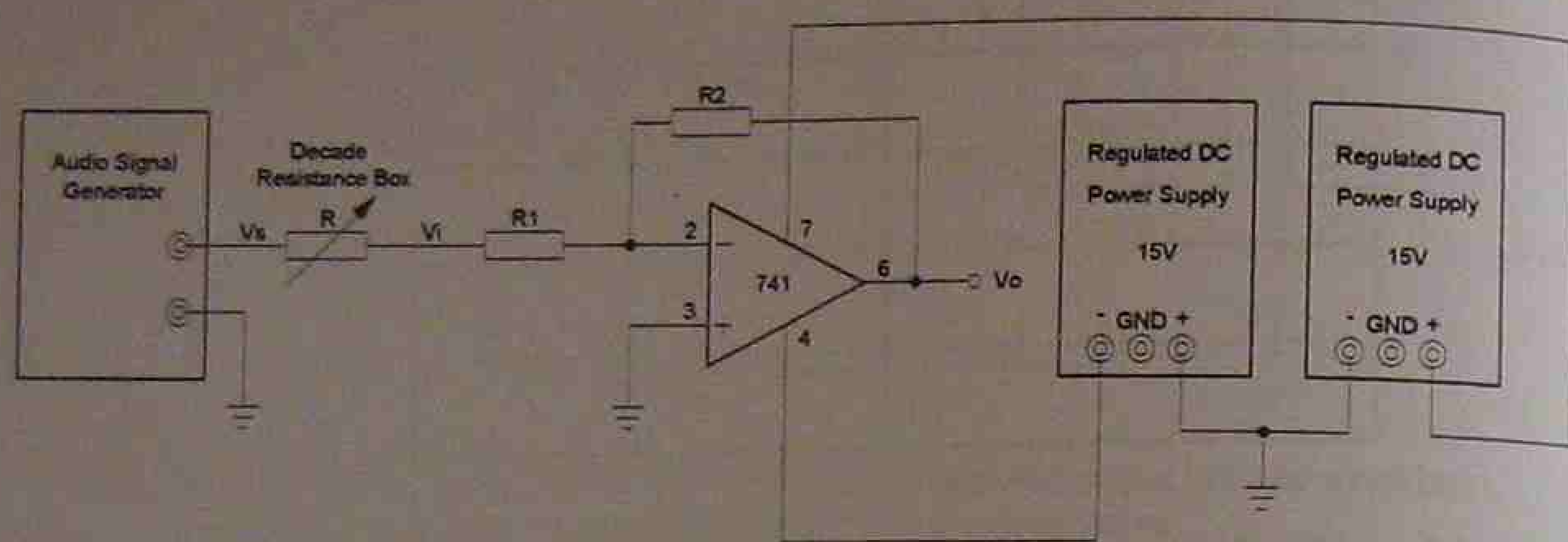
(c) Set input Vi for 1kHz 1Vpp square wave and observe input and output waveforms.



(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.

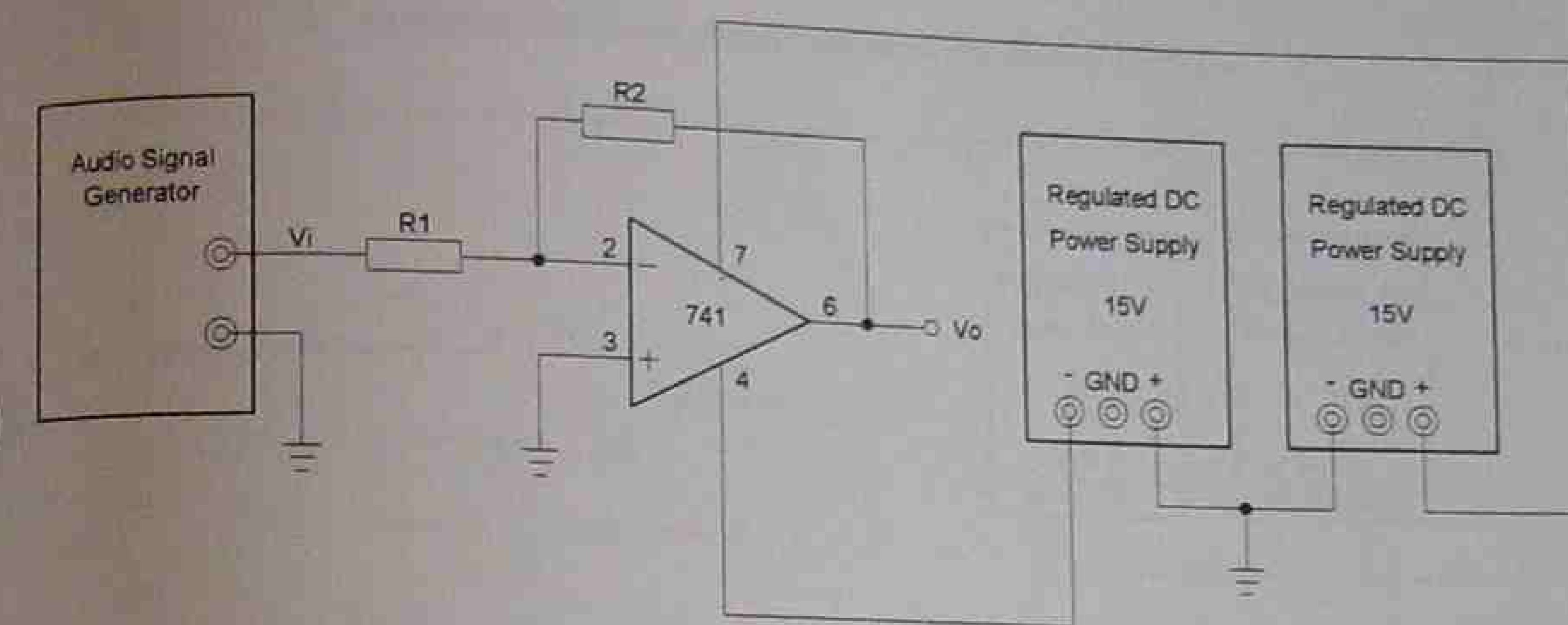
(d) Read the value of decade resistance box.

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.

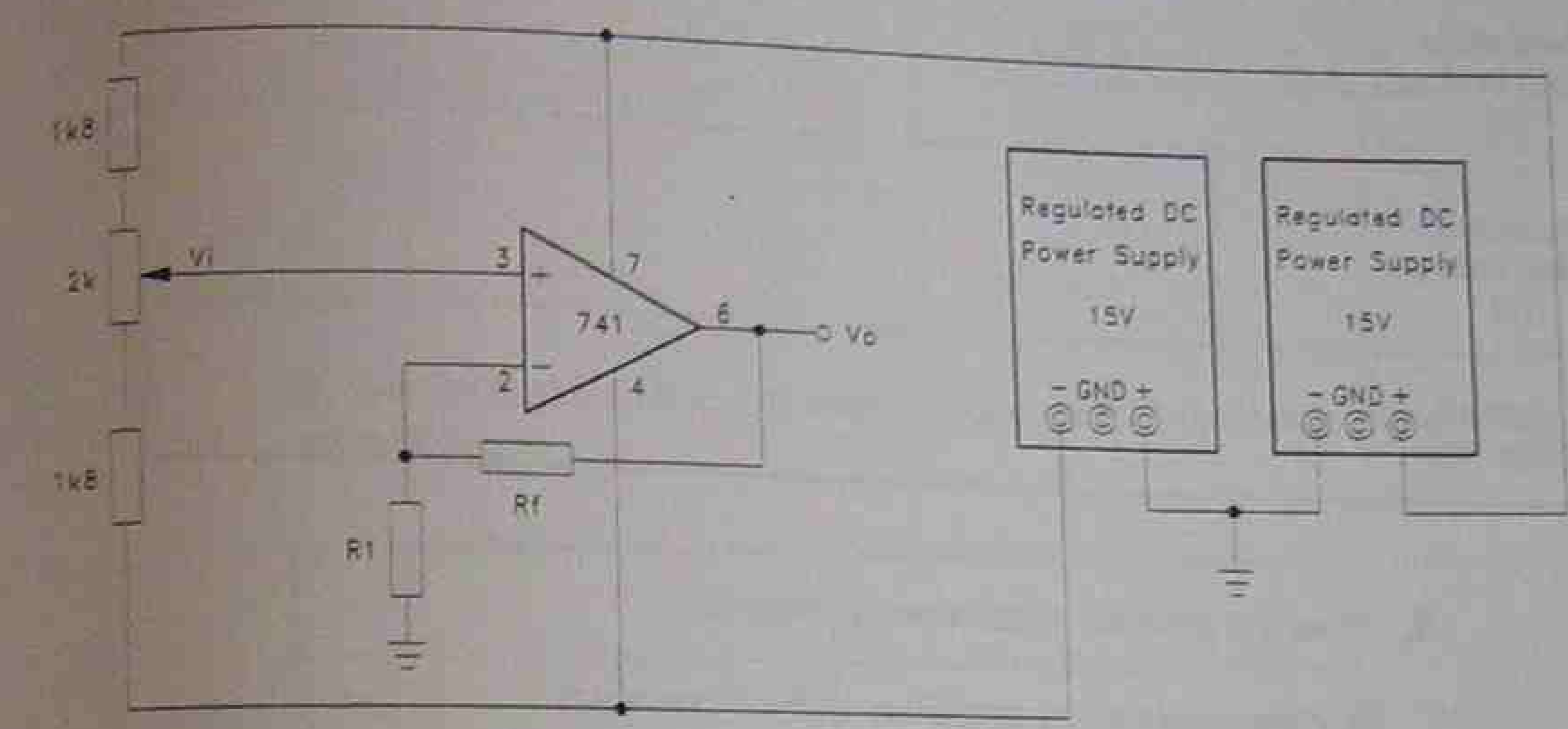
$$R_o = \frac{V_o - V_L}{V_L} \times R_L$$

(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.

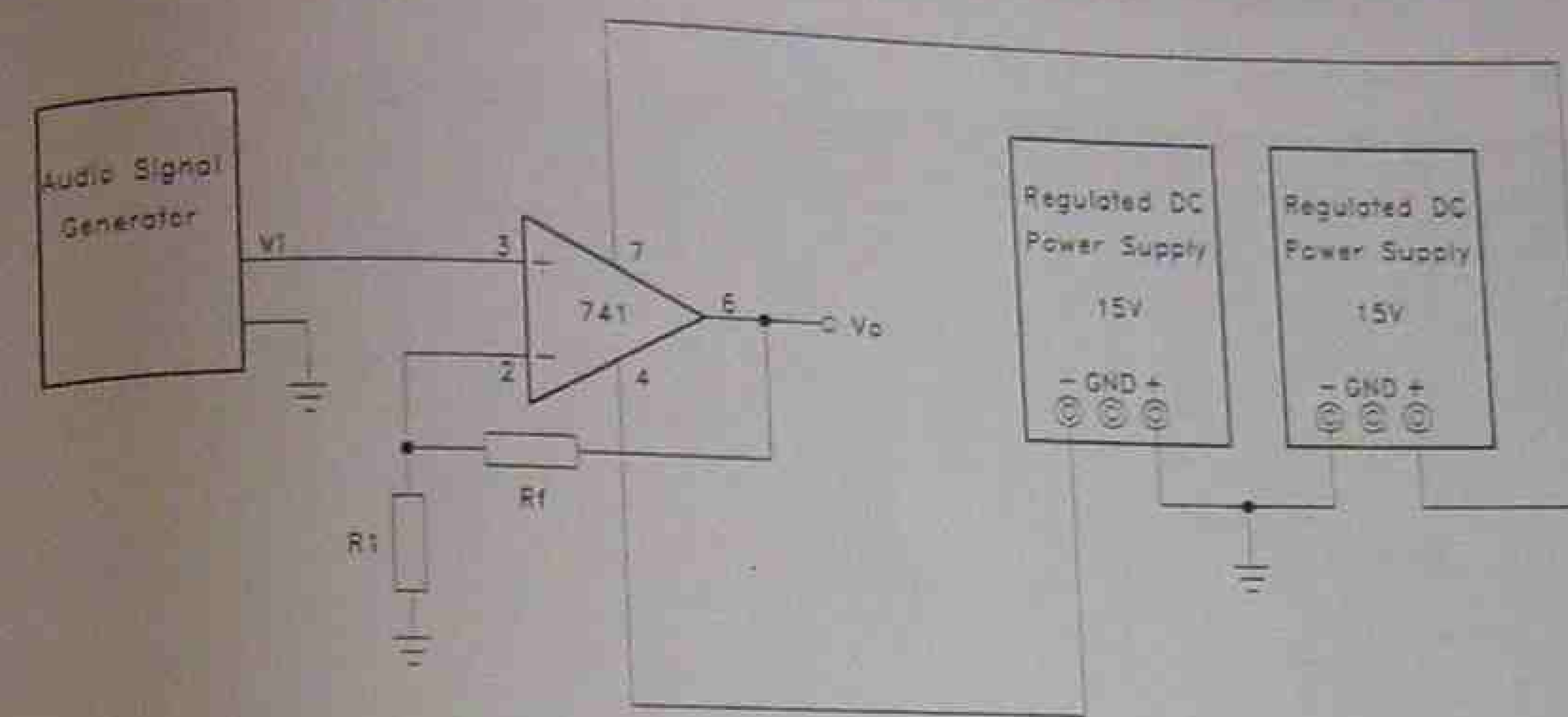
(ii) Positive saturation output voltage.

(iii) Negative saturation output voltage.

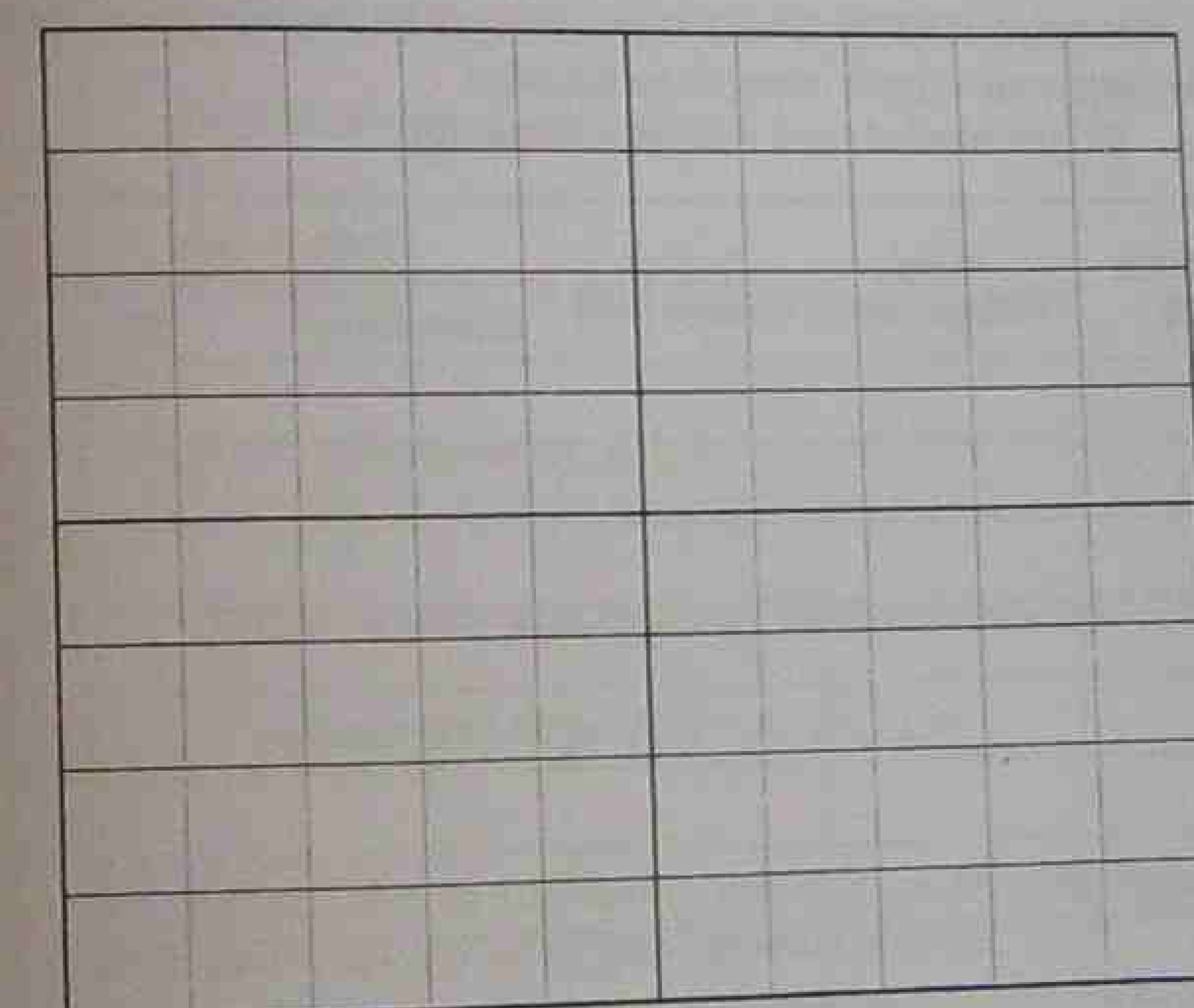
(iv) DC output offset voltage.

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.

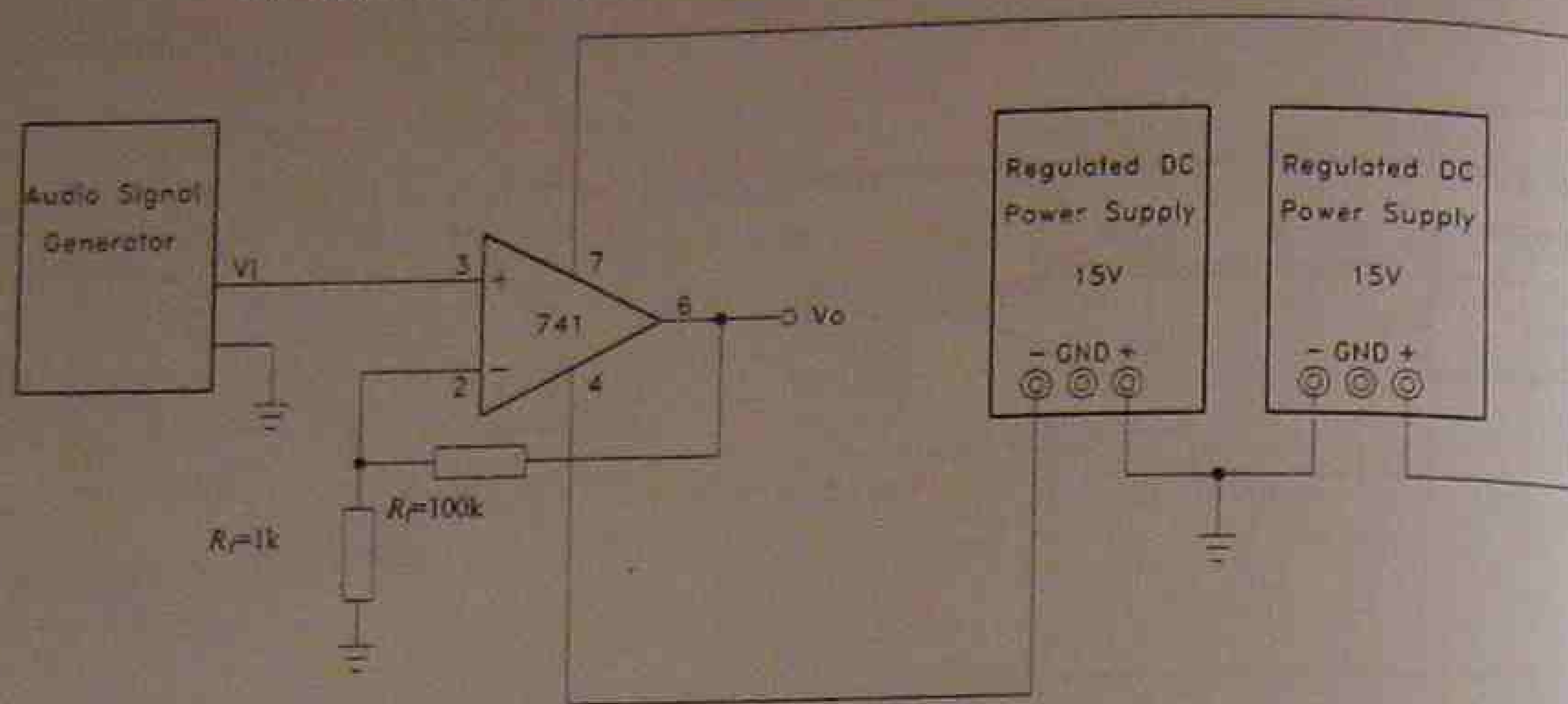


Determine the voltage gain (both in ratio and dB) of the amplifier from the observation.

Confirm the non-inverting amplification from the observation.

Frequency Response and Bandwidth Measurement

1. Construct the circuit shown below.

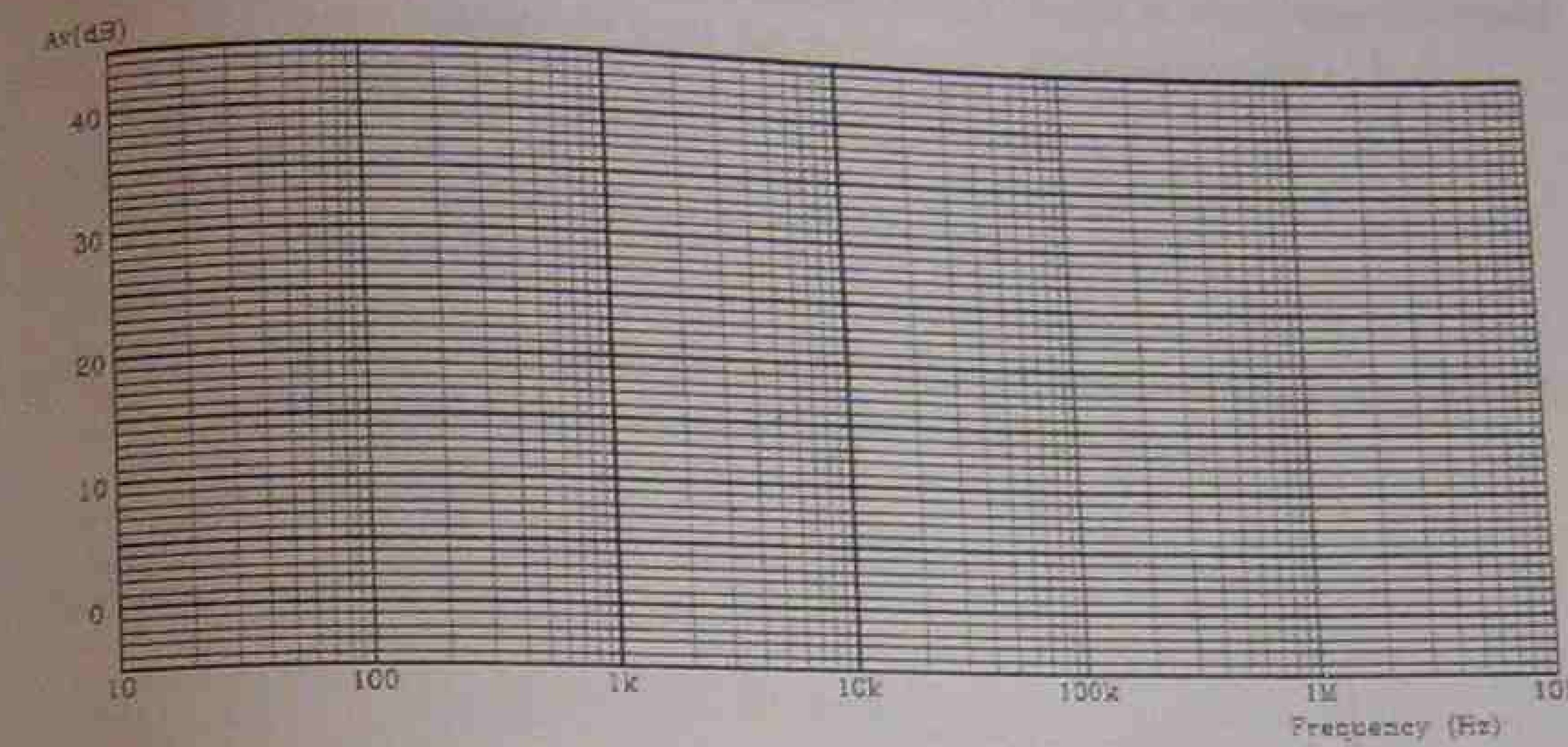


2. Determine the theoretical voltage gain of the amplifier.

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.

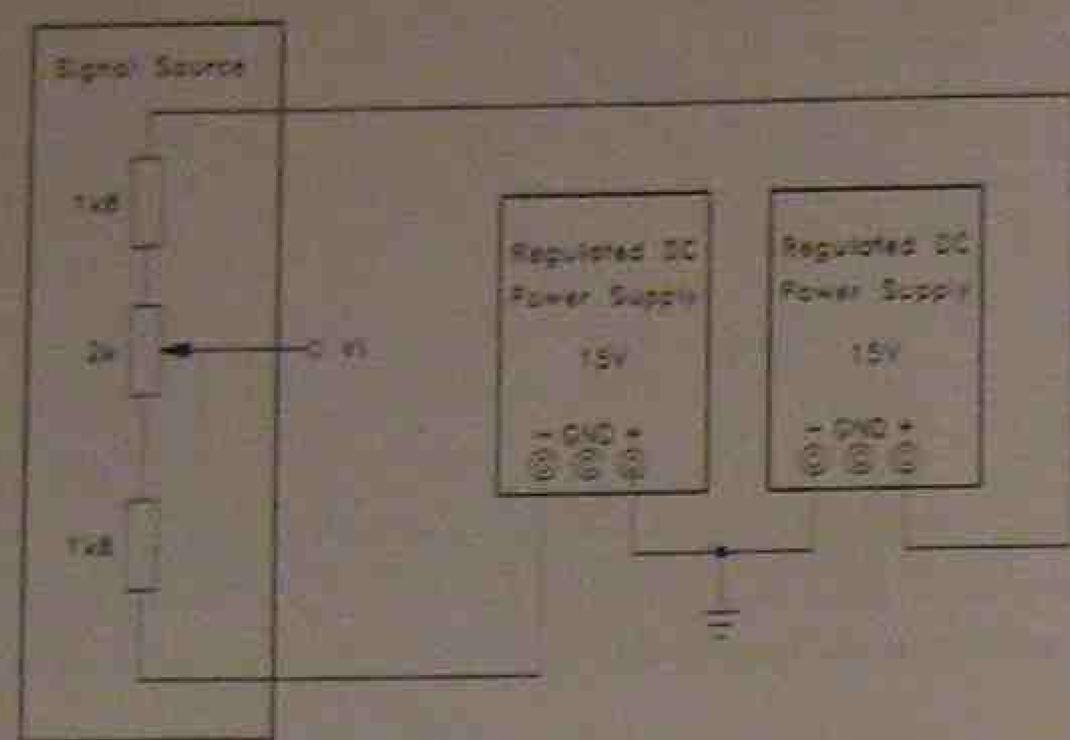
Gain-bandwidth product = $A_V \times f_{-3dB}$ =

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

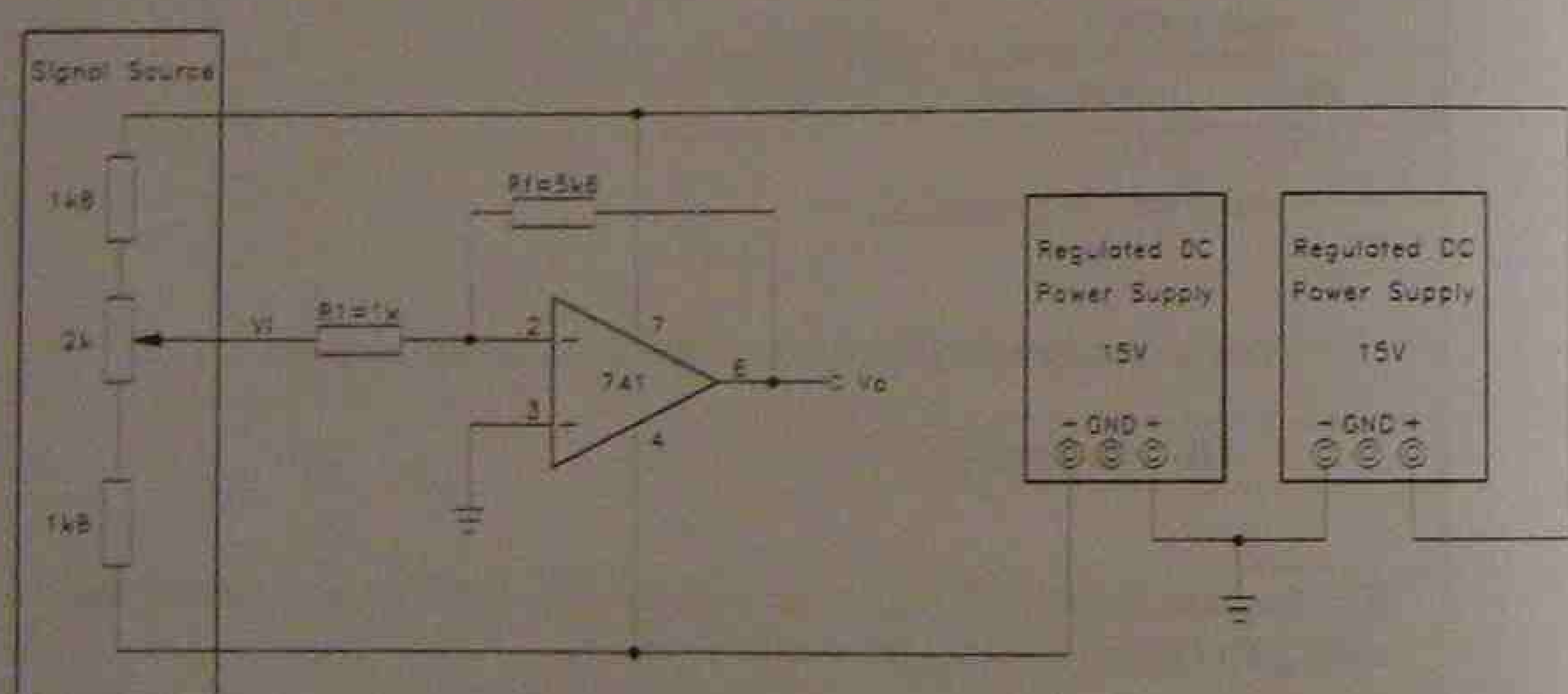
> The output voltage at -3dB frequency is :
>
> $$V_{O(-3dB)} = \frac{V_O}{\sqrt{2}} \approx 0.7V_O$$
>
> where the V_o is the output voltage for mid-band frequencies which is measured in step **3**. For example, if you measured V_o = 1.1Vpp, you just need to adjust output frequency of the audio signal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



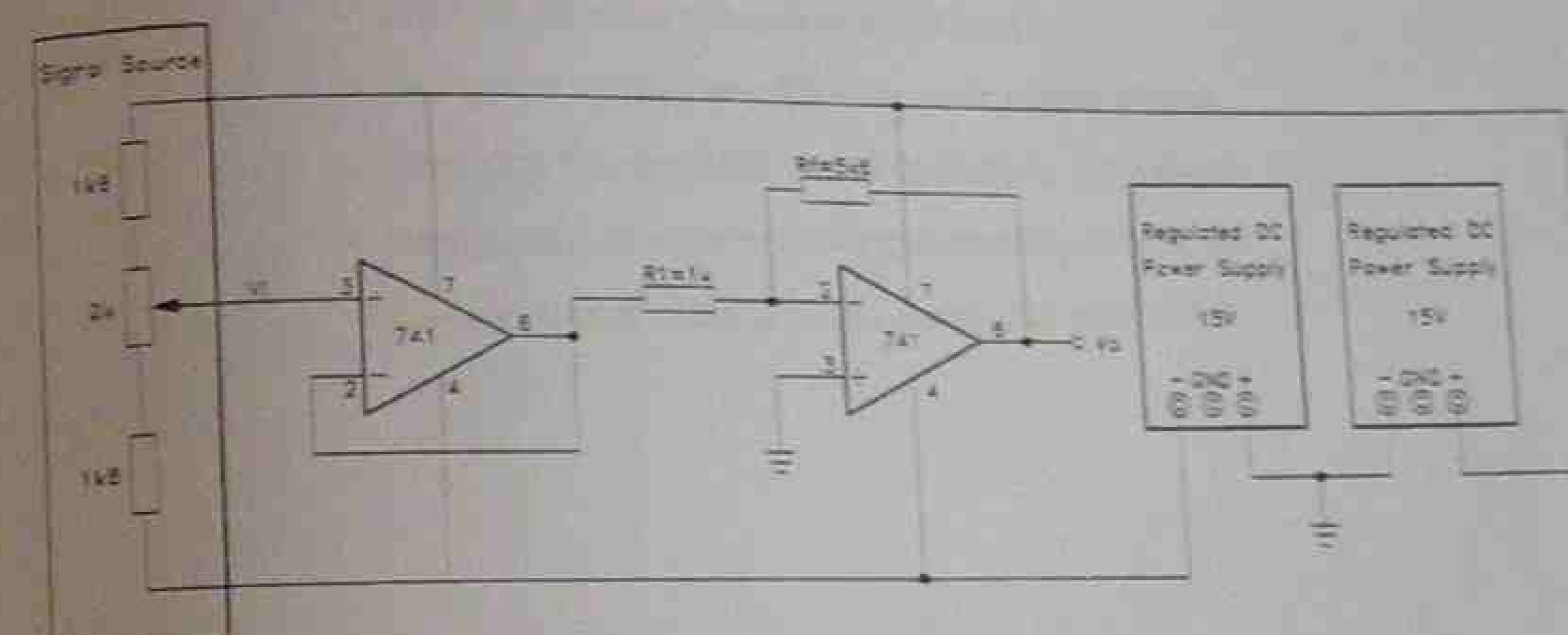
(i) Calculate theoretical voltage gain of the inverting amplifier.

(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

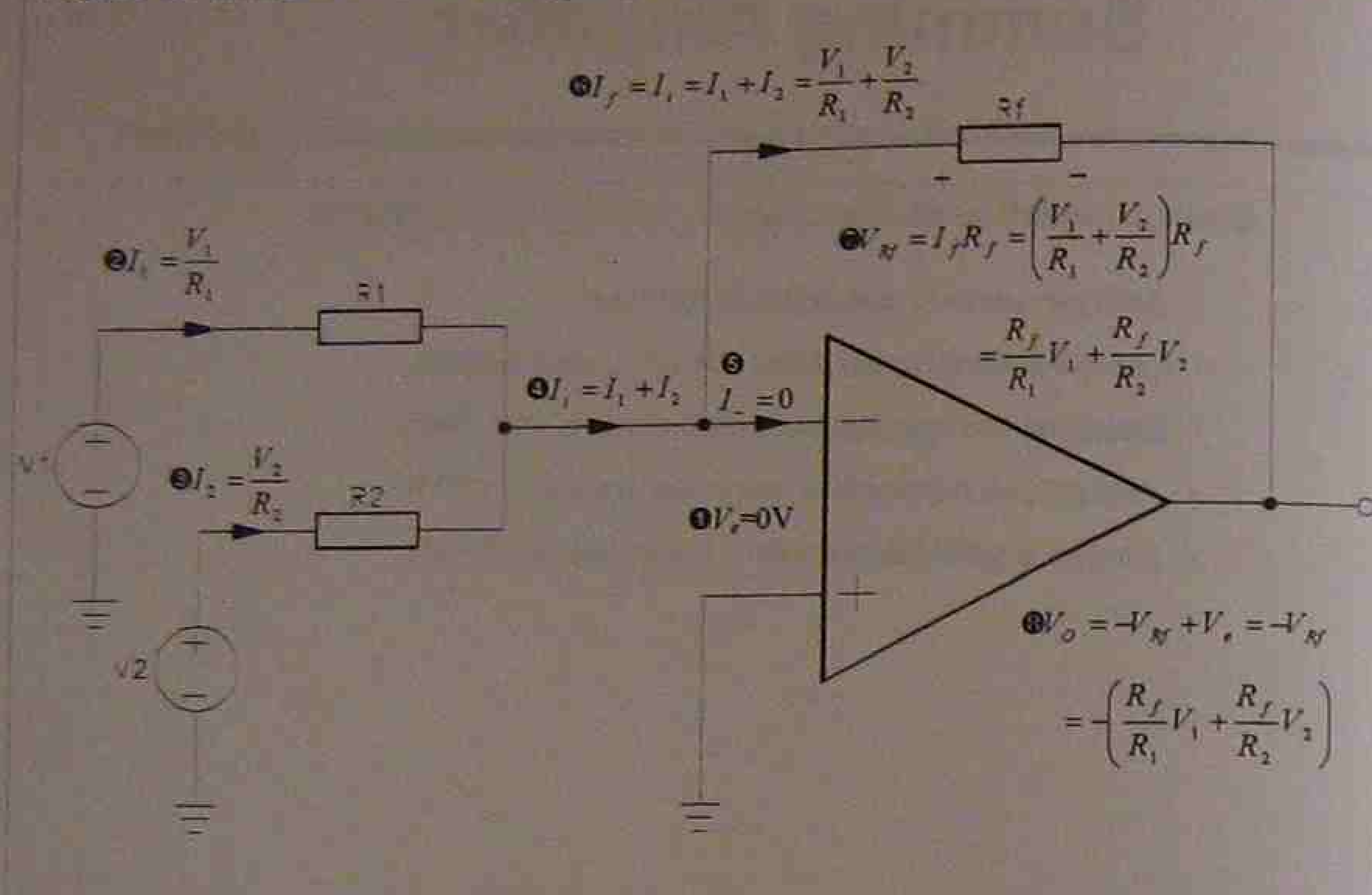
Upon completion of this chapter, you should be able to:

- 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

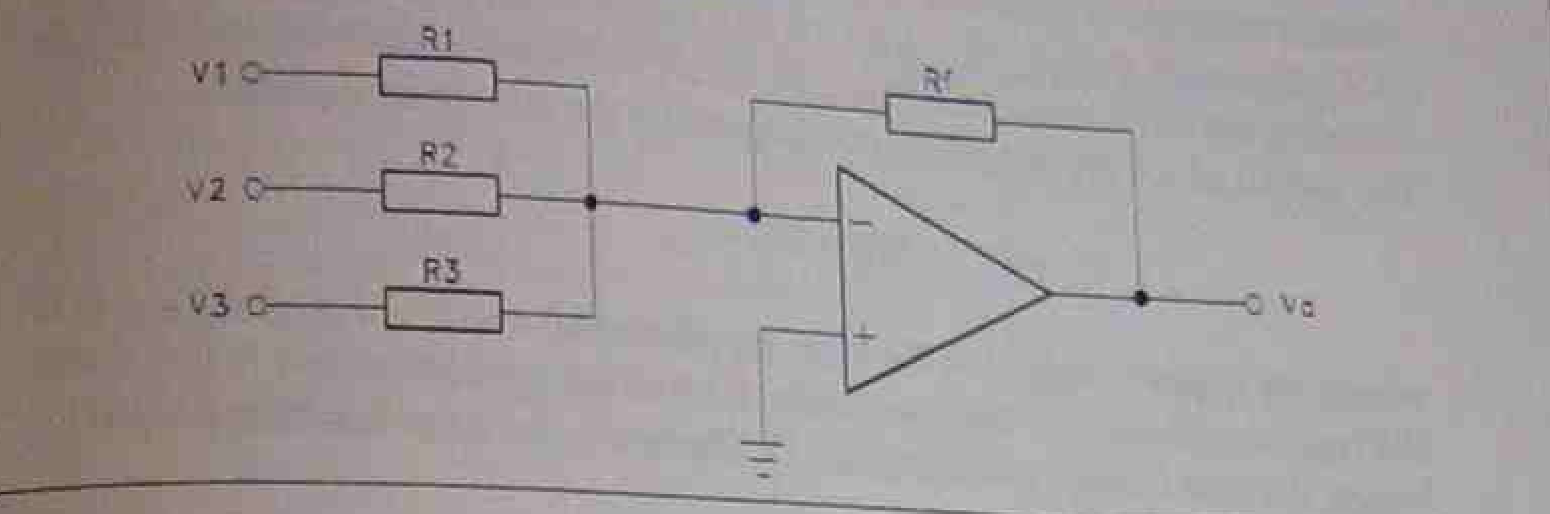
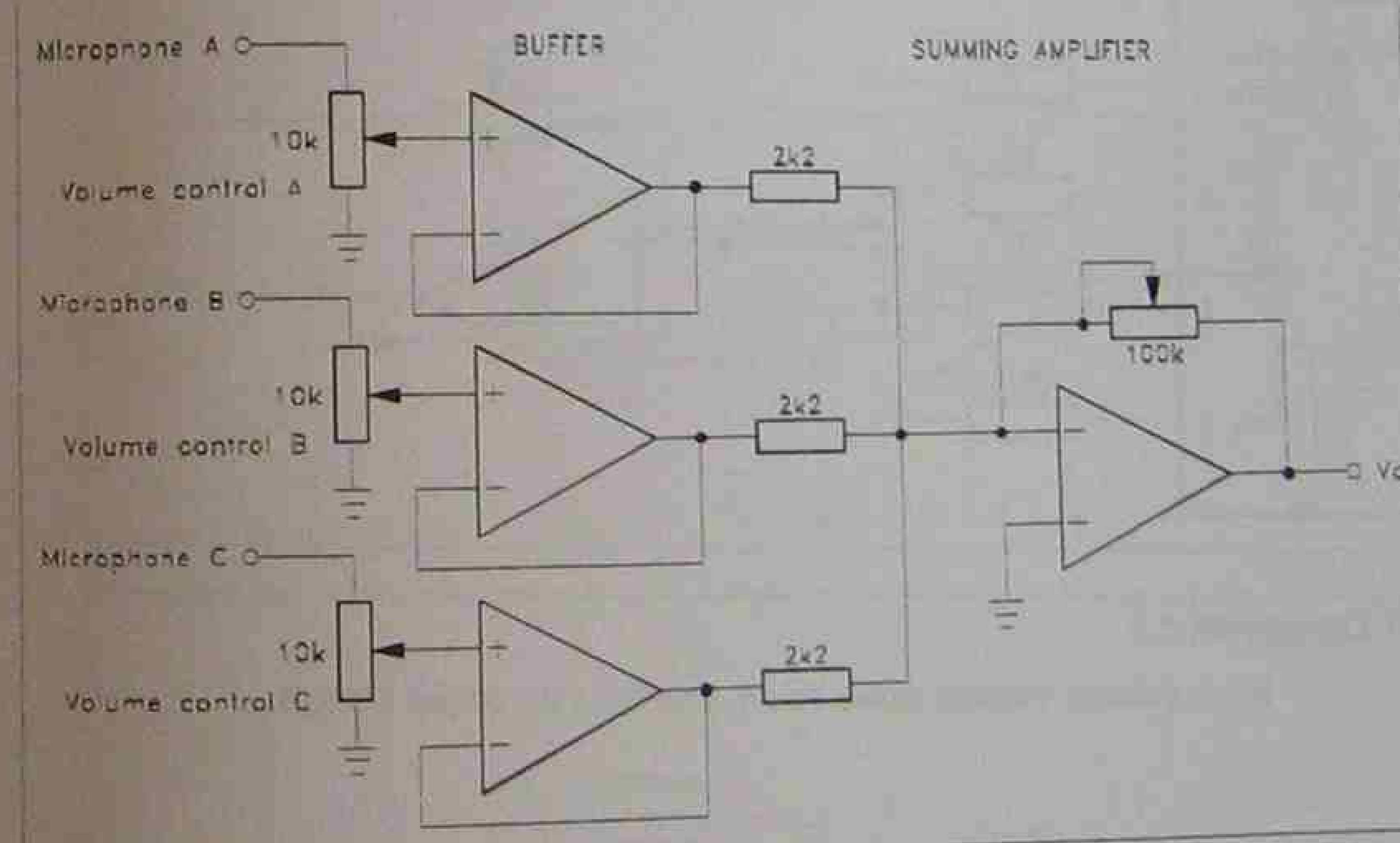


Figure 5.3 shows a convenient way to additively mix three microphone signals with a gain control. Three 10k potentiometers allow us to set the level of each input and voltage followers (buffers) are employed to avoid signal attenuation between the potentiometers and summing amplifier. The summing amplifier 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:

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

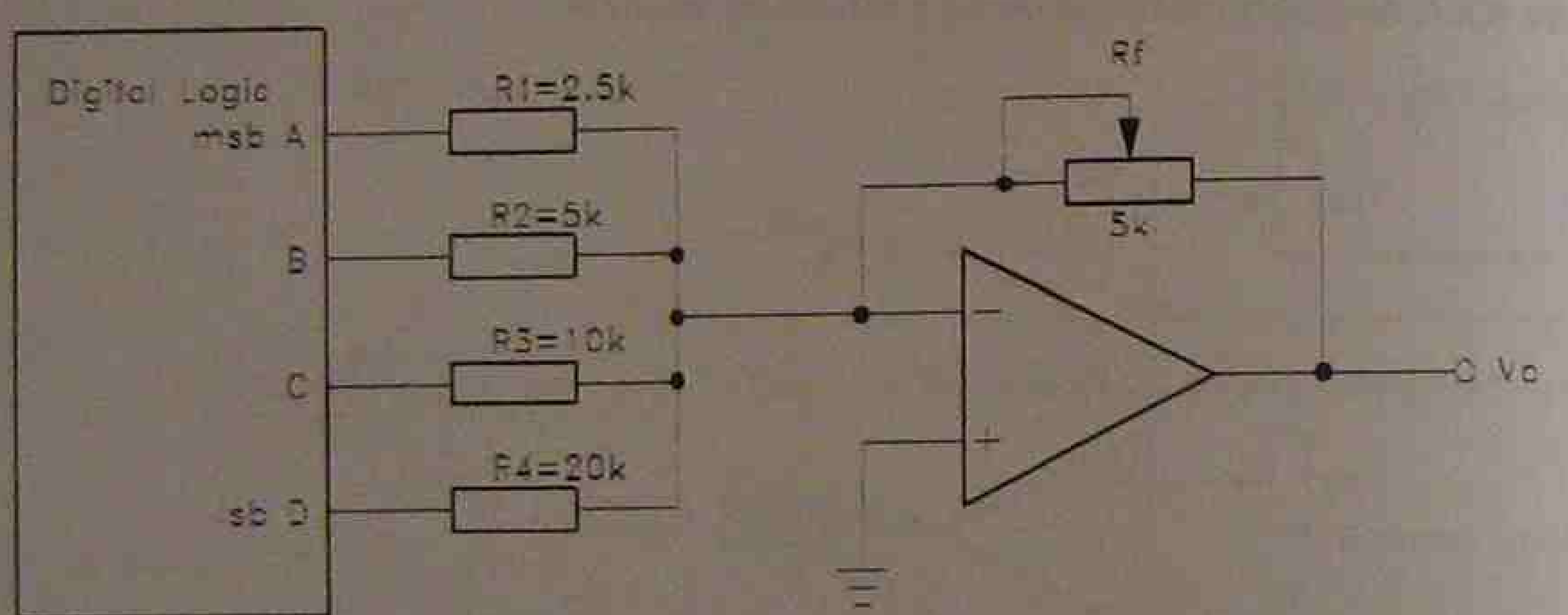
where the digital output A is the most significant bit and the digital output D is the least significant bit. In other words, the voltage gain for signal A is 8 times bigger than the voltage gain for signal D. Therefore, the input resistance ratio is

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

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

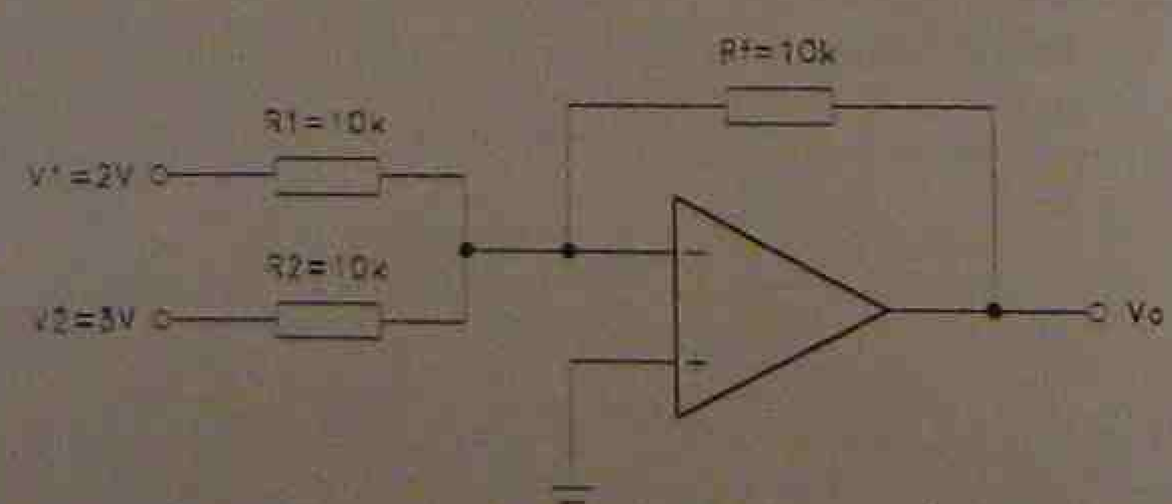
In Figure 5.4, a 5k variable resistor is employed to adjust the output voltage level.

Figure 5.4 A 4bit D/A converter



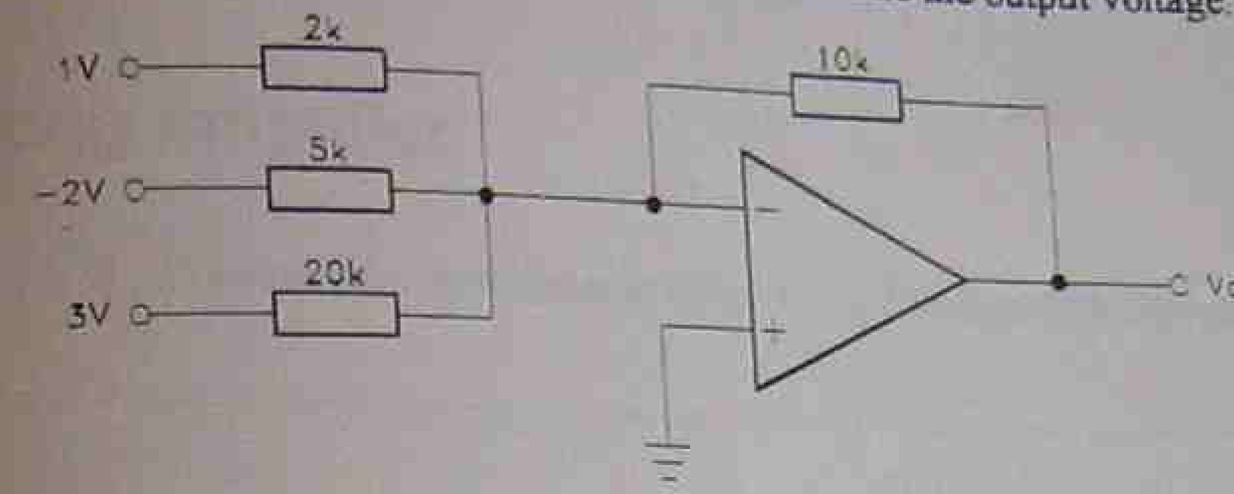
Drill Question 13

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



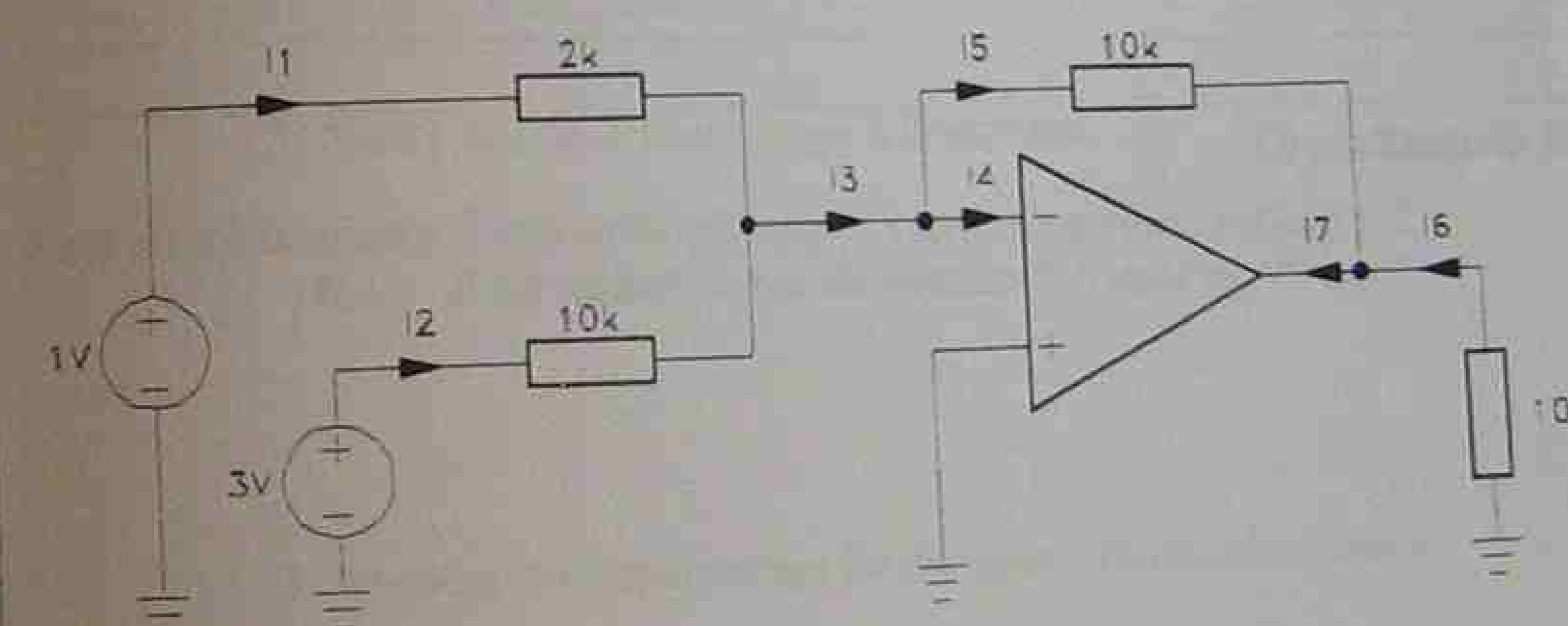
Drill Question 14

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

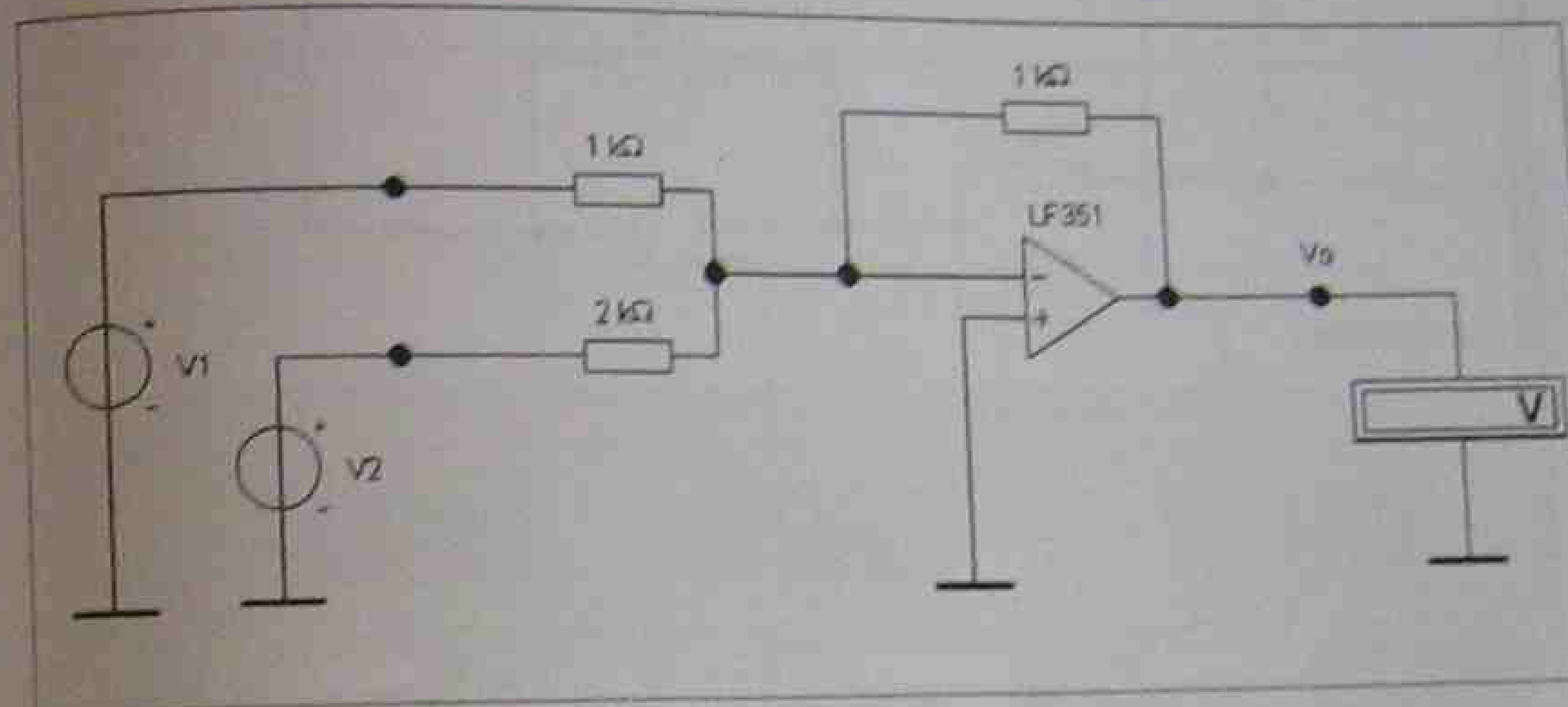
Drill Question 17

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:

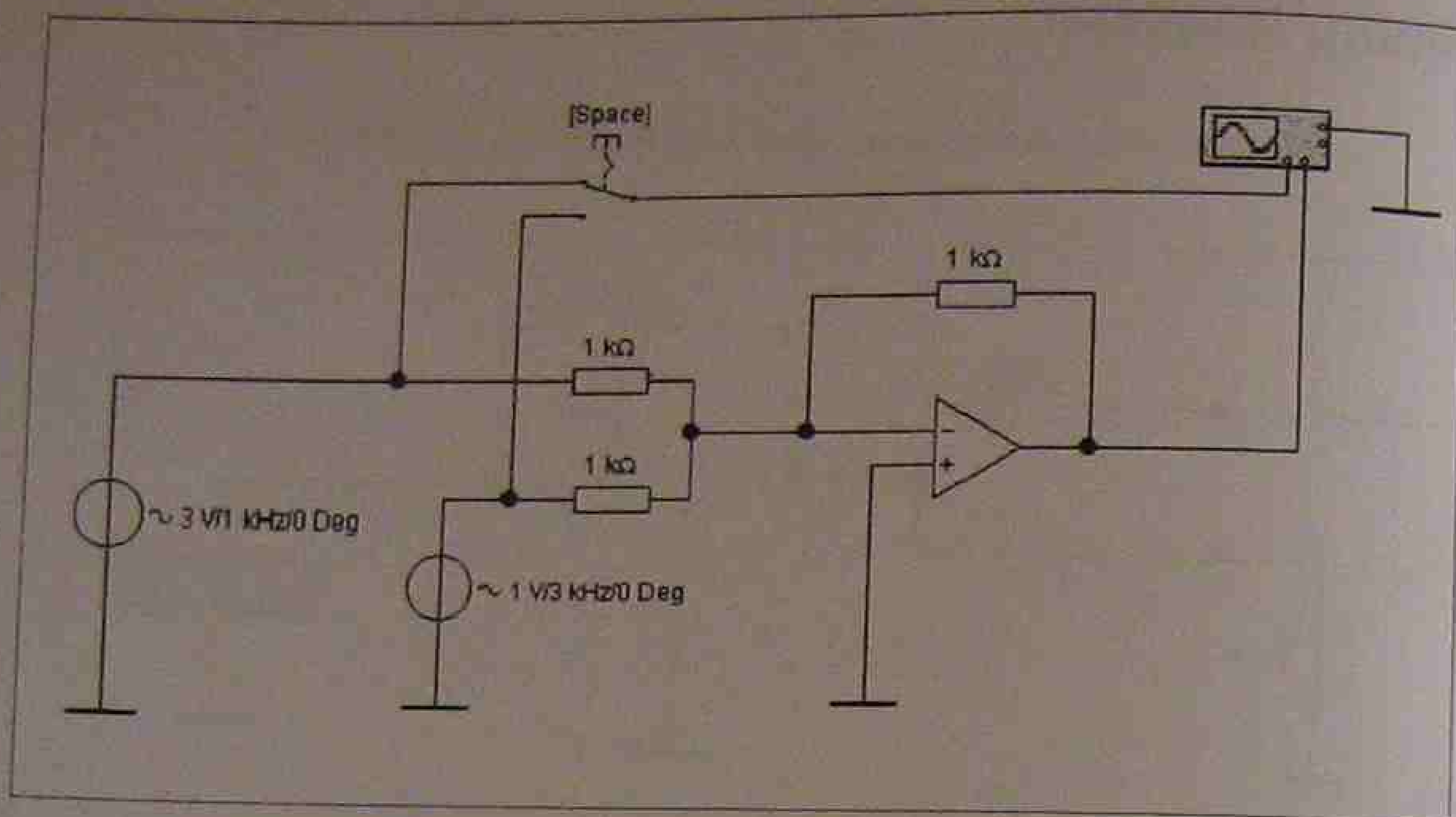
$$V_o = -(V_1 + 0.5V_2)$$

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

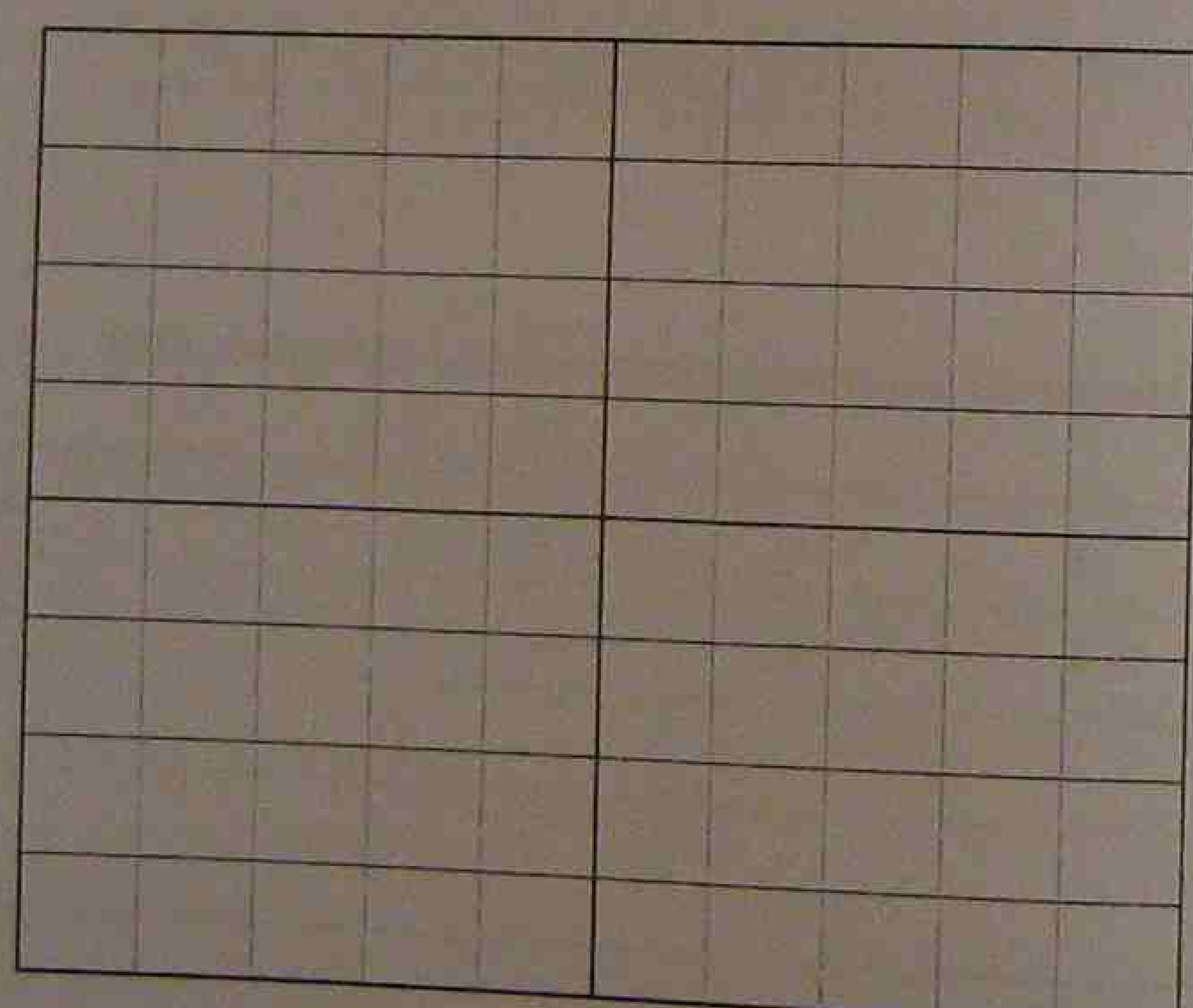
V_1 (V)	1	1	2	-2	-3
V_2 (V)	1	-2	1	2	-1
V_o (Theory)					
V_o (Measured)					

Signal Mixer

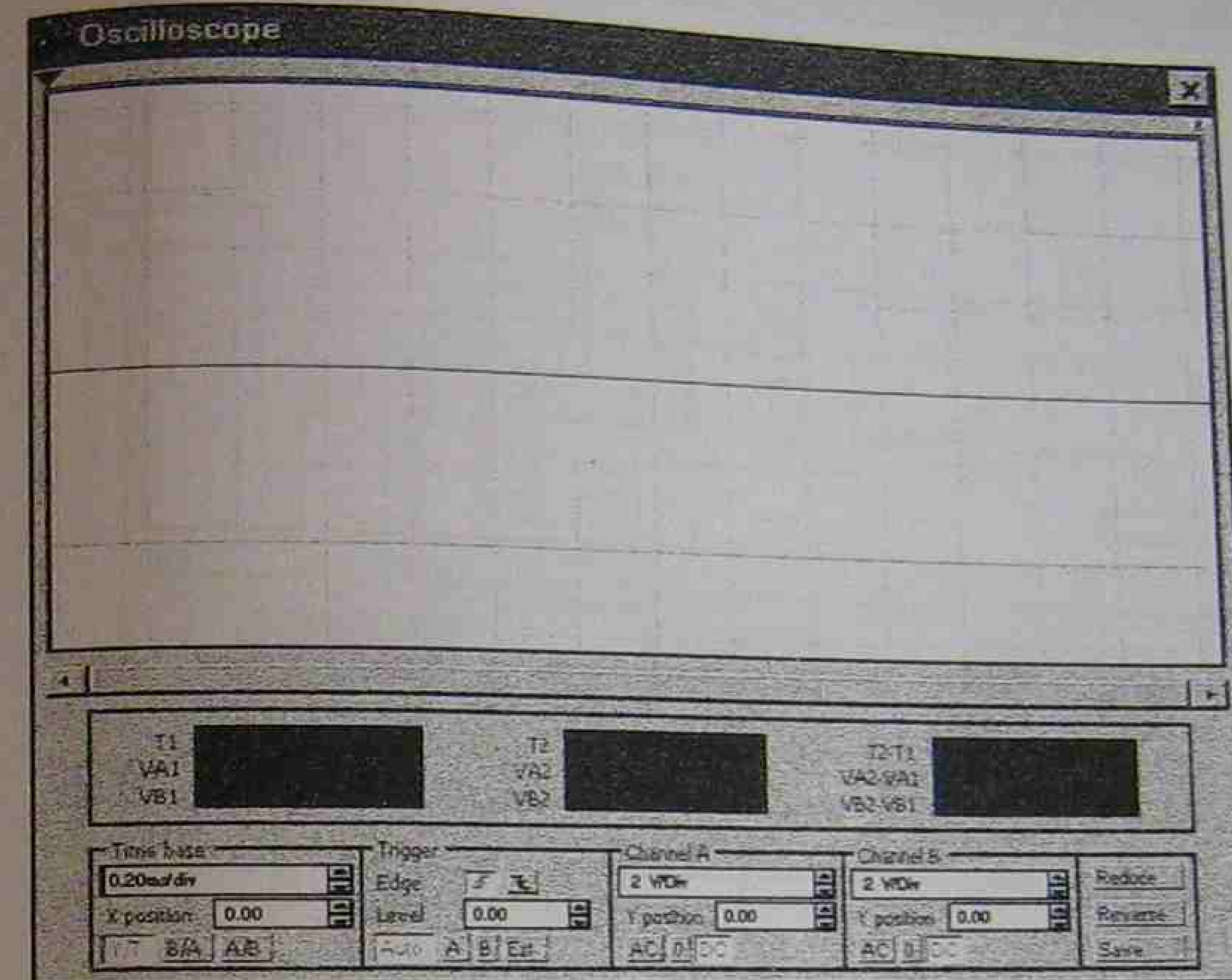
1. Construct the circuit shown below using Electronics Workbench.



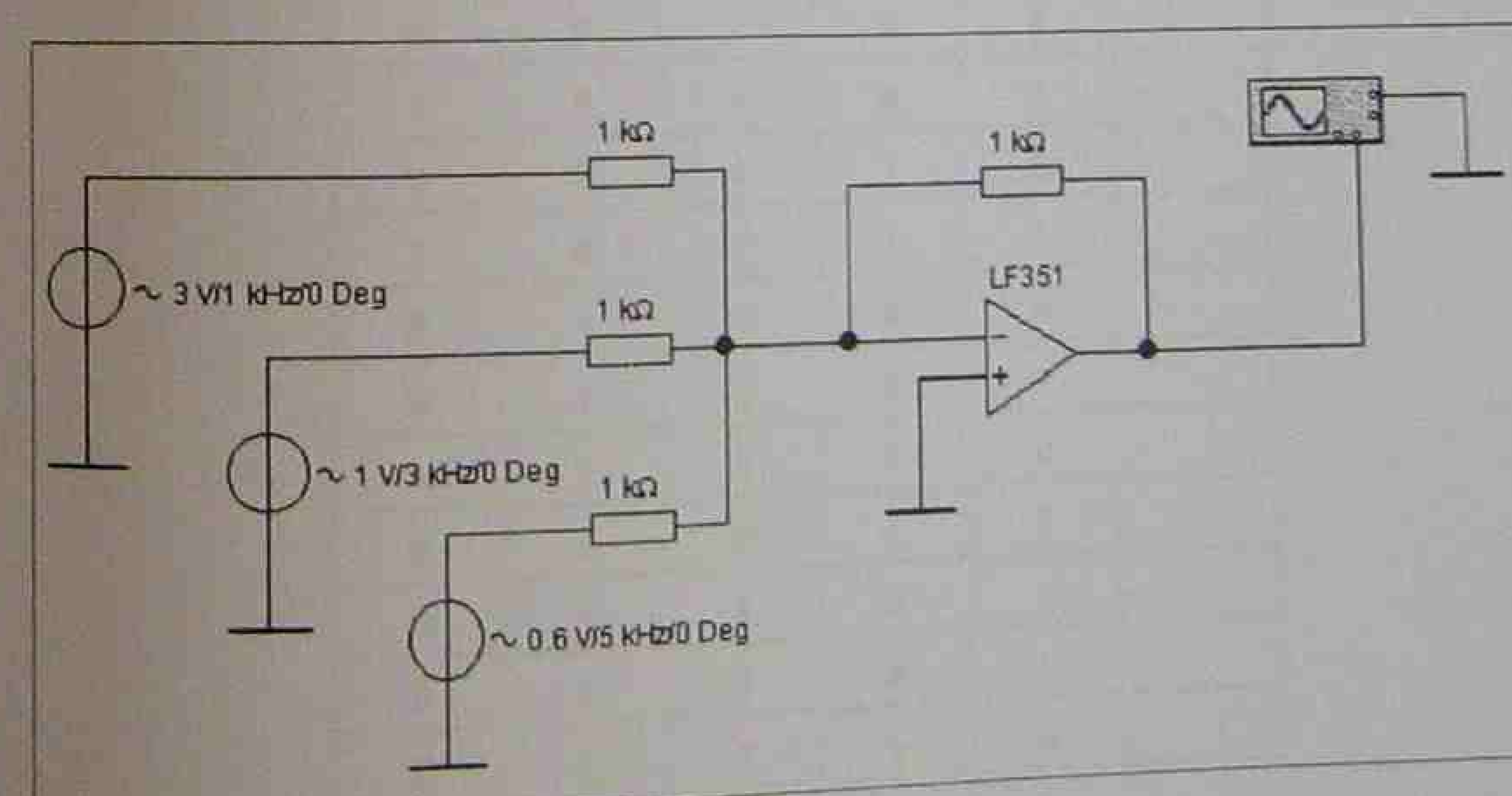
2. Sketch the expected input/output waveforms.



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

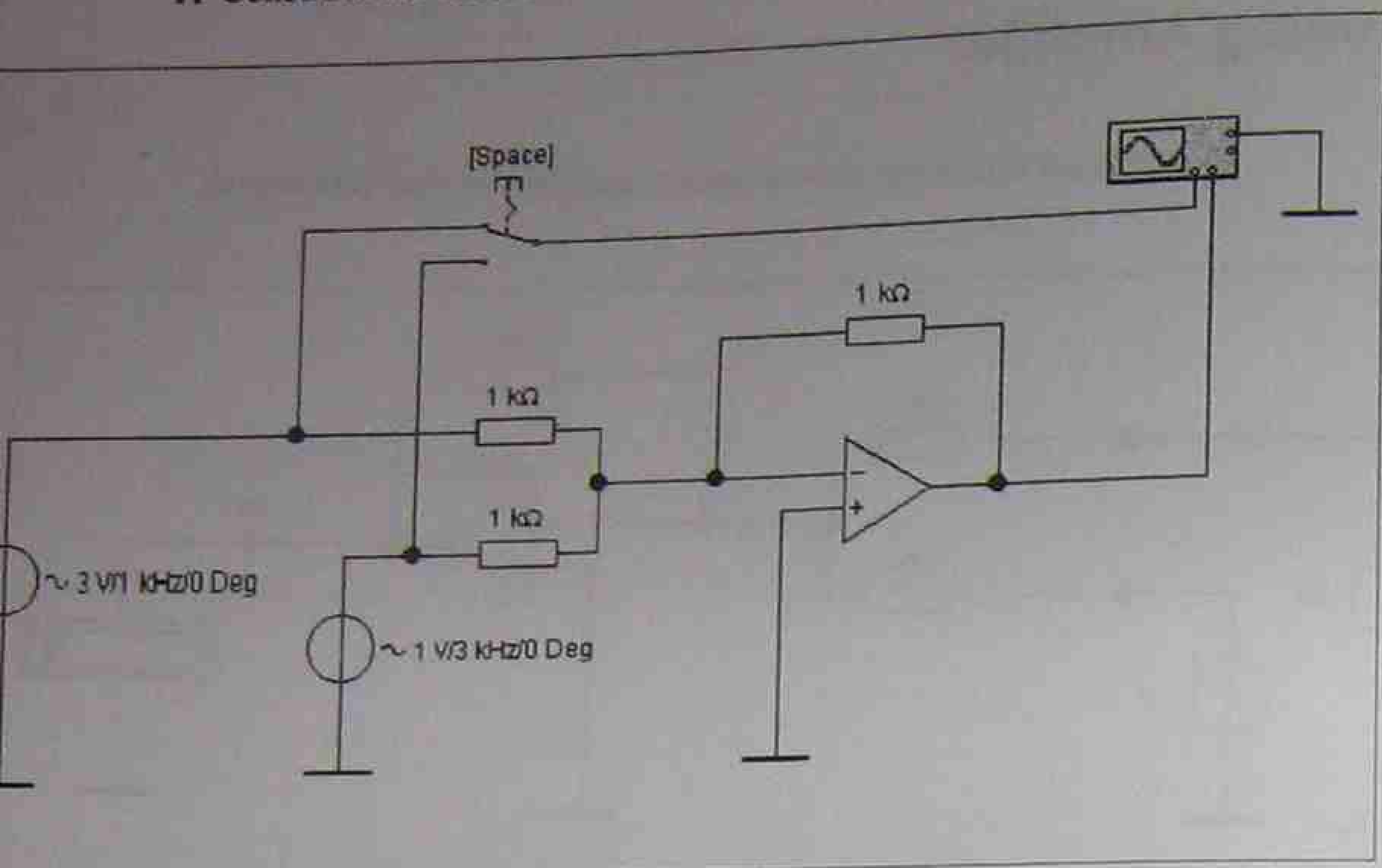


4. Add one more signal source as shown below and observe the output waveform.



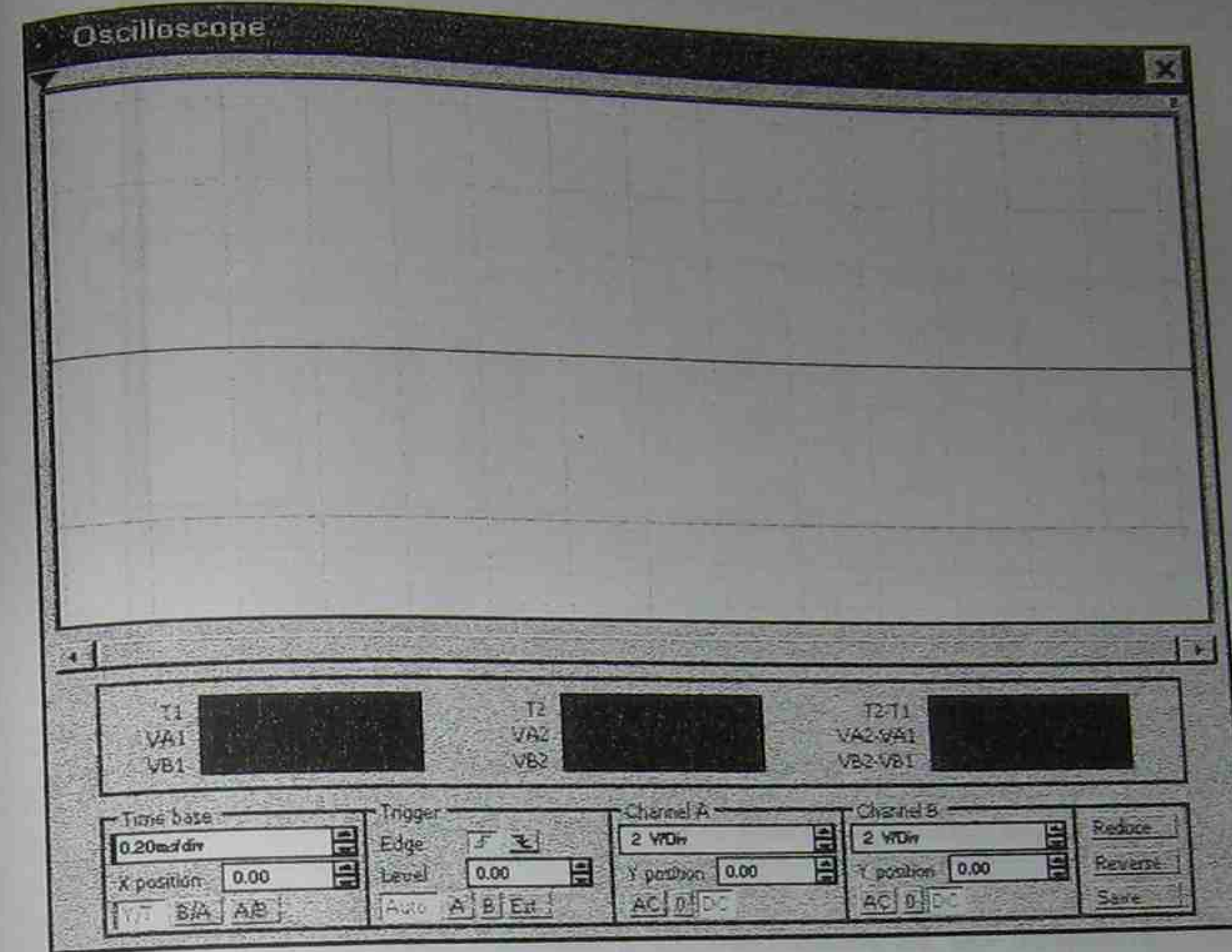
gnal Mixer

1. Construct the circuit shown below using Electronics Workbench.

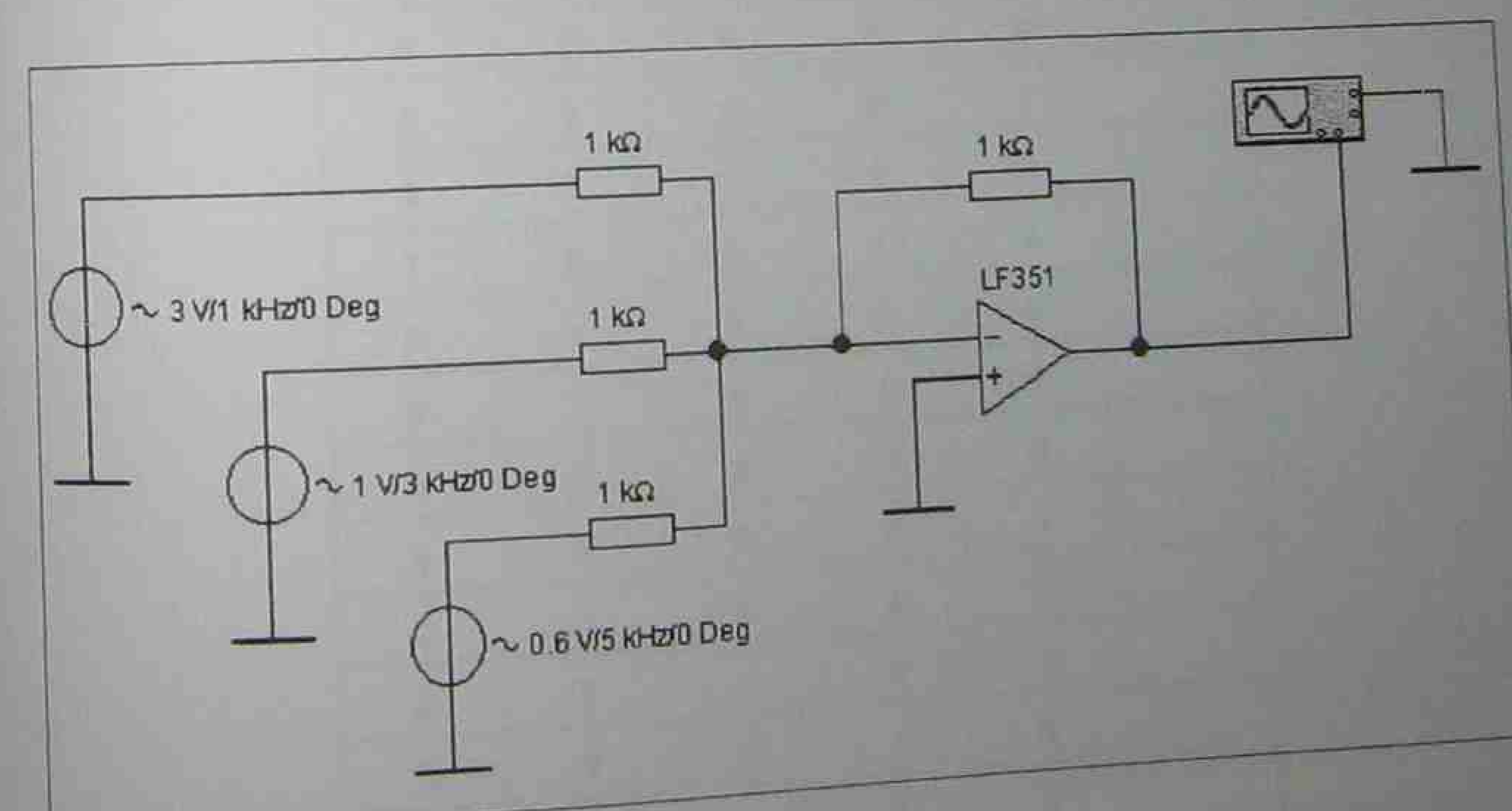


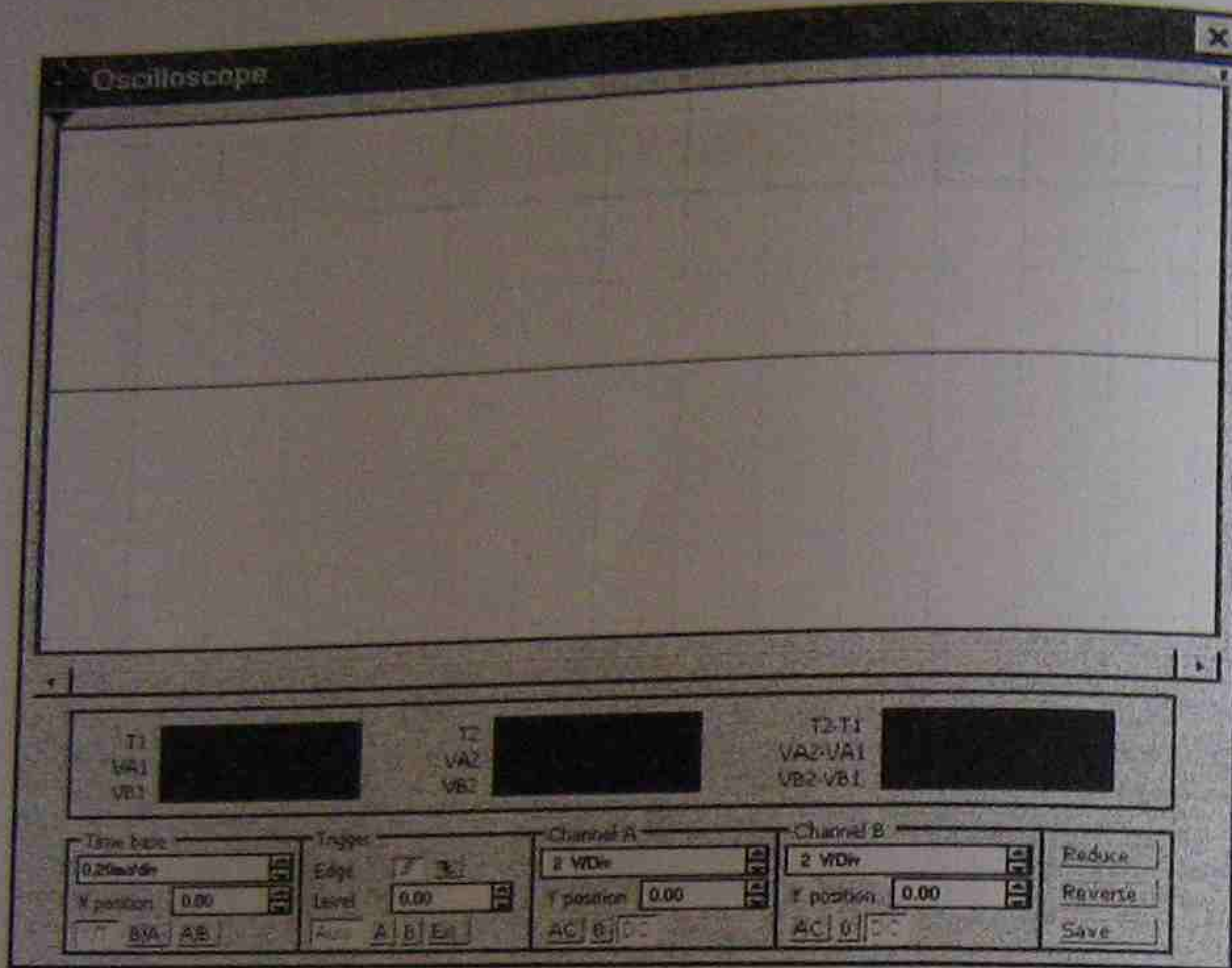
2. Sketch the expected input/output waveforms.

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



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

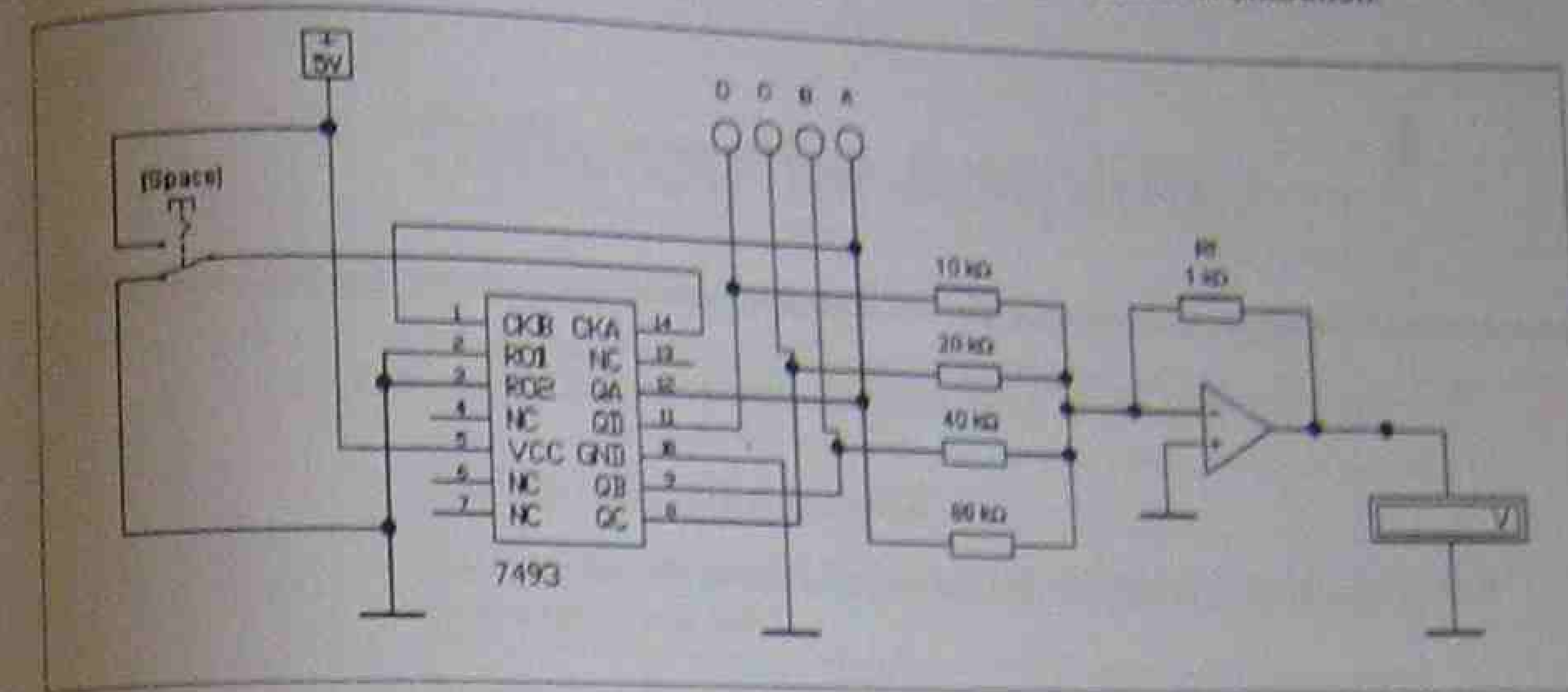




5. Explain why the voltage followers (buffers) are employed for the signal mixer in Fig. 5.3.

D/A Converter

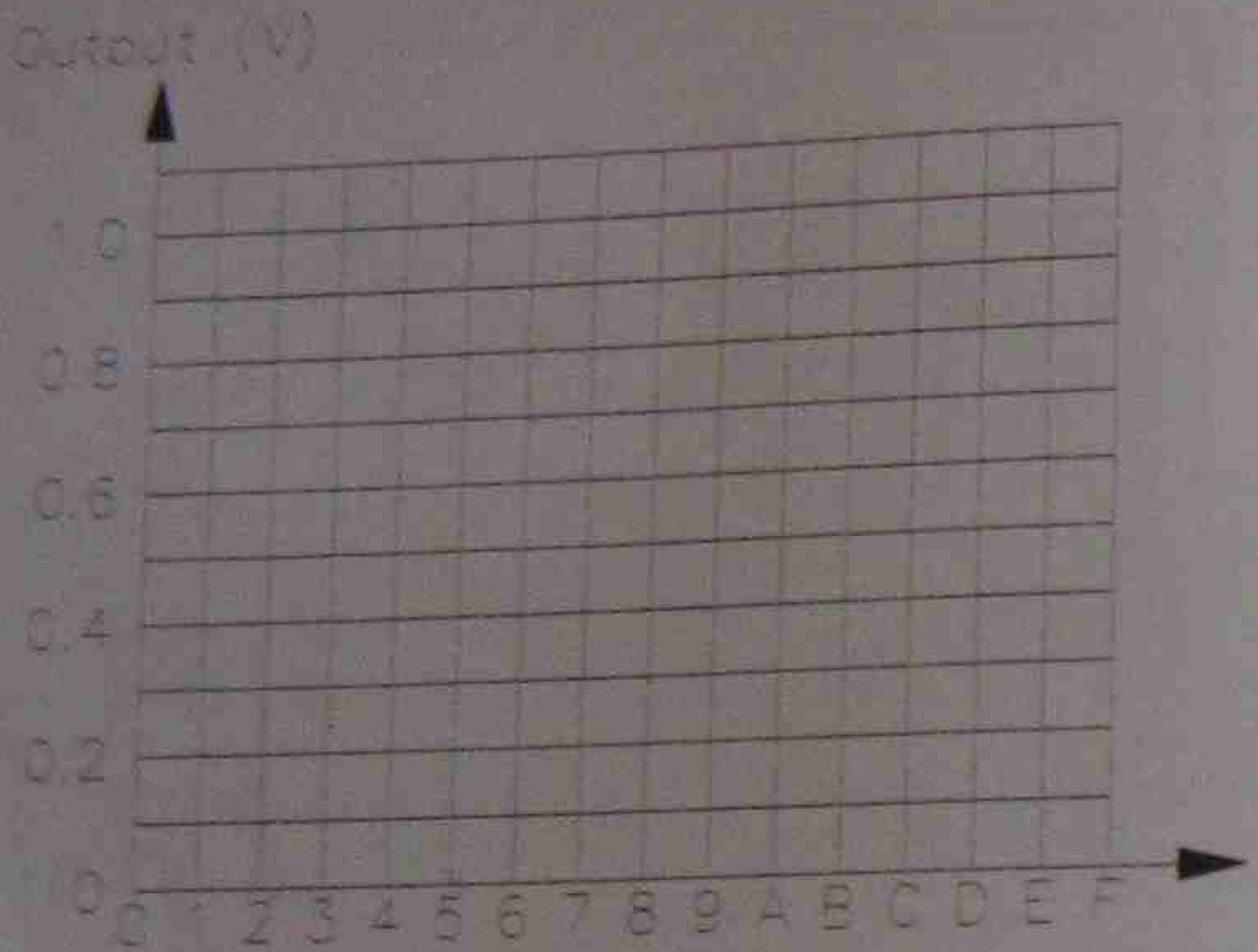
1. Construct the circuit shown below using Electronics Workbench



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

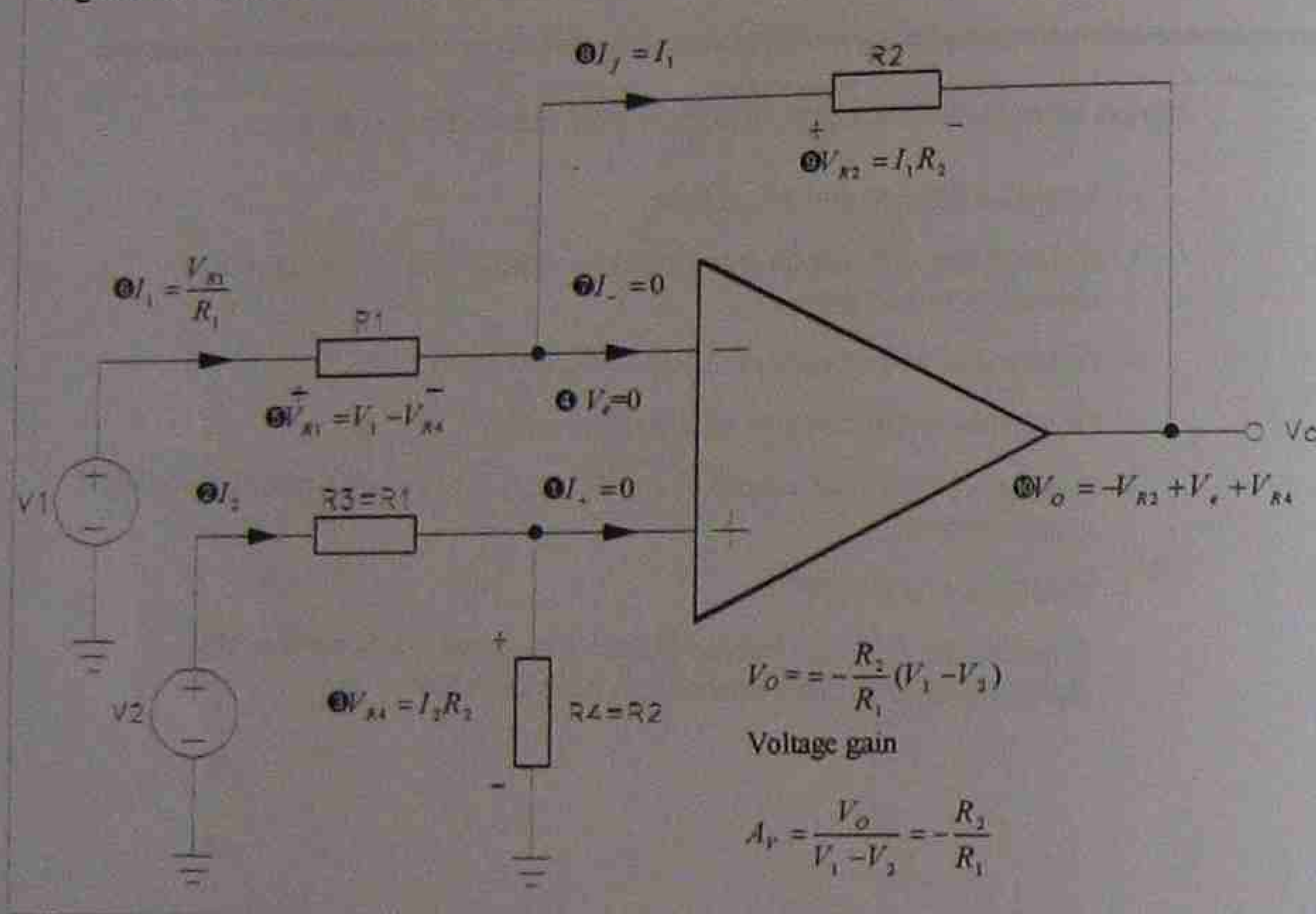
Upon completion of this chapter, you should be able to:

- 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:

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

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}).

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

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

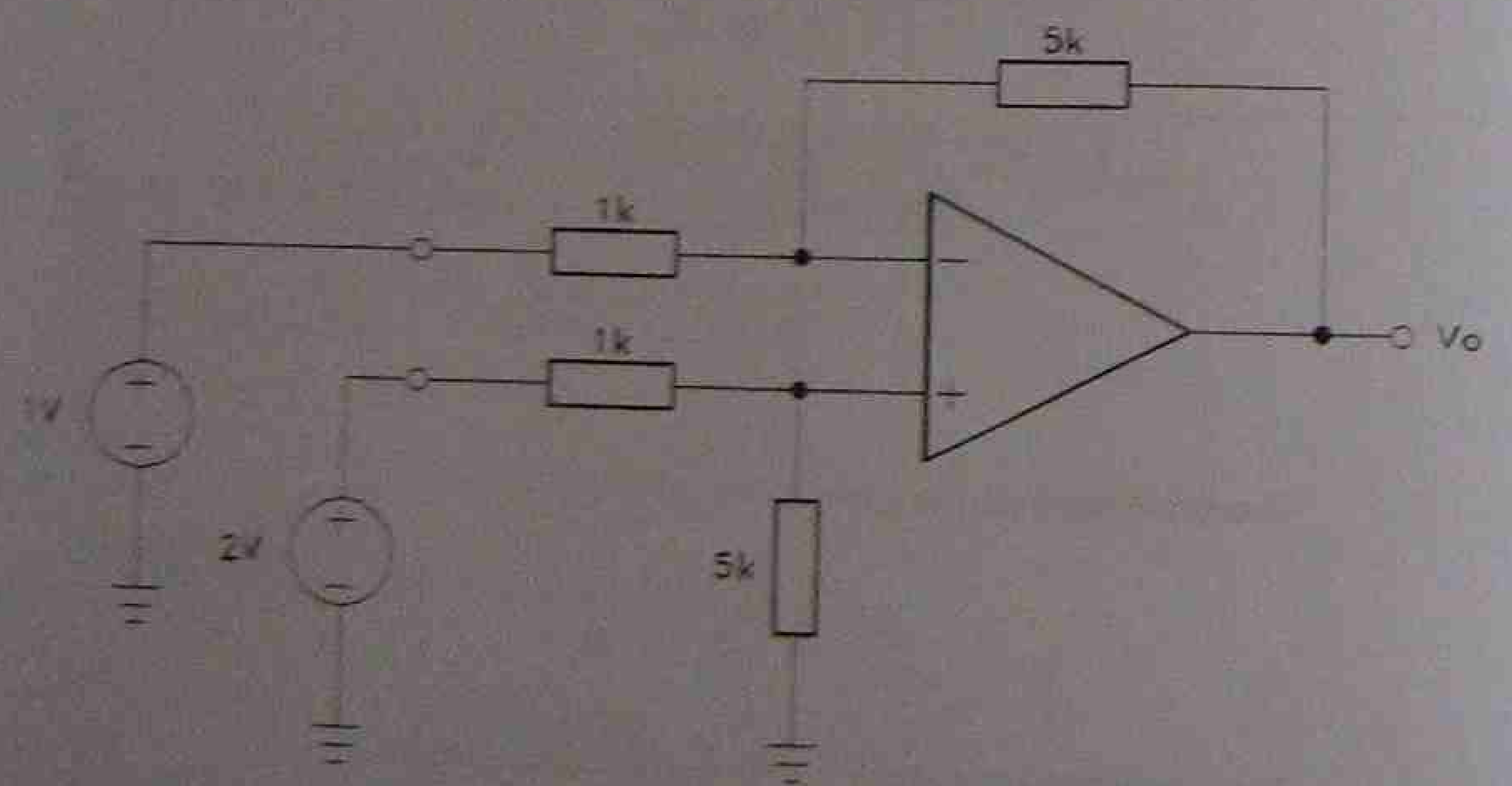
Output Resistance

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.



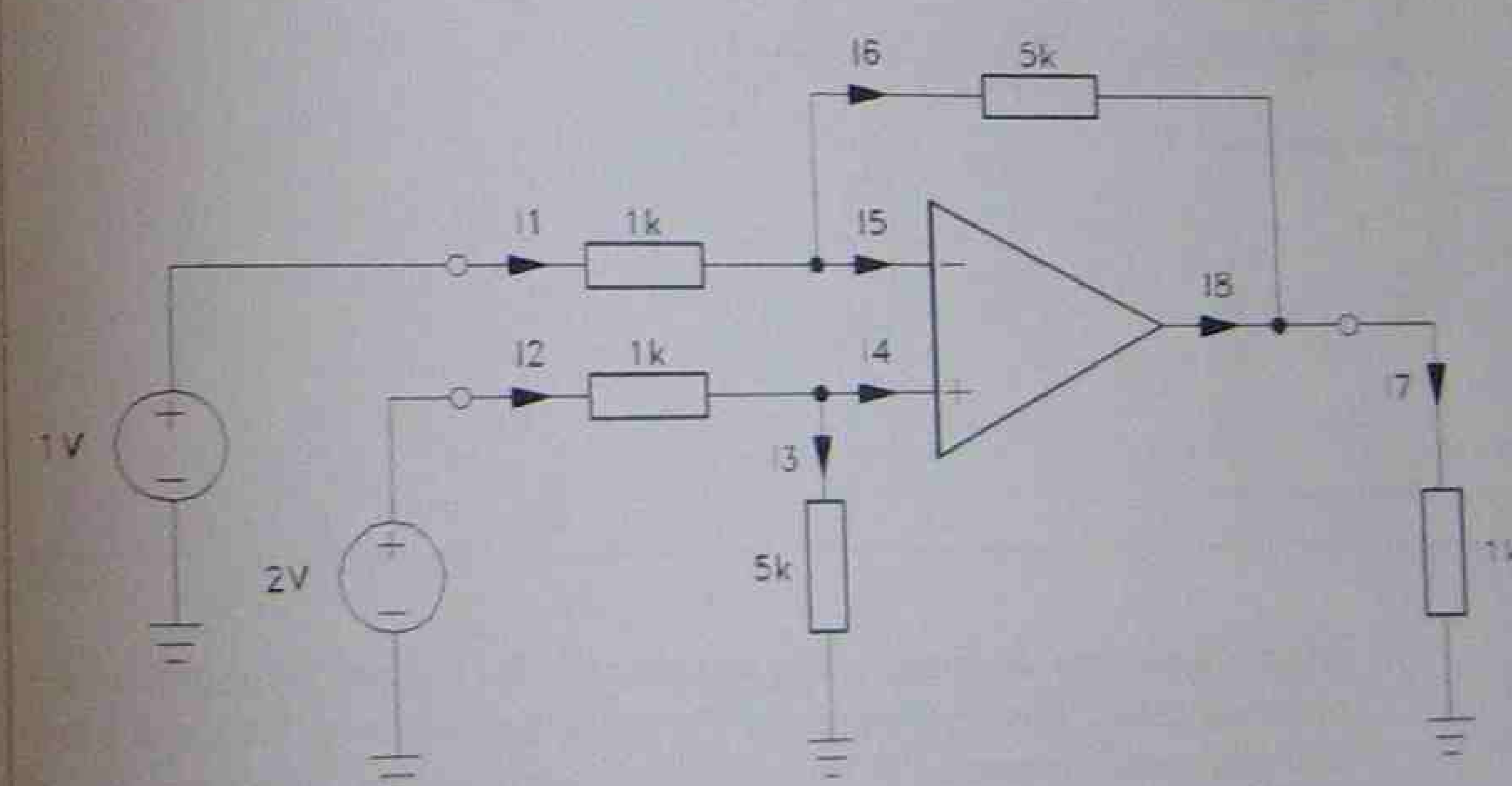
Drill Question 2

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

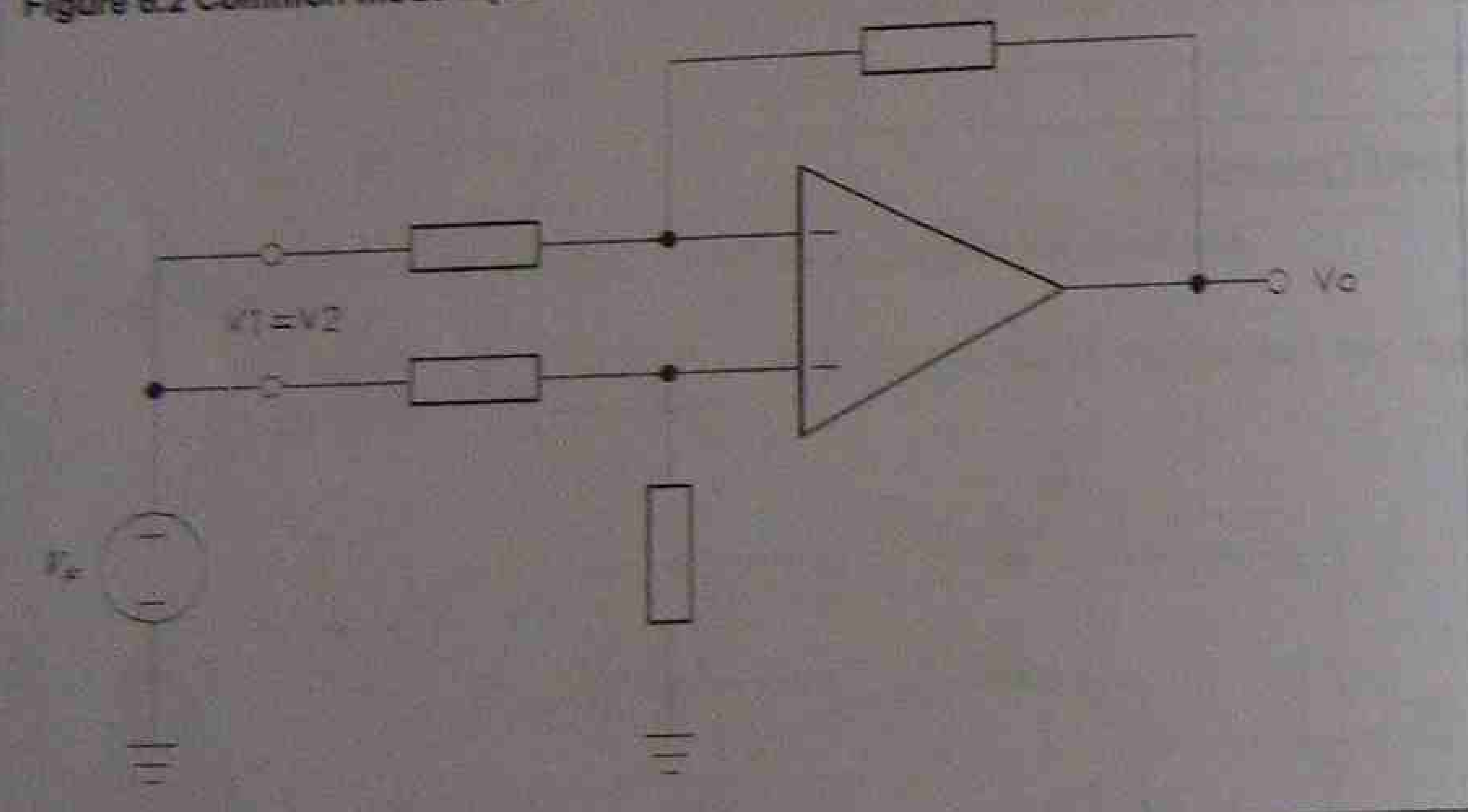
A common mode signal is one that drives both inputs of a diferential amplifier equally. Figure 4.18 shows a common mode input. Because two input voltages are same ($V_1 = V_2$), ideally the output voltage must be 0V. The common mode gain is defined as the ratio of output voltage to common mode input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

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

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

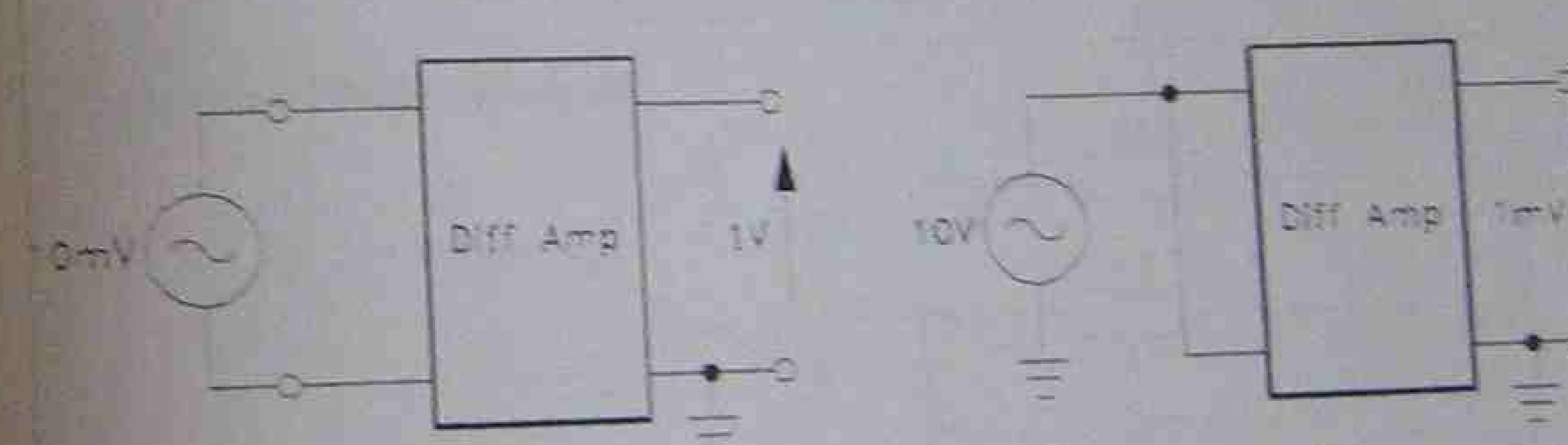
As we discussed in the common mode gain, the higher the CMMR is, the better the differential amplifier.

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:



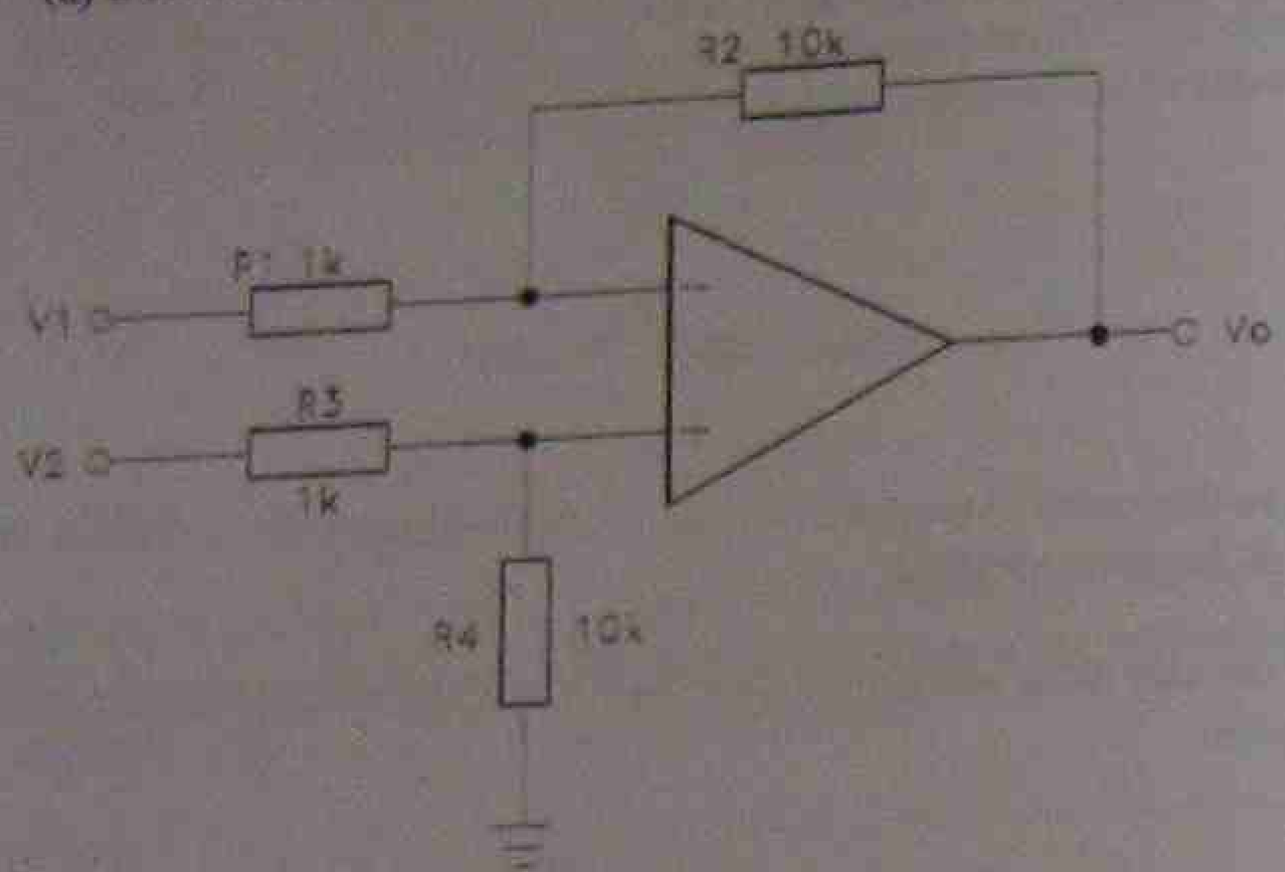
(a) Differential mode gain

(b) Commom mode gain

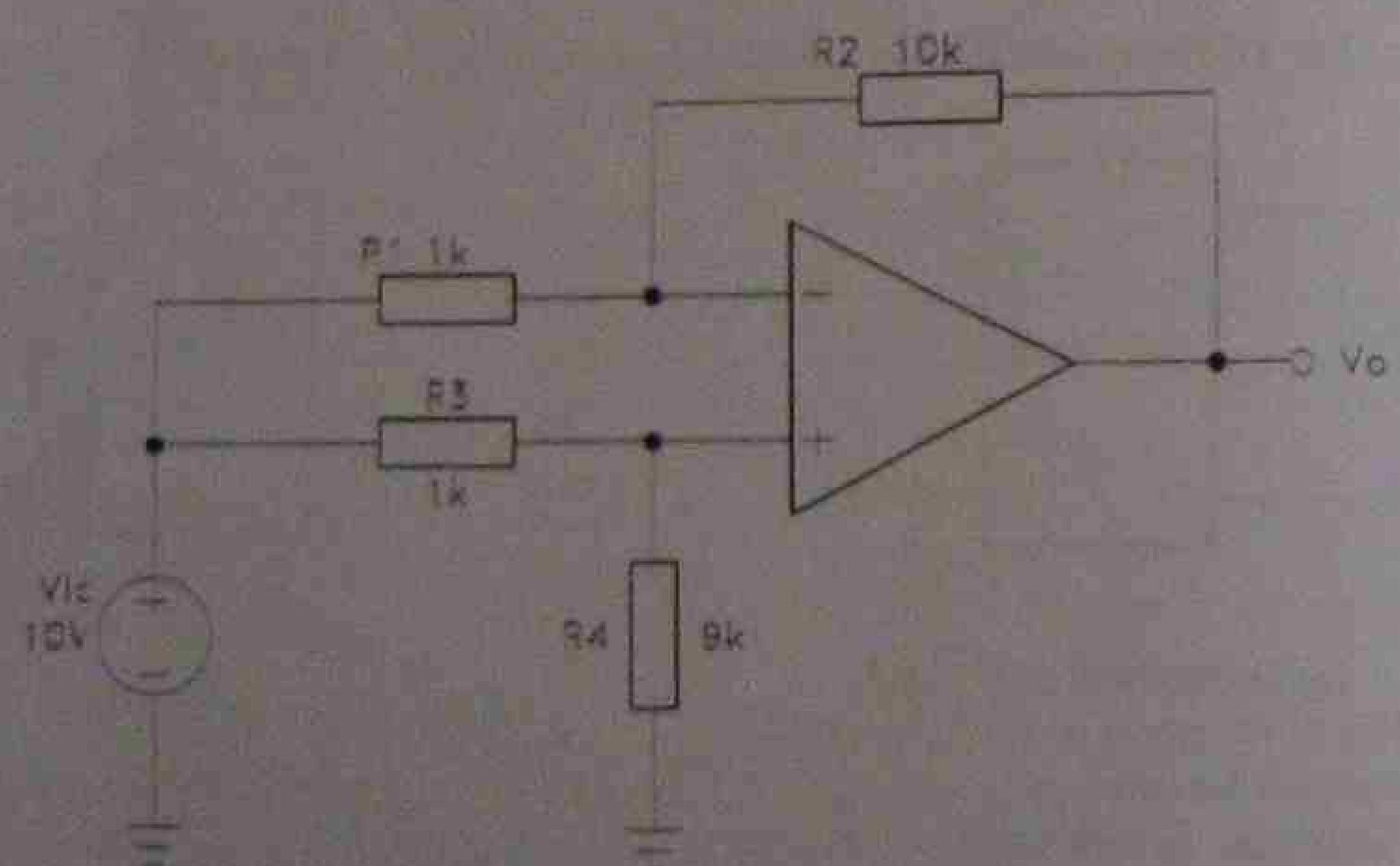
(c) CMMR

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.

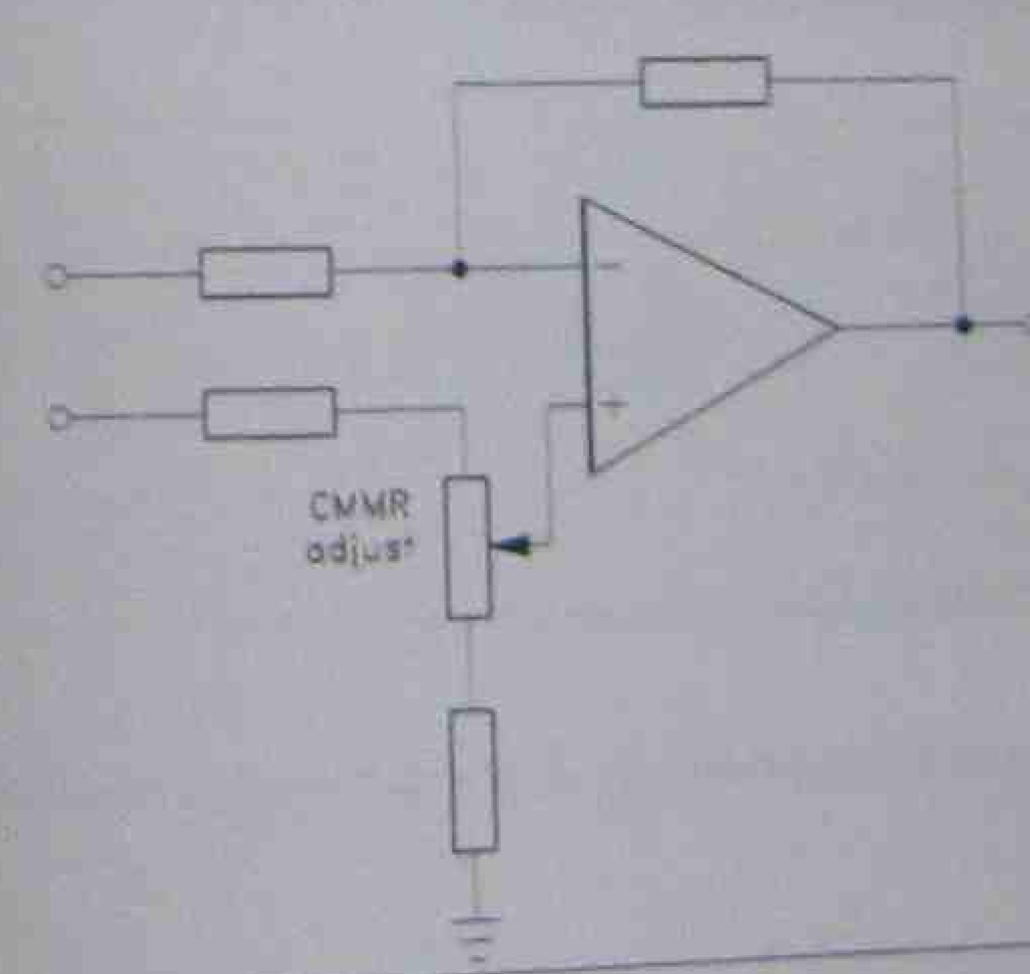
(v) Determine the output voltage.

(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.

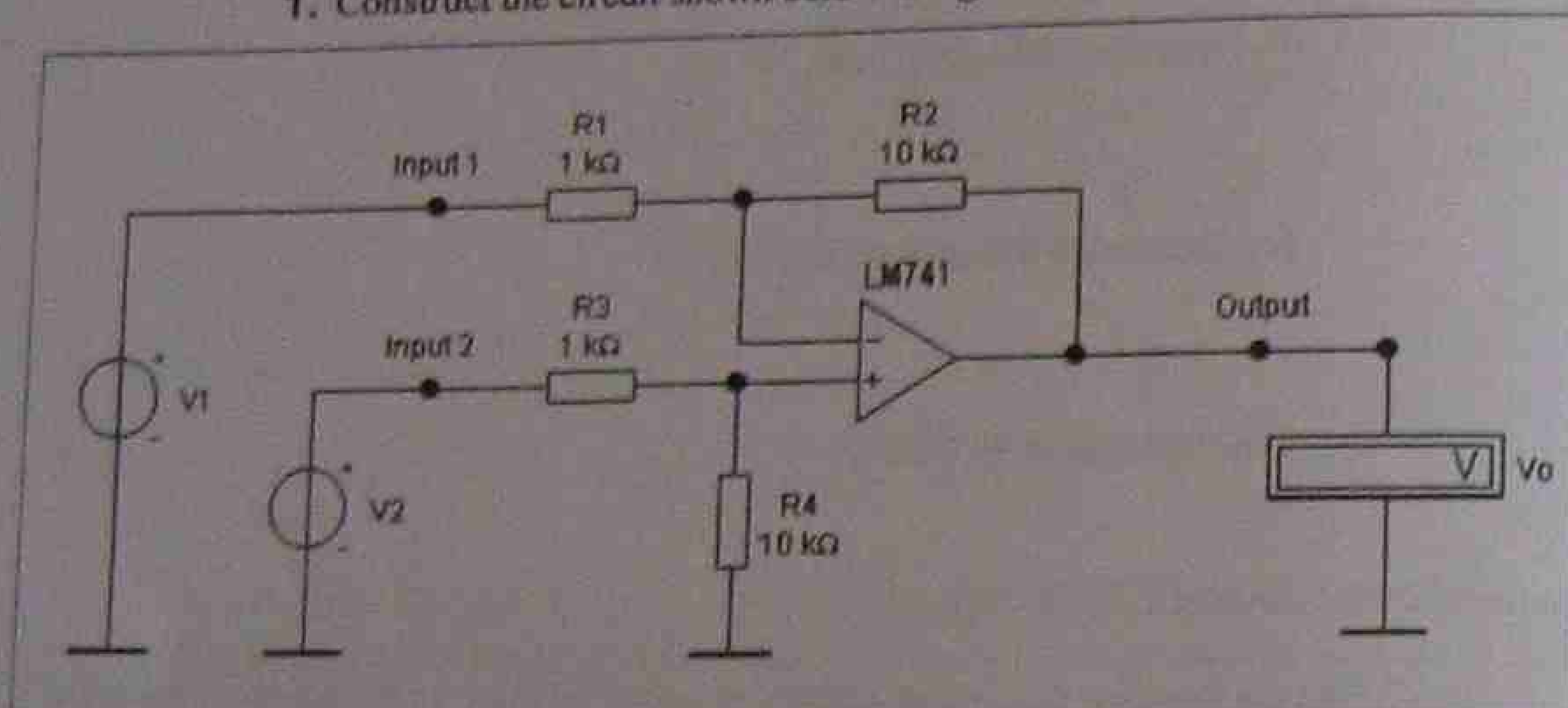


Skill Practice 7

Differential Amplifier

Differential Mode Gain

1. Construct the circuit shown below using Electronics Workbench.

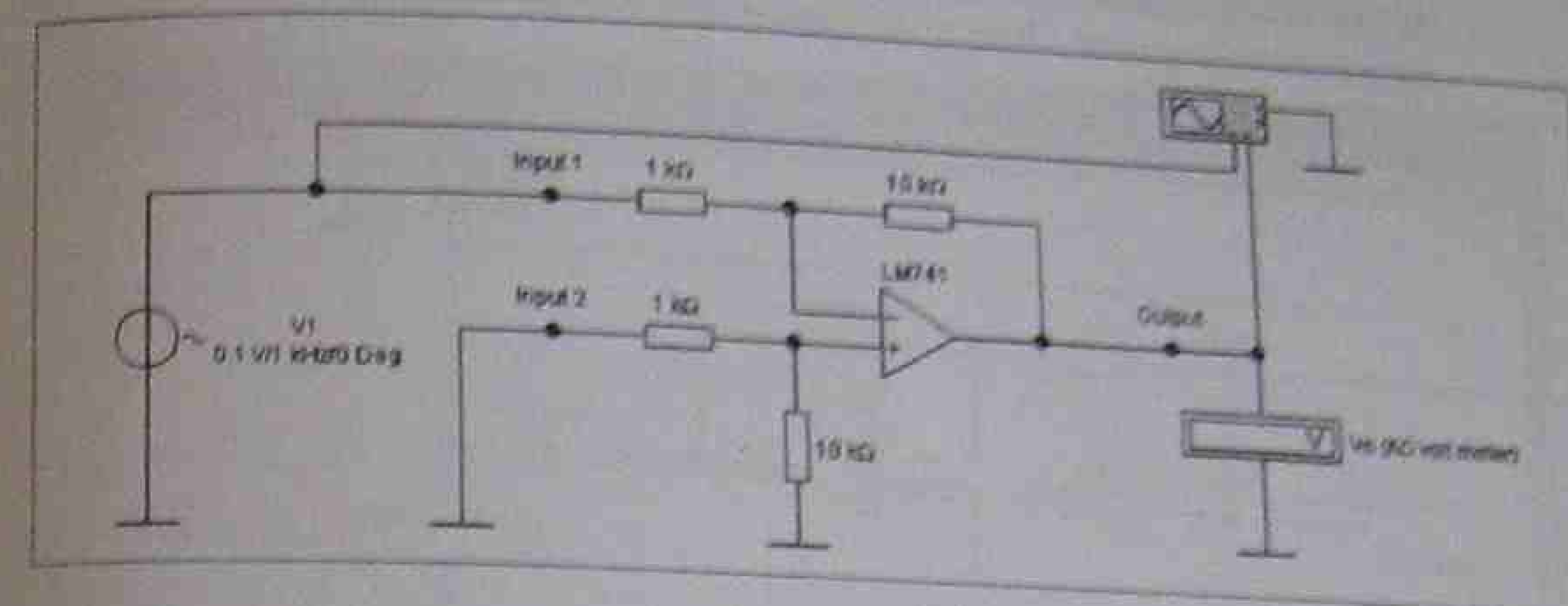


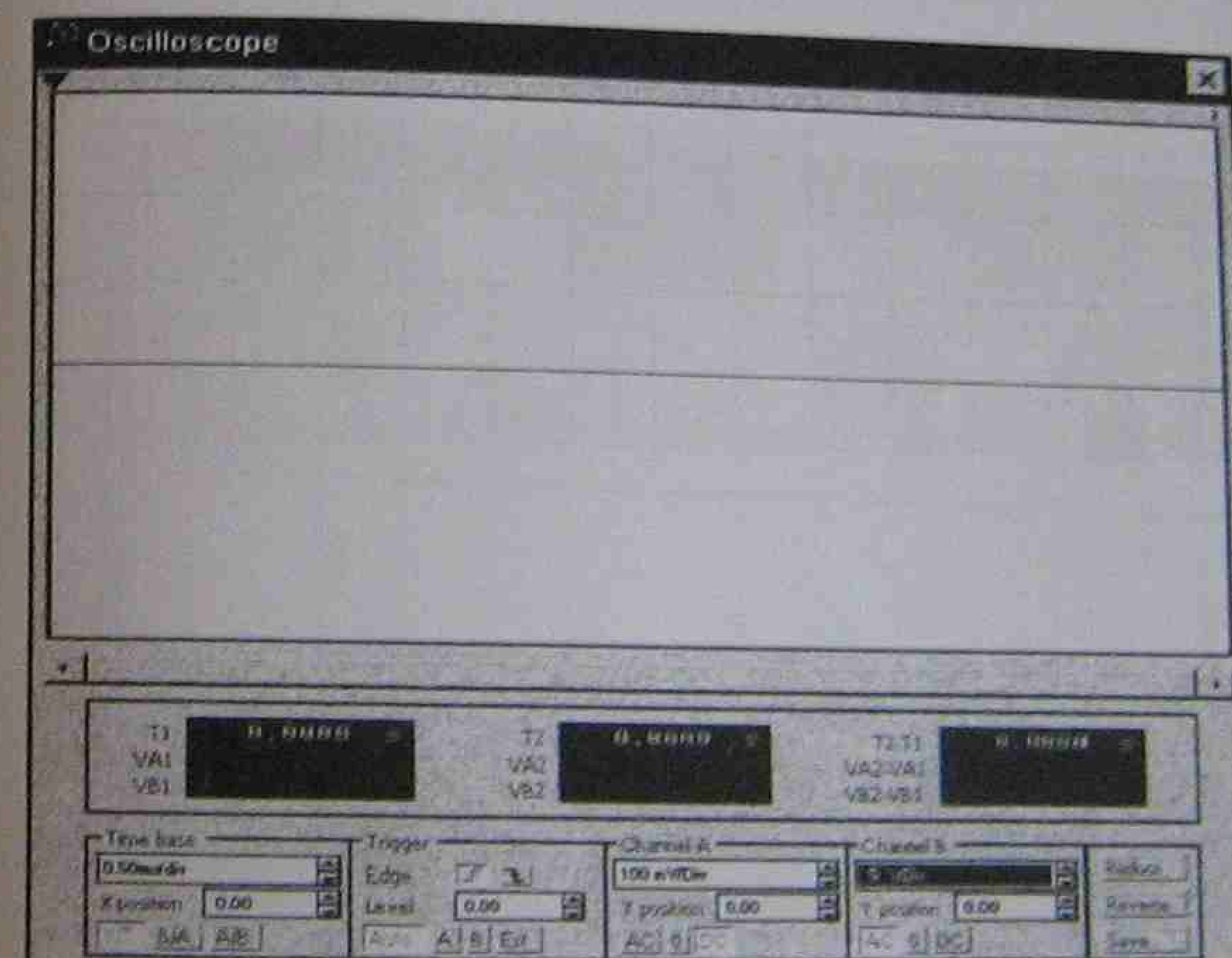
2. Measure the output voltages for the different input voltages shown below. Also, calculate the differential mode voltage gain A_d.

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.



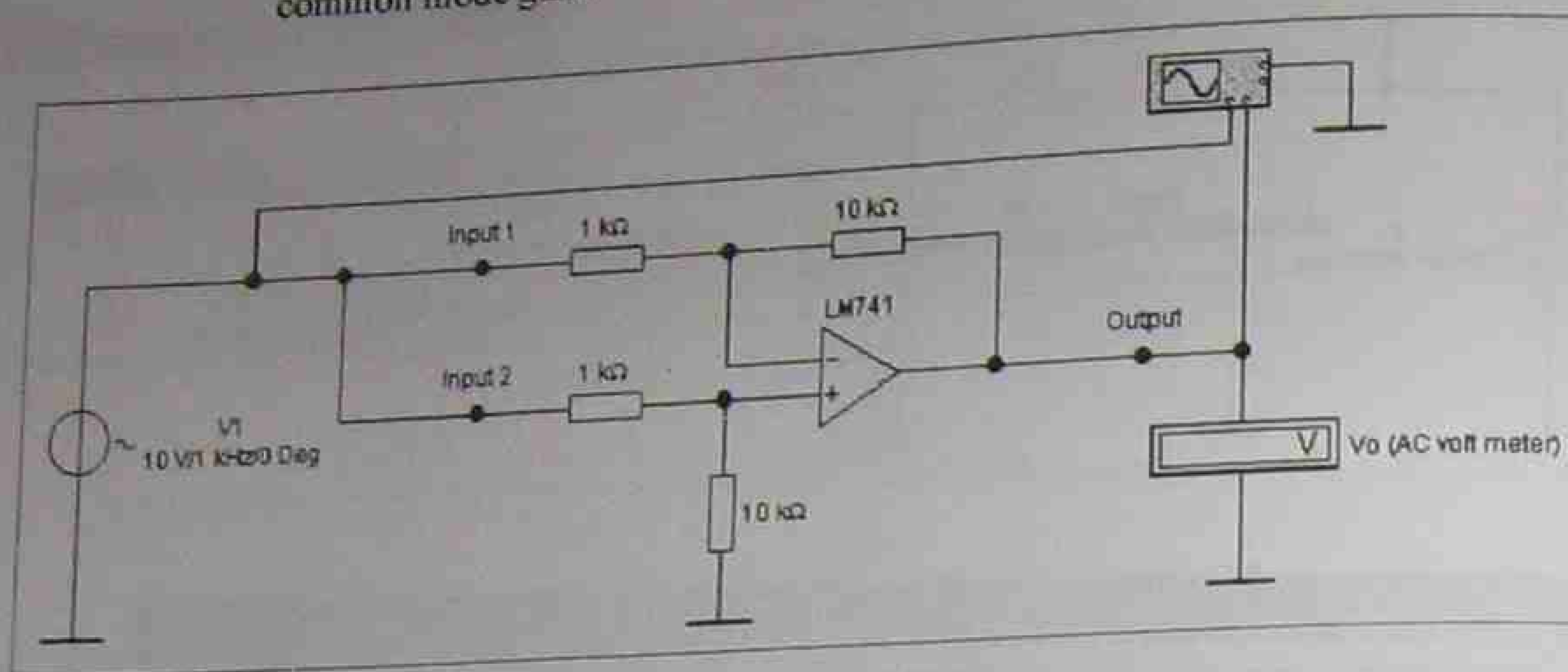


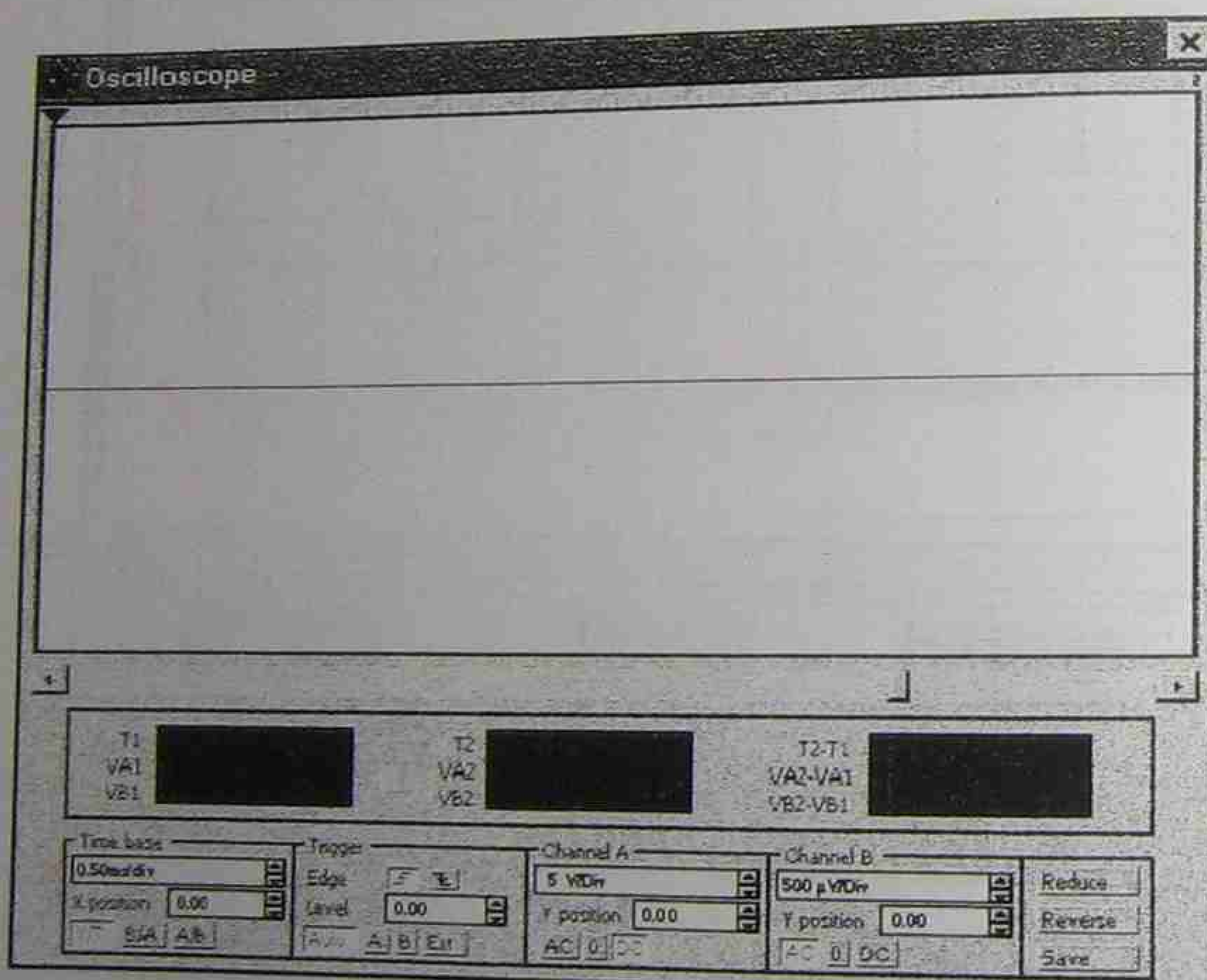
$V_O =$

$A_d =$

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.

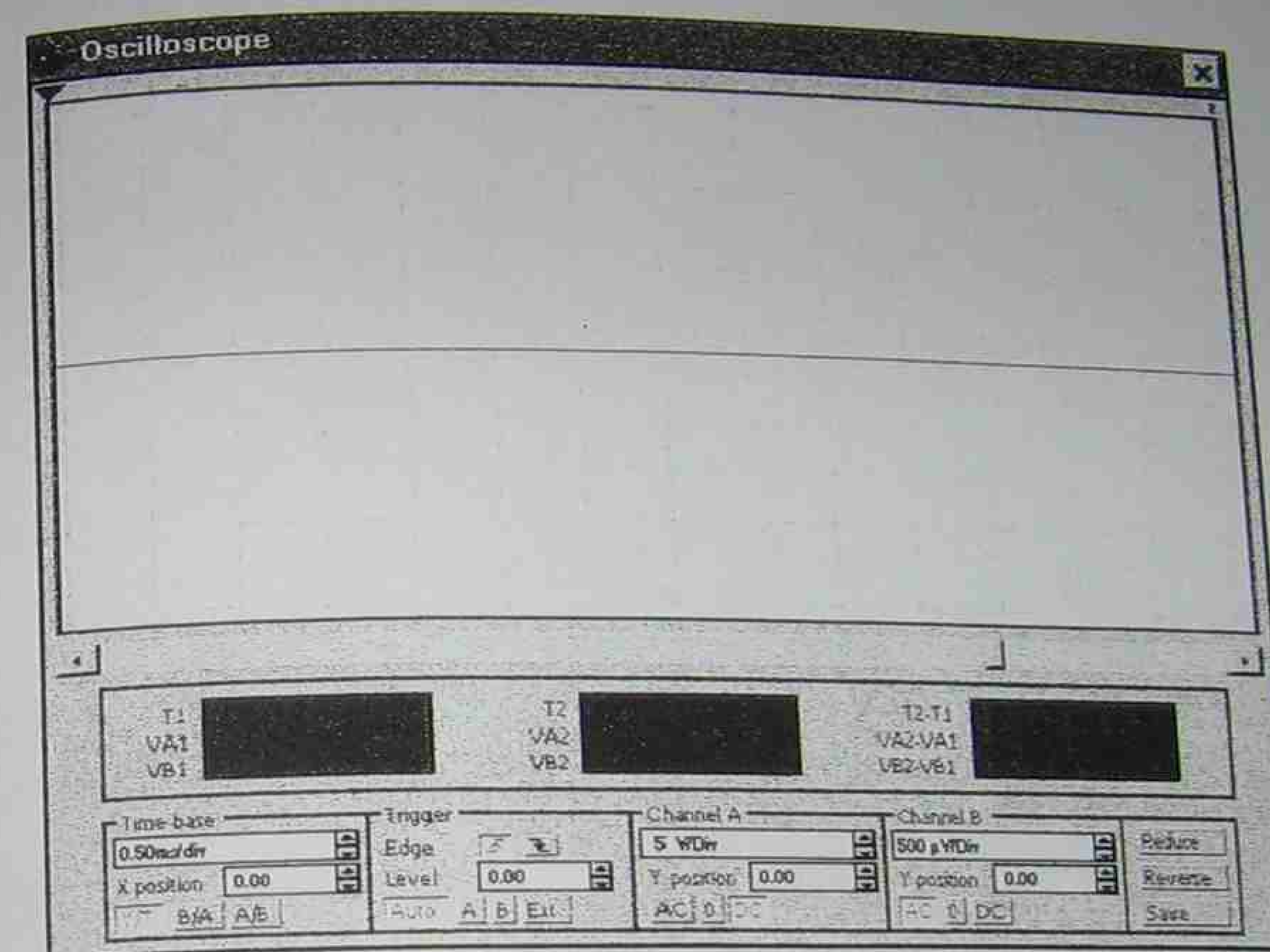




$V_O =$

Common mode gain $A_C =$

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



$V_O =$

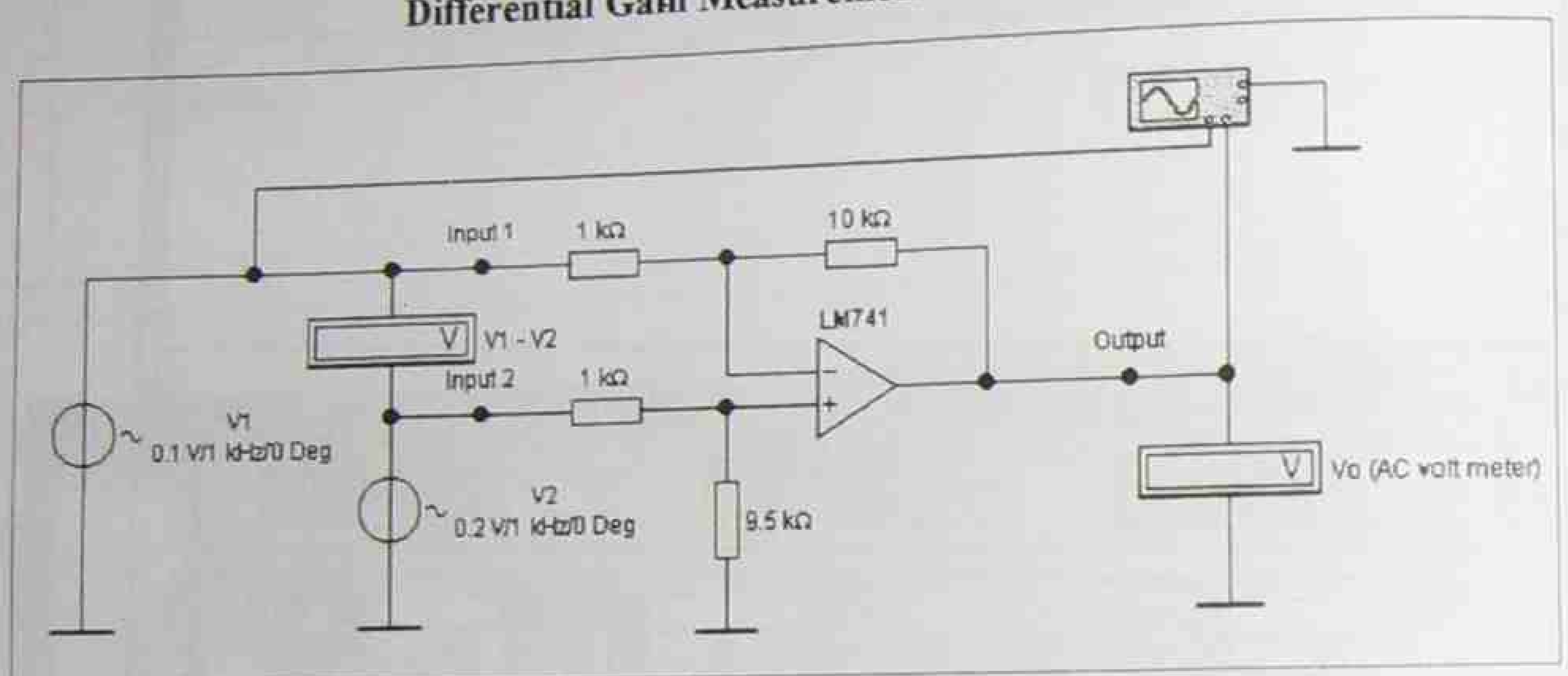
$A_C =$

Common Mode Rejection Ratio (CMRR)

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

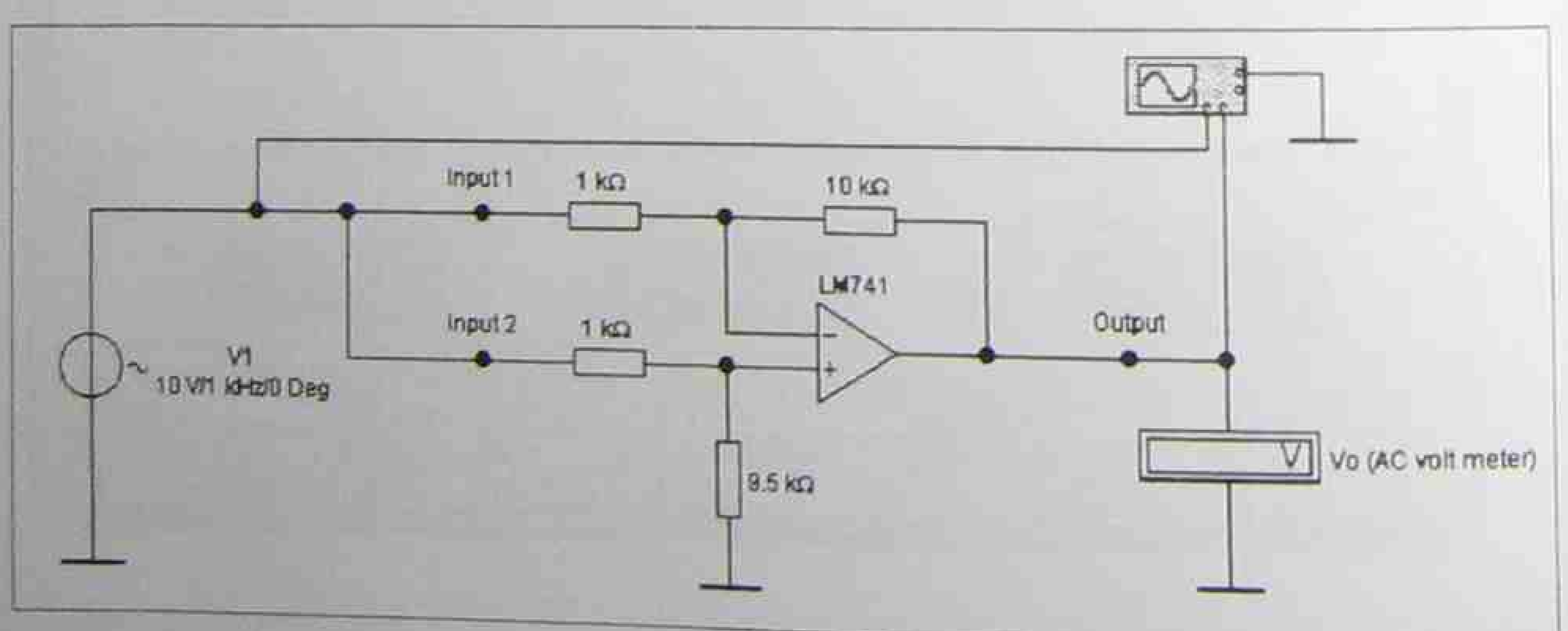
2. Replace R4 with 9.5kΩ and measure the CMRR (dB).

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.

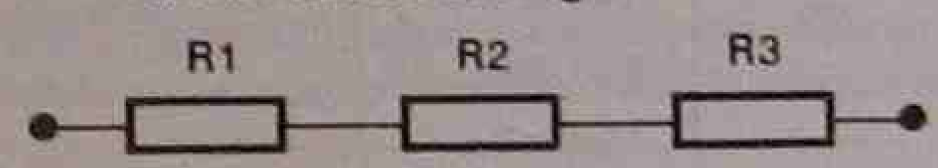
1. INTRODUCTION

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.

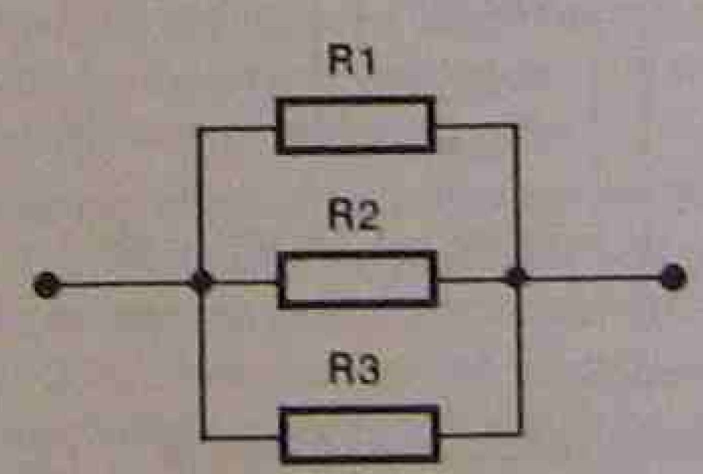
(a) The Series Circuit: The total resistance of a series circuit equals the sum of the individual resistor values. See Fig.1.



☐ $R_T = R1 + R2 + R3$

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.



☐ $1/R_T = 1/R1 + 1/R2 + 1/R3$

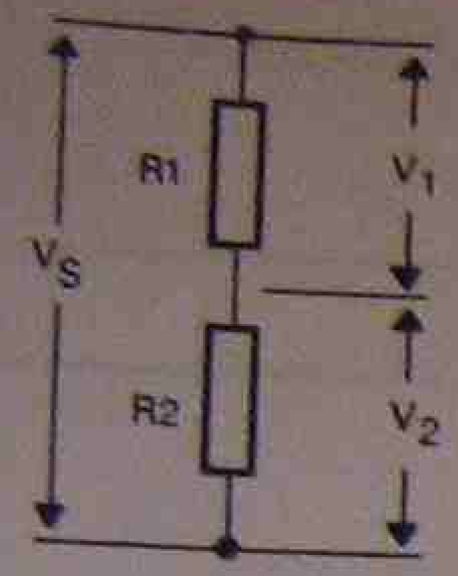
☐ $R_T = \dfrac{1}{1/R1 + 1/R2 + 1/R3}$

☐ $R_T = \dfrac{R1\,R2}{R1 + R2}$ (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%

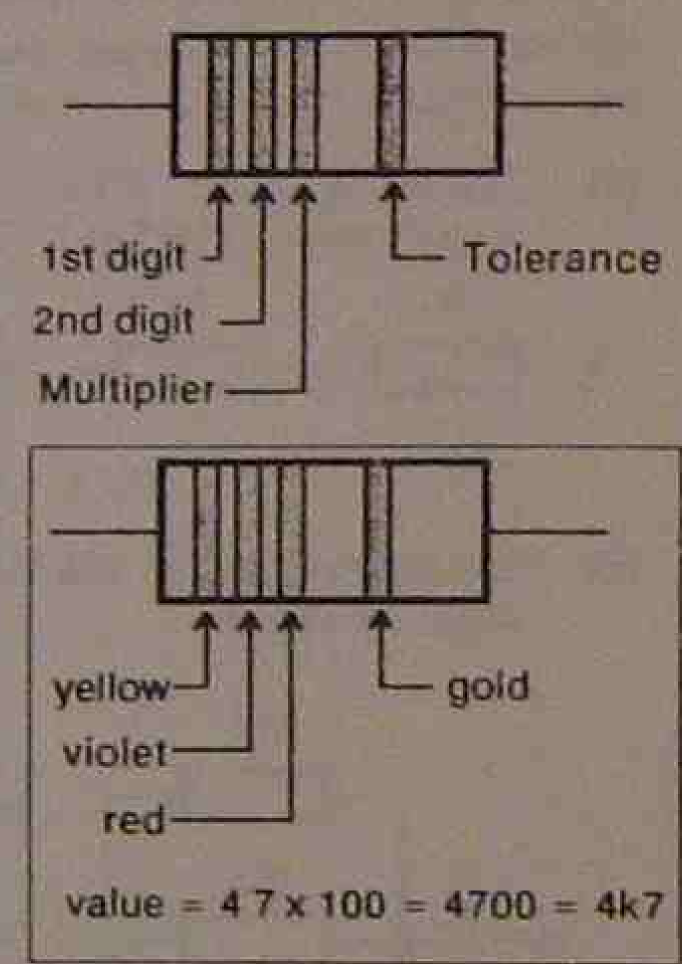


Fig.4: *The resistor colour code*

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.

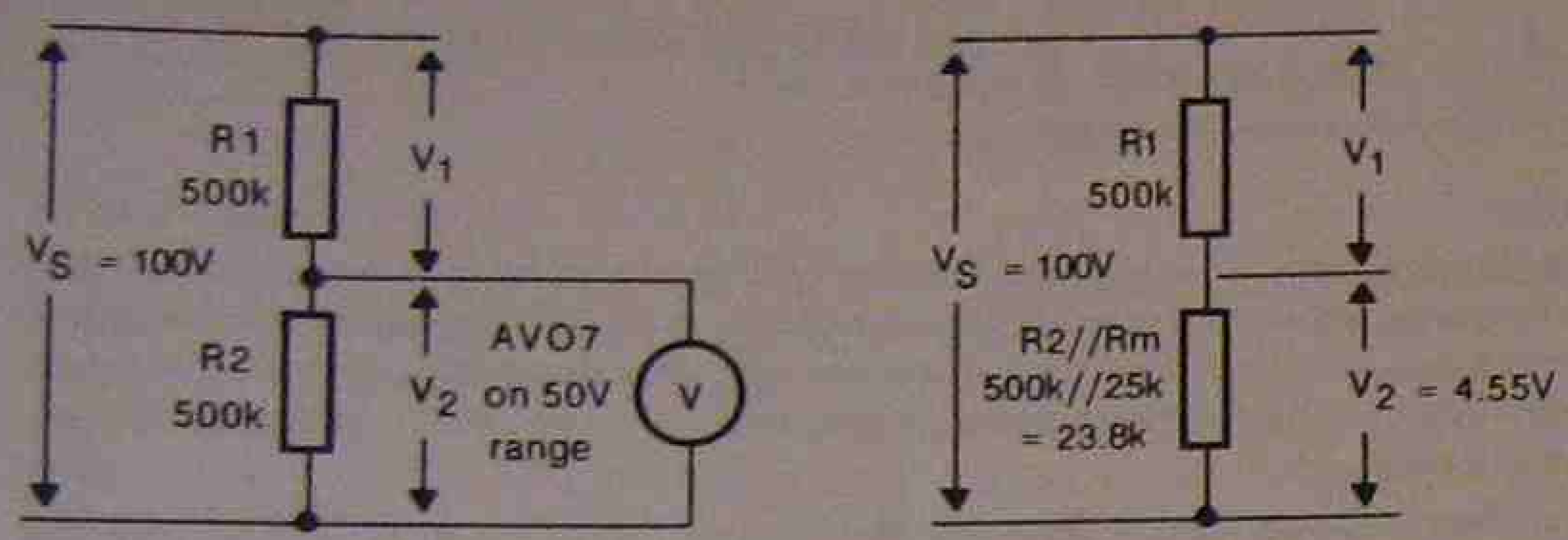
AVO7	=500 ohm/volt	(FSD current = 2mA)	On 10V range, equals a 5k resistor
AVO8	=20k ohm/volt	(FSD current = 50uA)	On 10V range, equals a 200k resistor
Arix	=100k ohm/volt	(FSD current = 10uA)	On 10V range, equals a 1M resistor
Digital	=11M ohm	(fixed resistance)	

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.

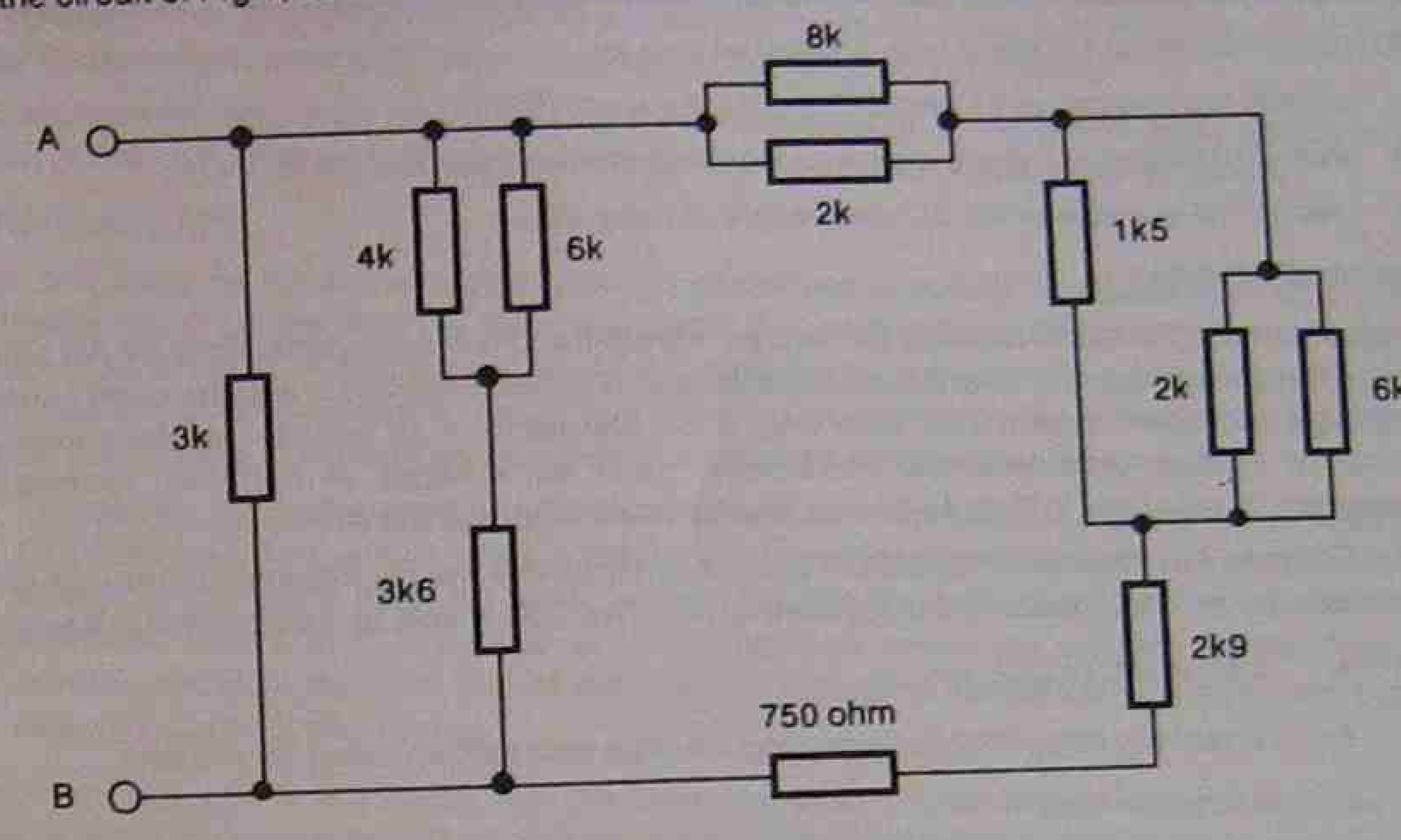


Fig.1

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

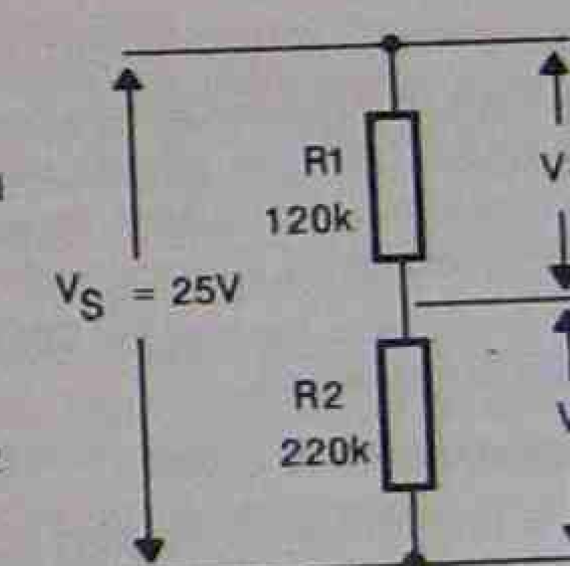


Fig.2

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.

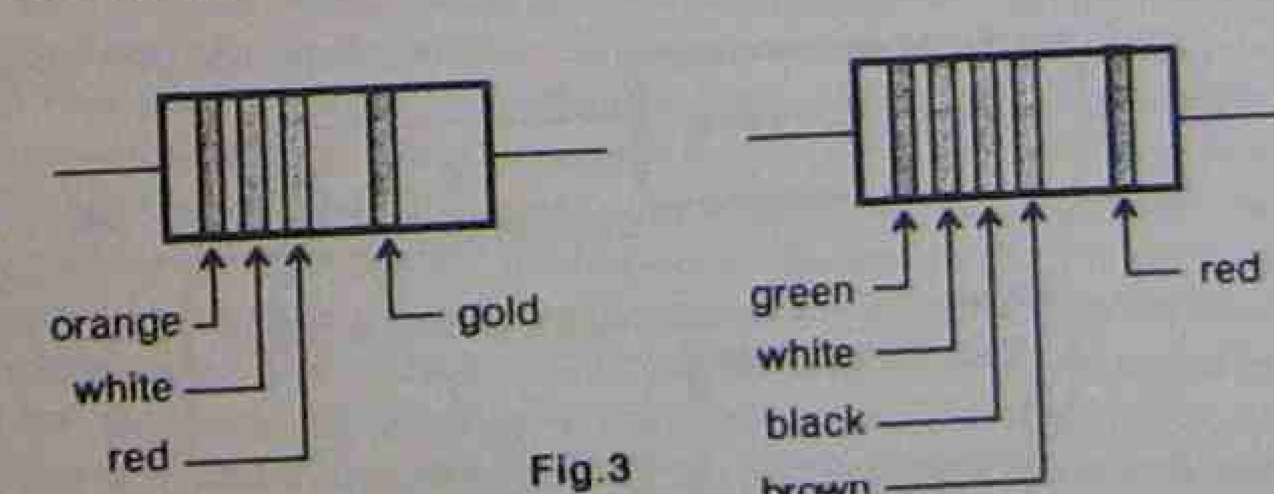


Fig.3

****

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.

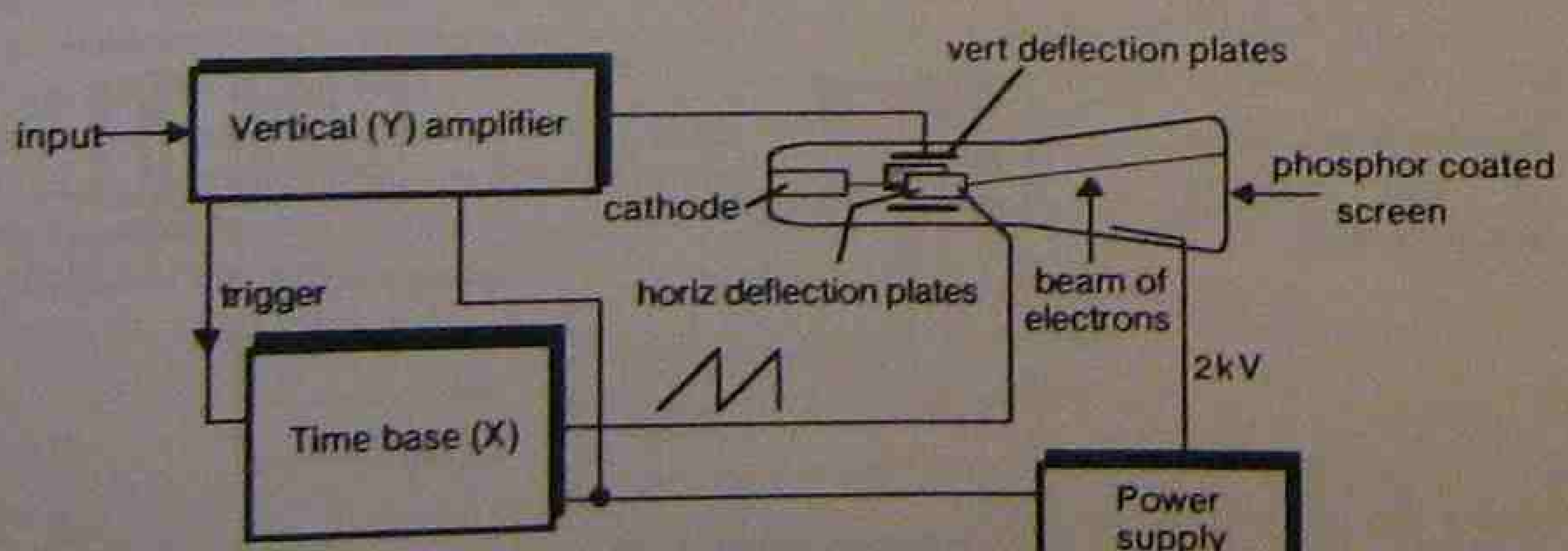


Fig.1: Block diagram of a CRO

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.

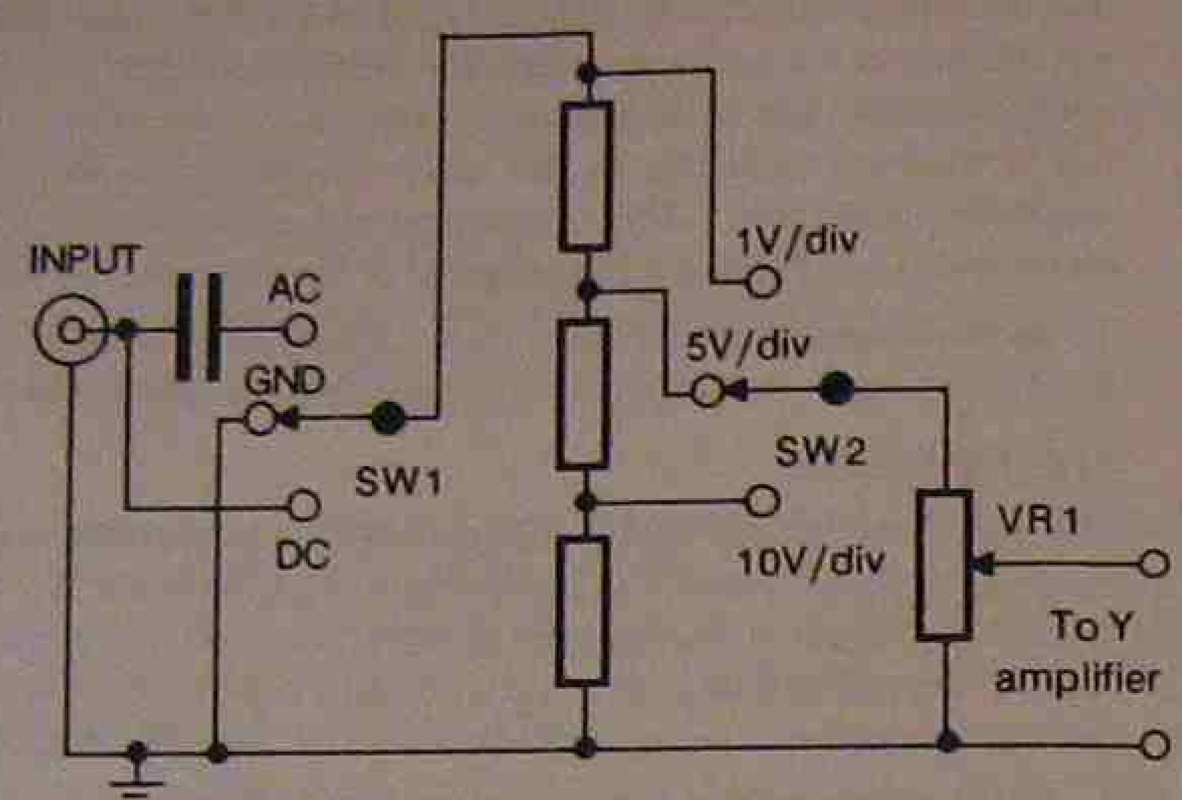


Fig.2: Input attenuators

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)

$V_{RMS} = 0.71V_{max}$

$V_{average} = V_{DC} =$ 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.

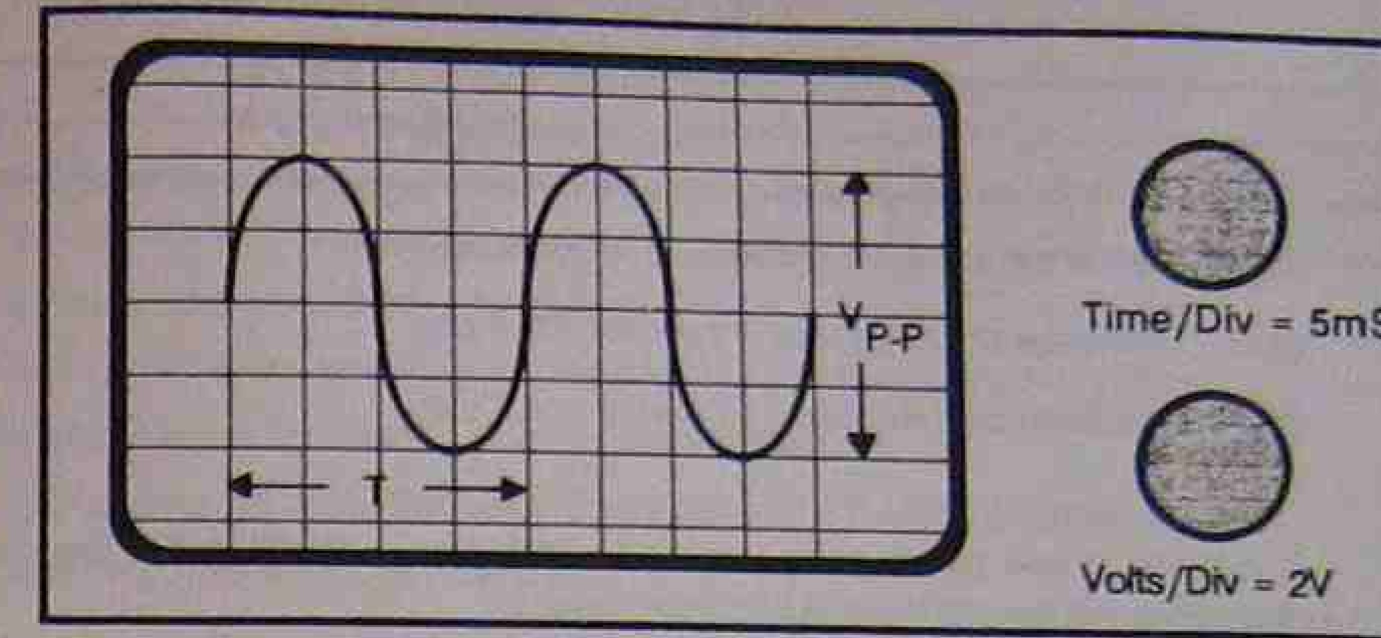


Fig.3: *Measuring period and peak to peak voltage on a CRO*

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.

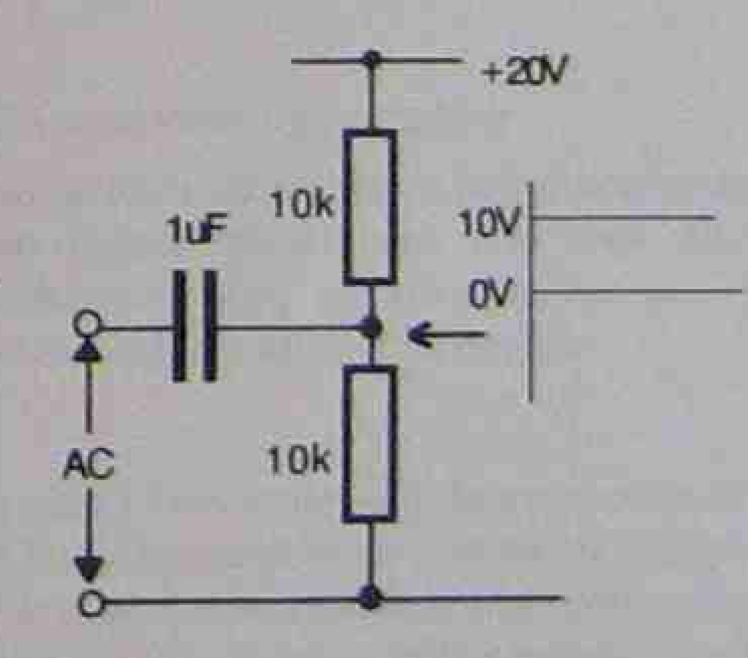


Fig.4: *AC combined with DC*

****

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.

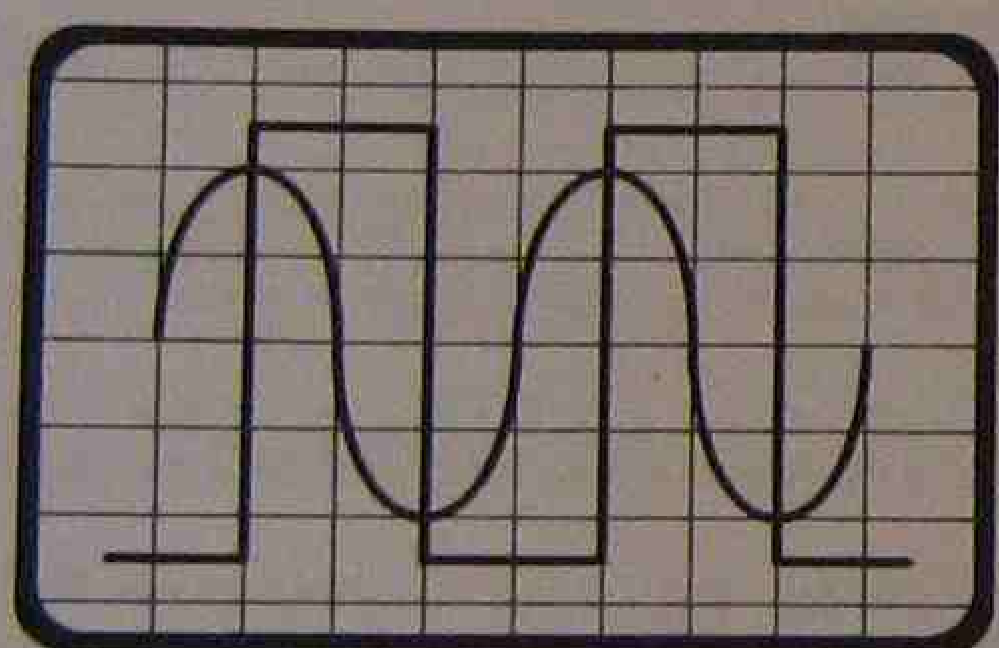


Fig.1

****

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.

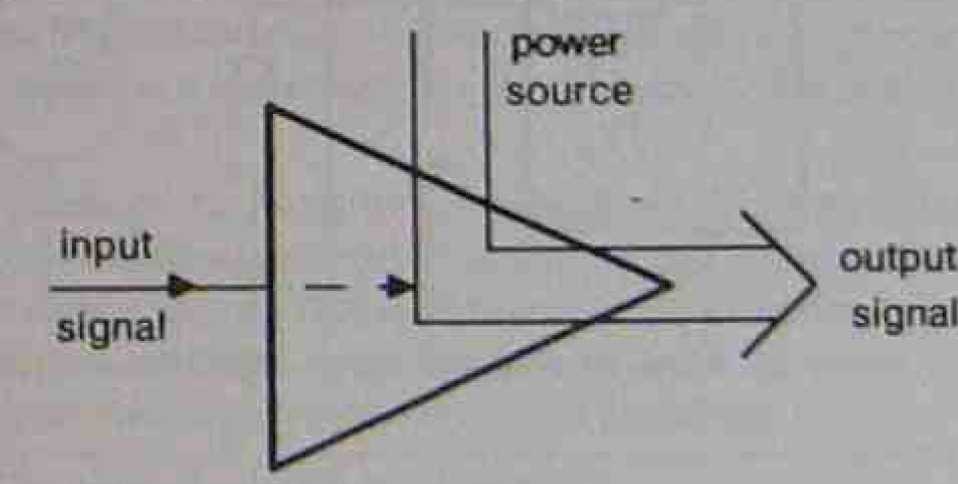


Fig.1: General form of an amplifier

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.

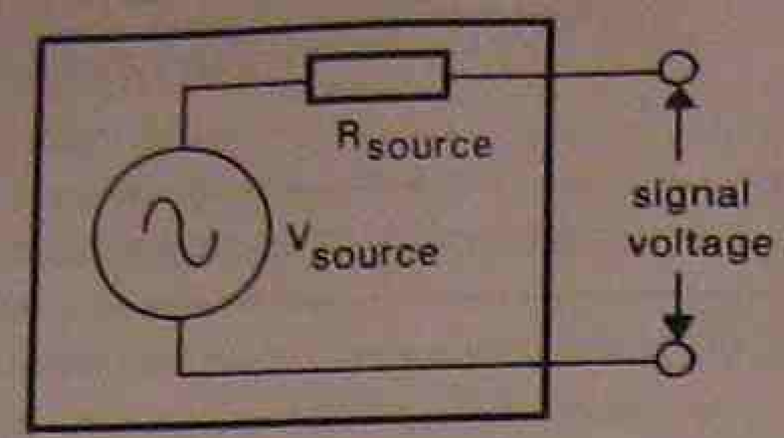


Fig.2: Equivalent circuit of a signal source

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.

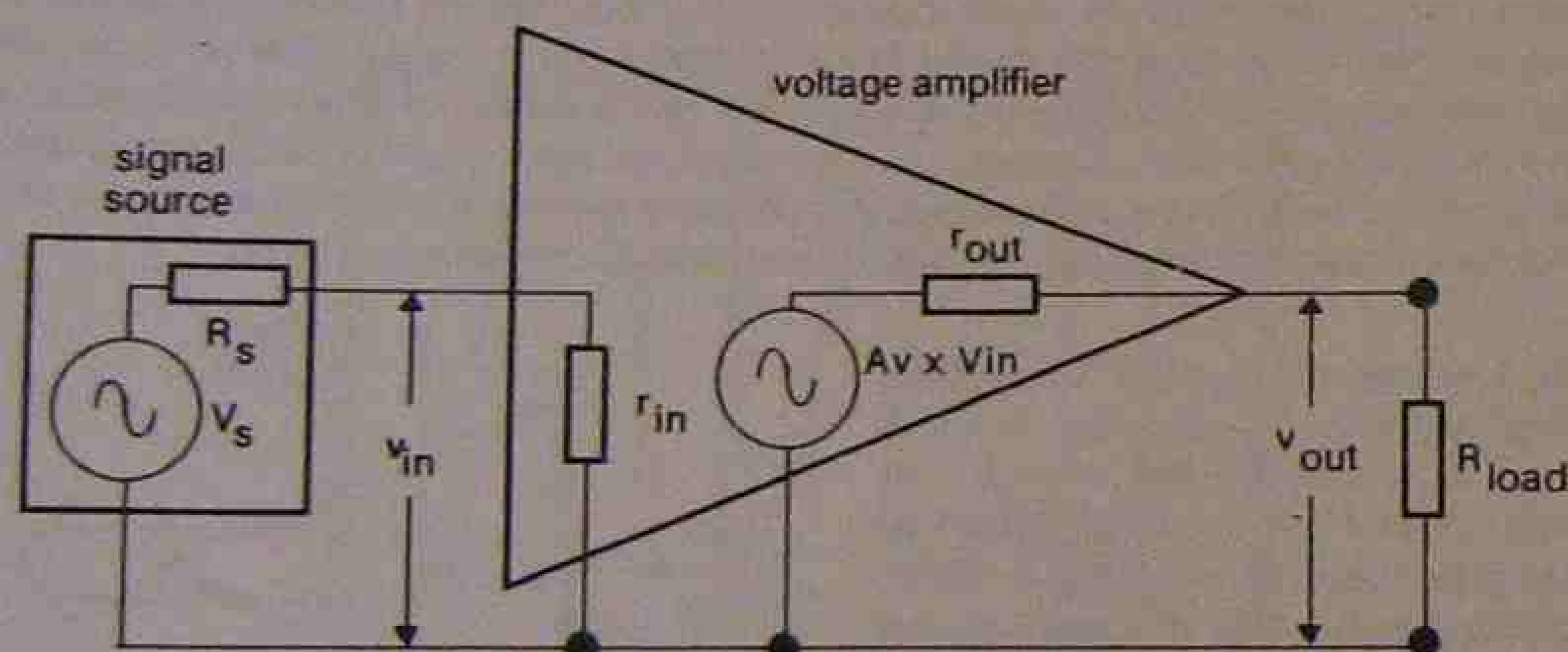


Fig.3: Equivalent circuit of a voltage amplifier

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.

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

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:

☐ $rin = \frac{Vin}{Iin}$

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.

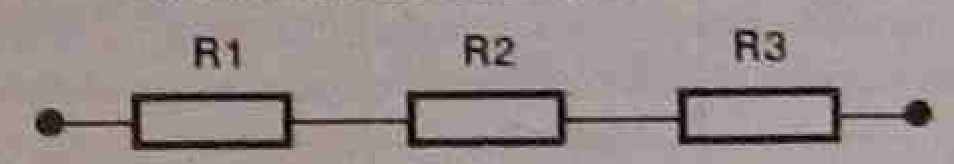
1. INTRODUCTION

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.

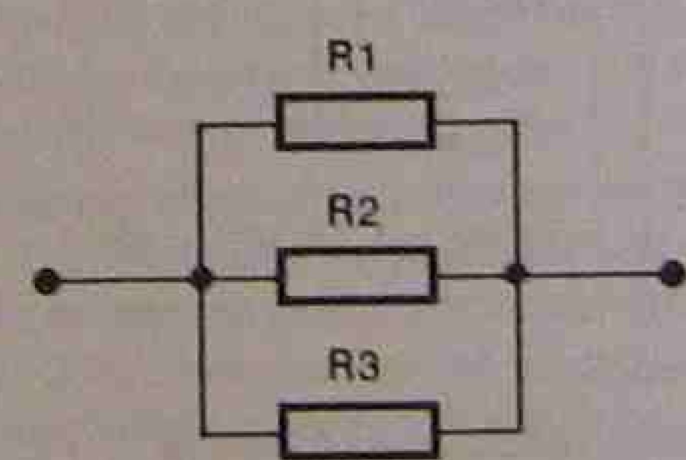
(a) The Series Circuit: The total resistance of a series circuit equals the sum of the individual resistor values. See Fig.1.



☐ $R_T = R1 + R2 + R3$

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.



☐ $1/R_T = 1/R1 + 1/R2 + 1/R3$

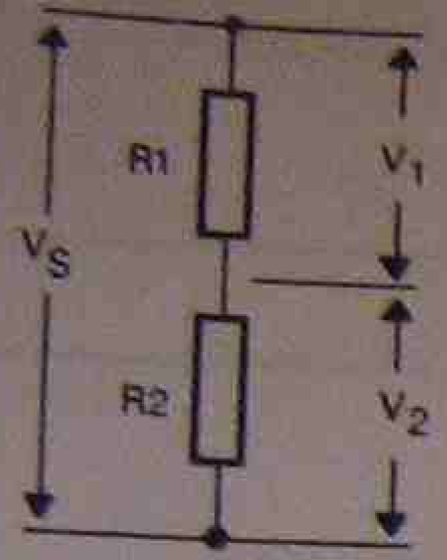
☐ $R_T = \dfrac{1}{1/R1 + 1/R2 + 1/R3}$

☐ $R_T = \dfrac{R1\,R2}{R1 + R2}$ (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



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

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

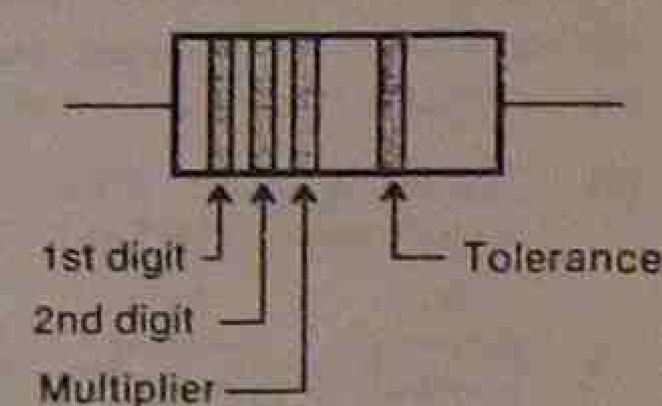
<u>NOTE</u>: $\frac{V_s}{R1 + R2}$ = current in R1 & R2

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%



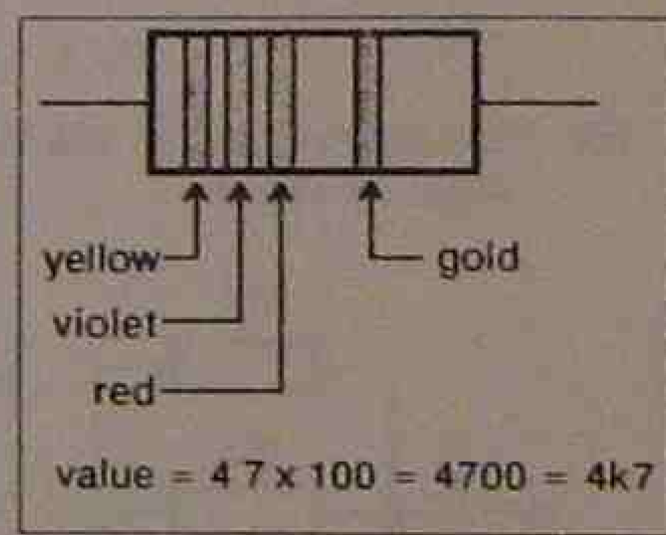


Fig.4: *The resistor colour code*

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 x 10[illegible]
0.0000047 = 4.7 x 10[illegible]

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.

<u>Table of metric units</u>

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**

<u>Analog Multimeters</u> 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.

<u>Digital Multimeters</u> 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.

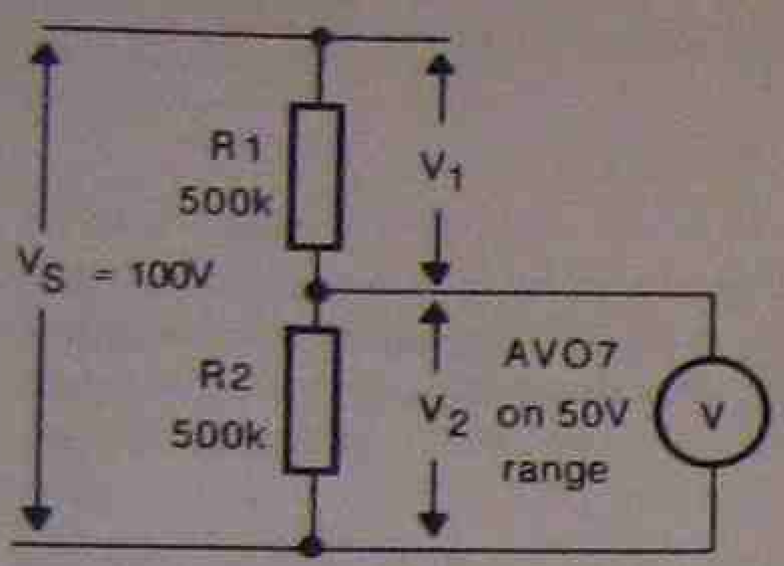
AVO7	=500 ohm/volt	(FSD current = 2mA)	On 10V range, equals a 5k resistor
AVO8	=20k ohm/volt	(FSD current = 50uA)	On 10V range, equals a 200k resistor
Arix	=100k ohm/volt	(FSD current = 10uA)	On 10V range, equals a 1M resistor
Digital	=11M ohm	(fixed resistance)	

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.

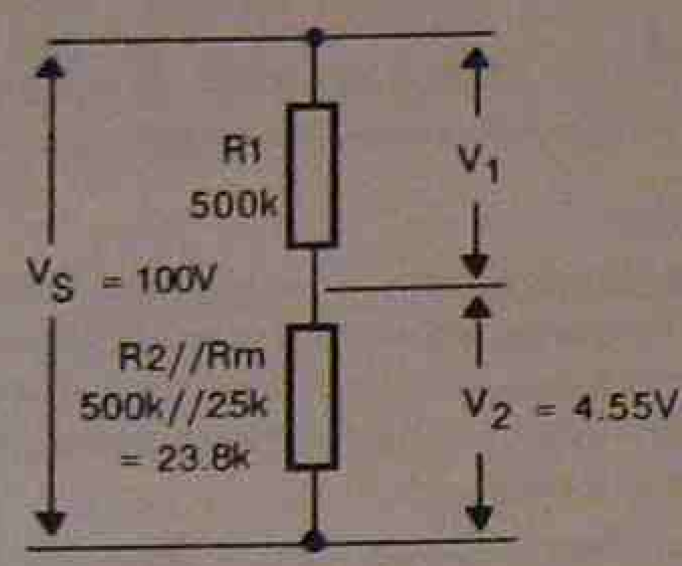
7. **Circuit Loading**

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.

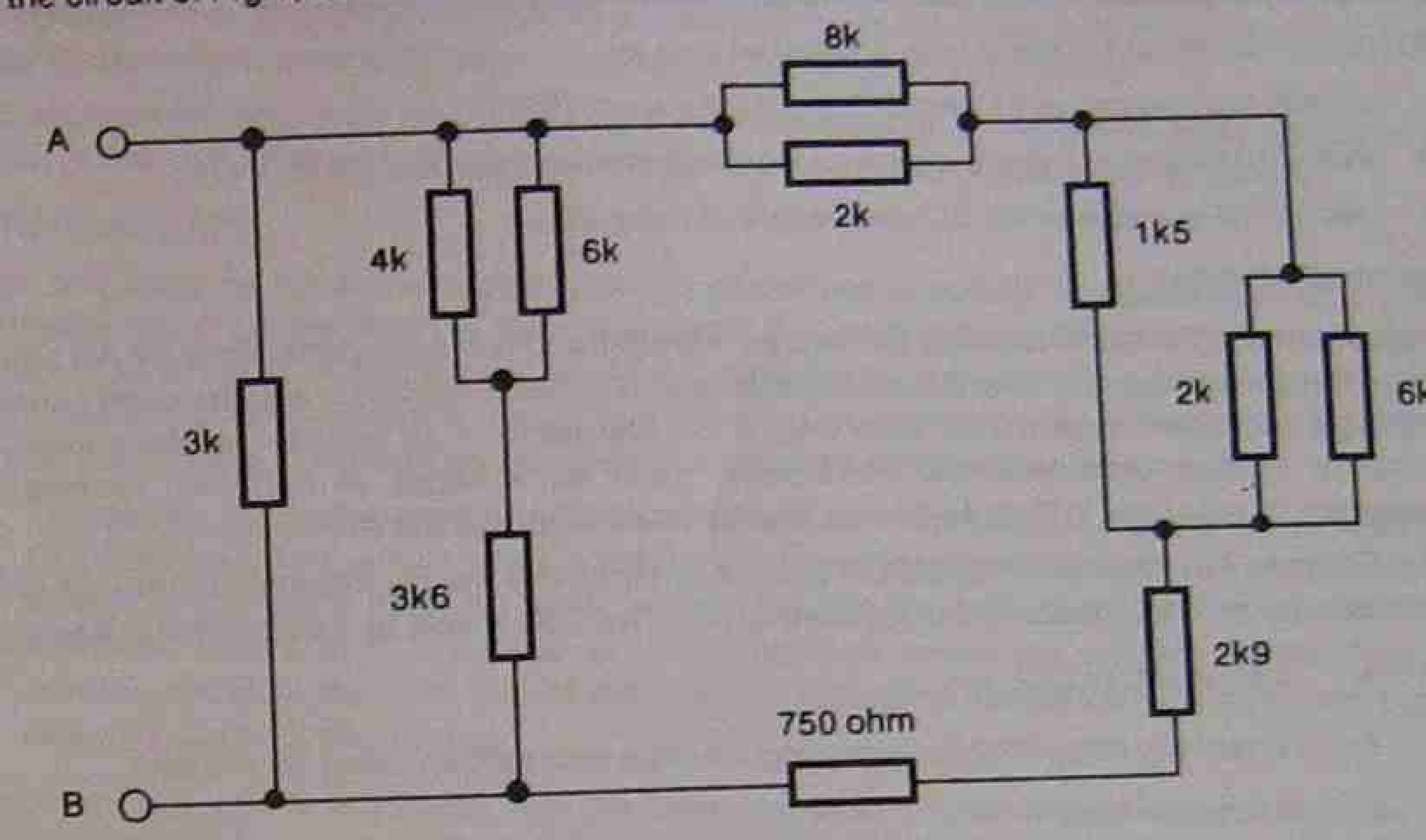


Fig.1

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

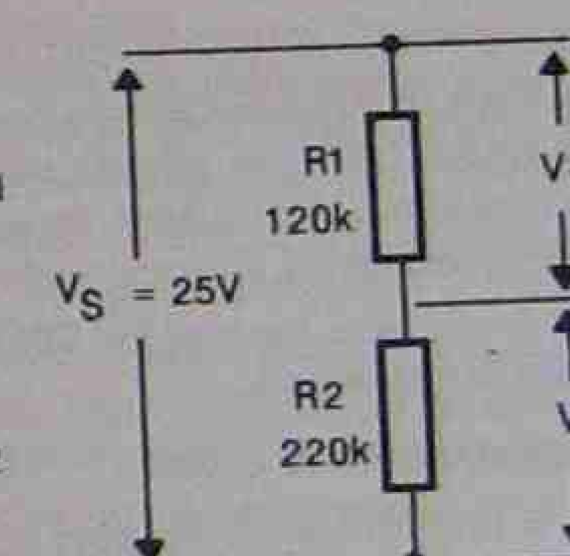


Fig.2

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.

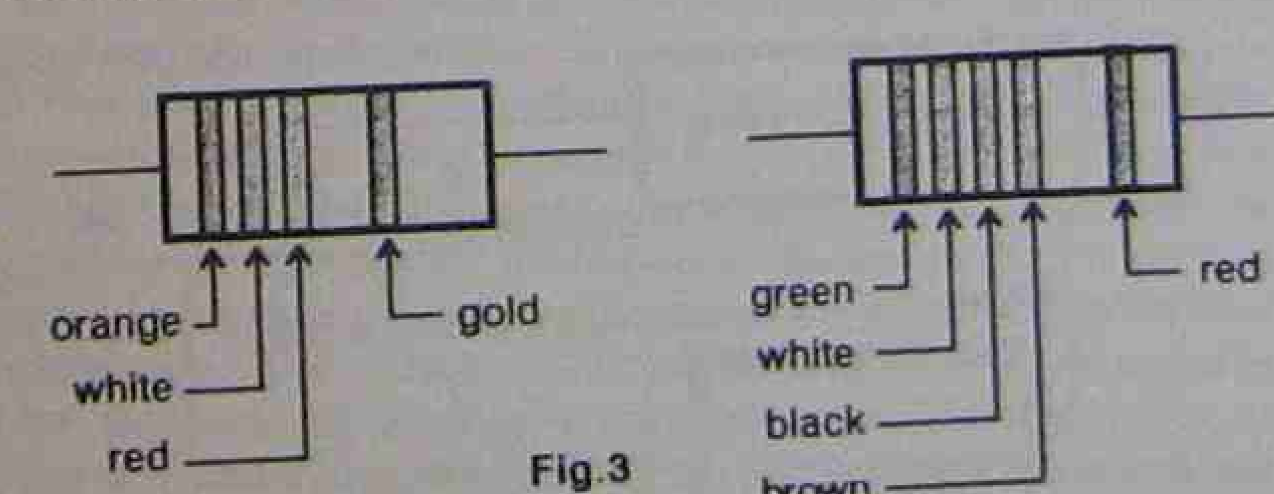


Fig.3

****

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.

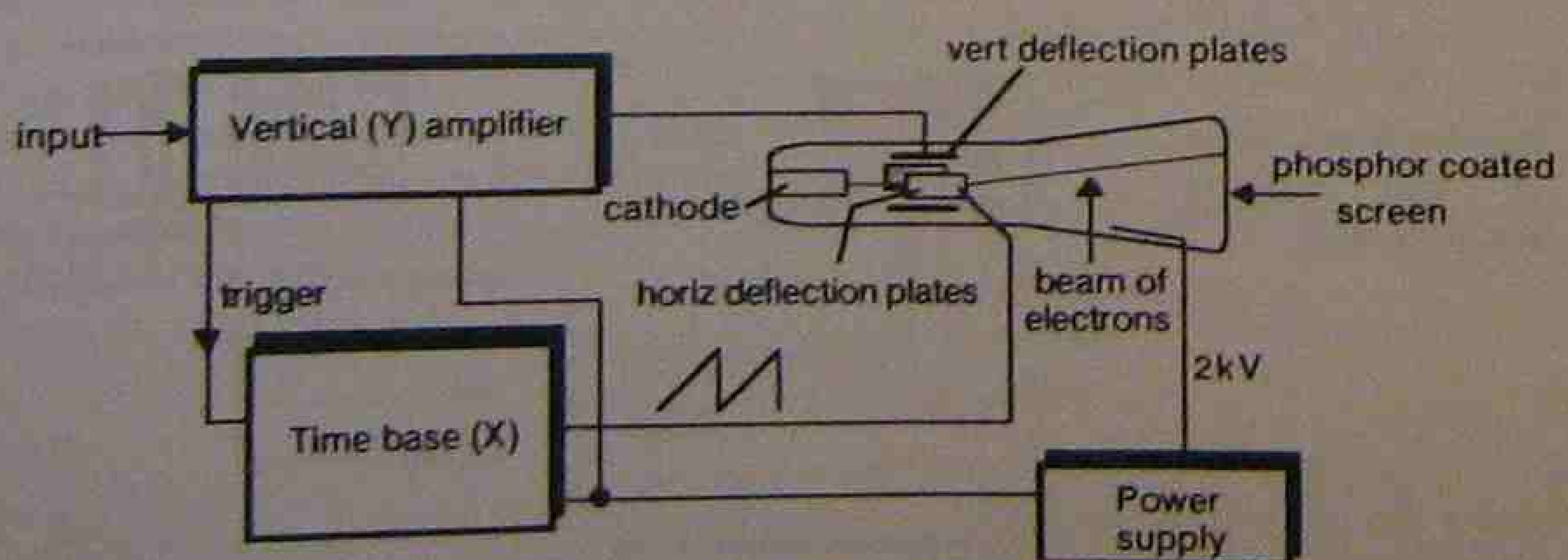


Fig.1: Block diagram of a CRO

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.

- <u>The sweep generator</u> 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.

- <u>The horizontal amplifier</u> 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.

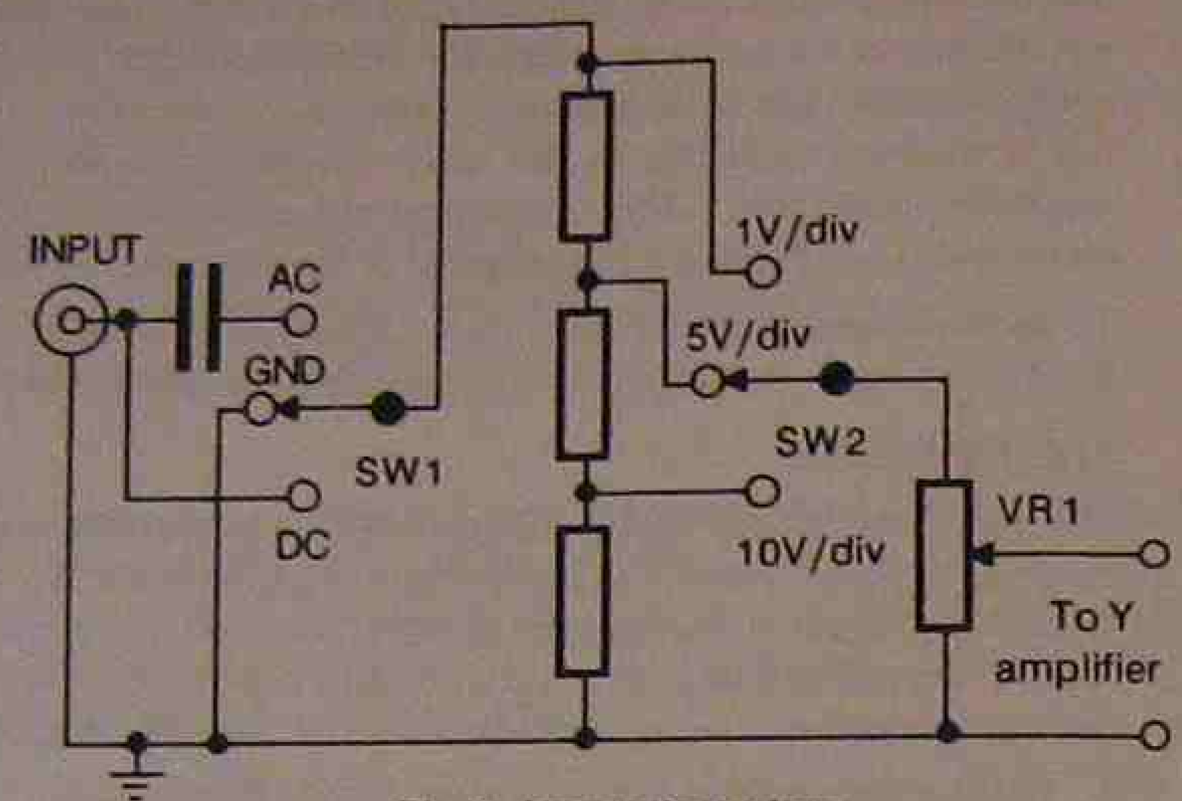


Fig.2: Input attenuators

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.

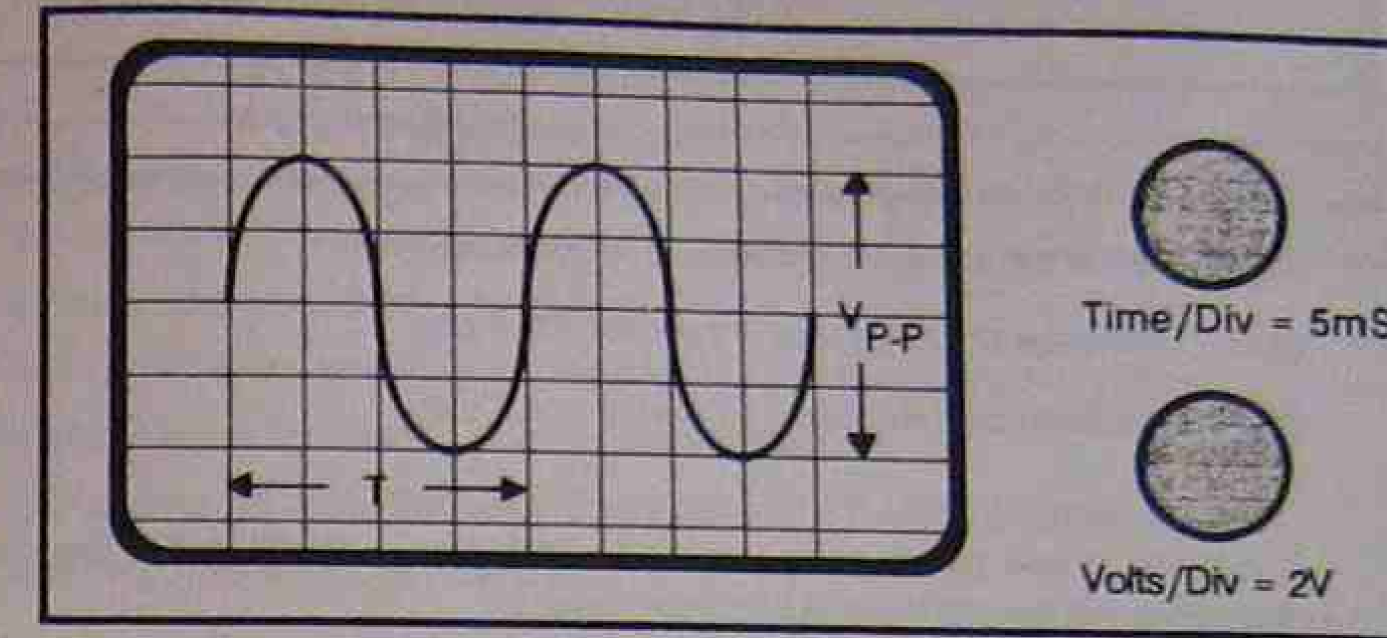


Fig.3: *Measuring period and peak to peak voltage on a CRO*

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.

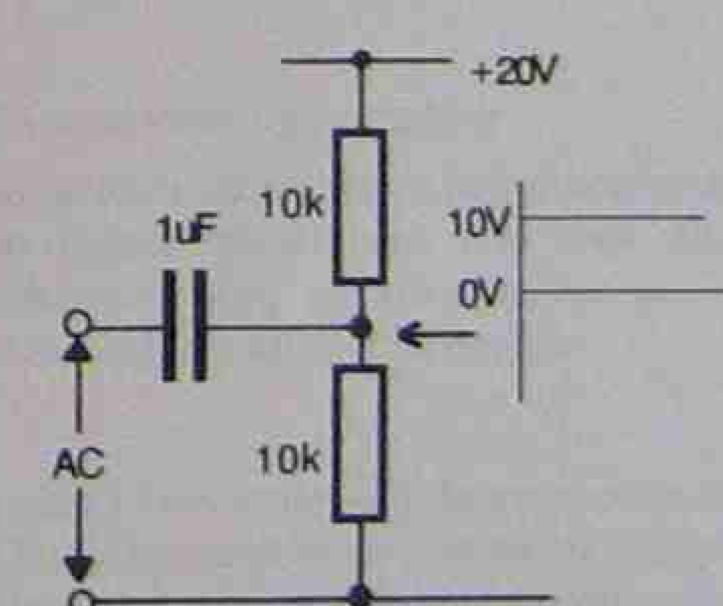


Fig.4: *AC combined with DC*

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES

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.

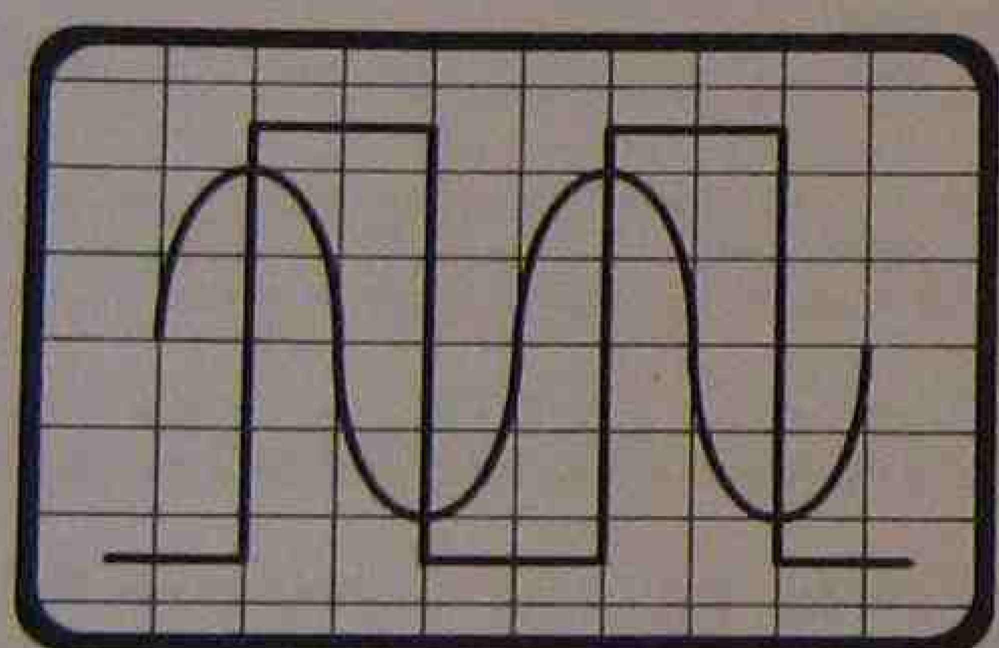


Fig.1

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES 6016A

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.

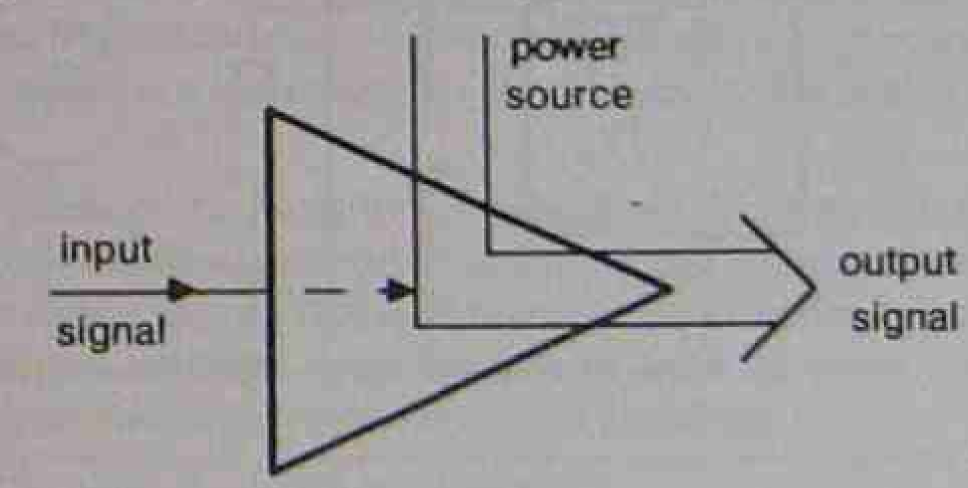


Fig.1: General form of an amplifier

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.

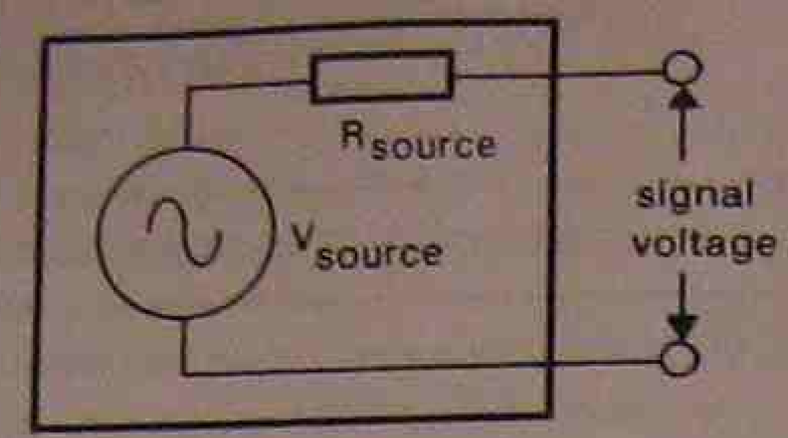


Fig.2: Equivalent circuit of a signal source

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.

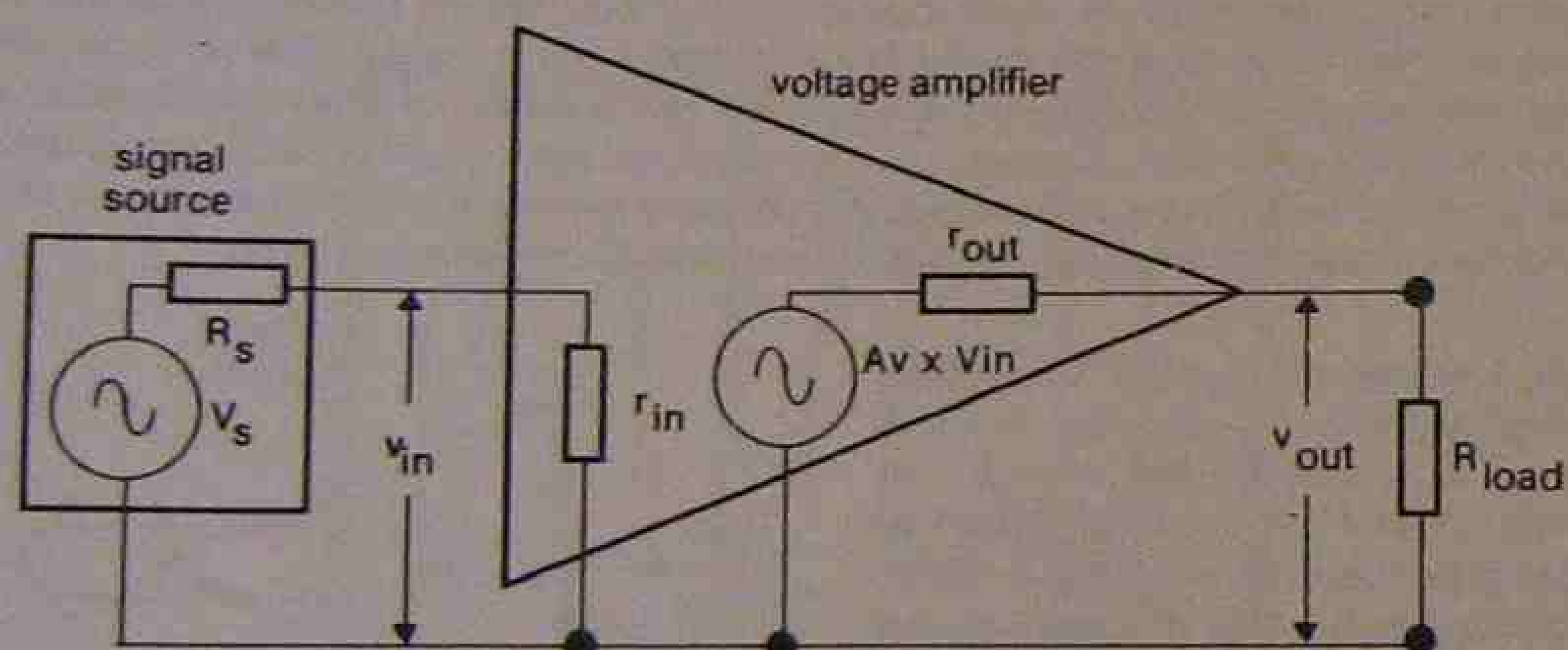


Fig.3: Equivalent circuit of a voltage amplifier

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:

☐ $rin = \frac{Vin}{Iin}$

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.

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.

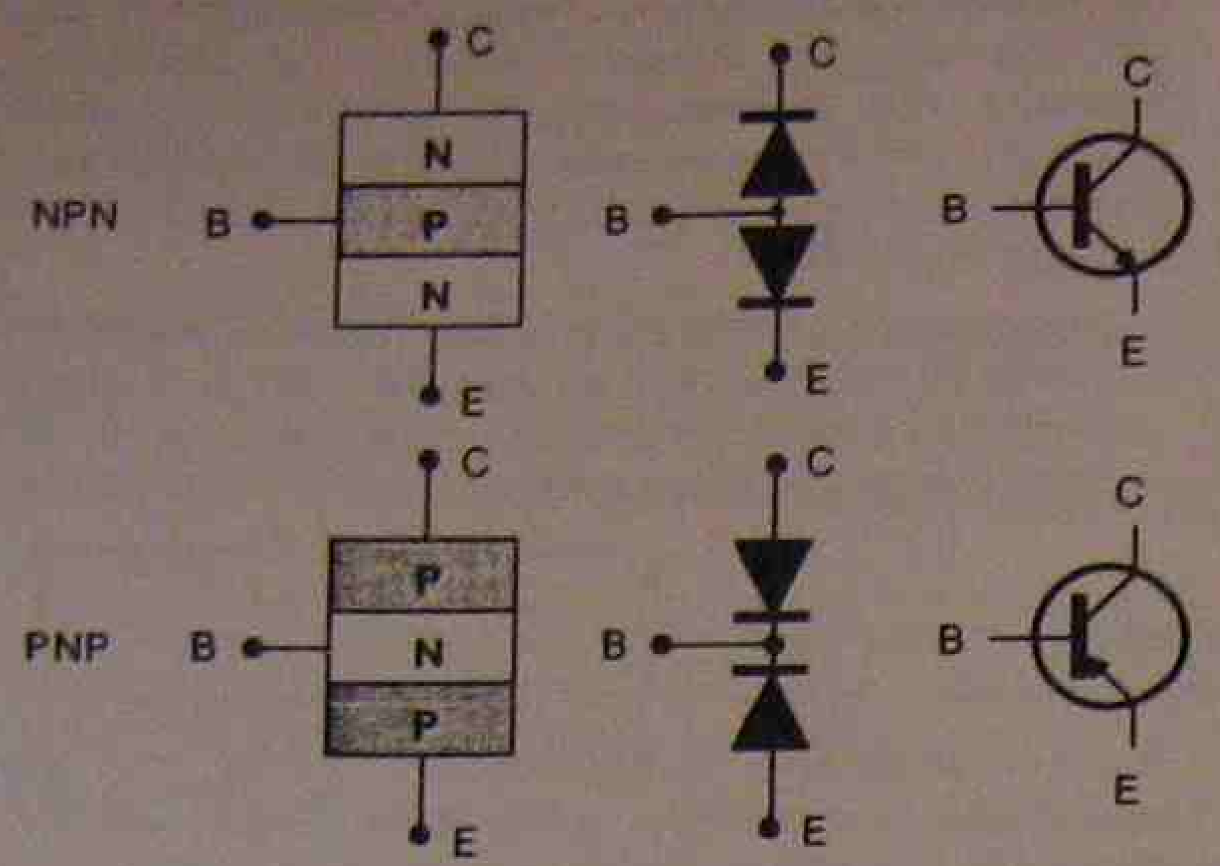


Fig.4 (a) construction (b) diode equivalent (c) schematic symbol

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.

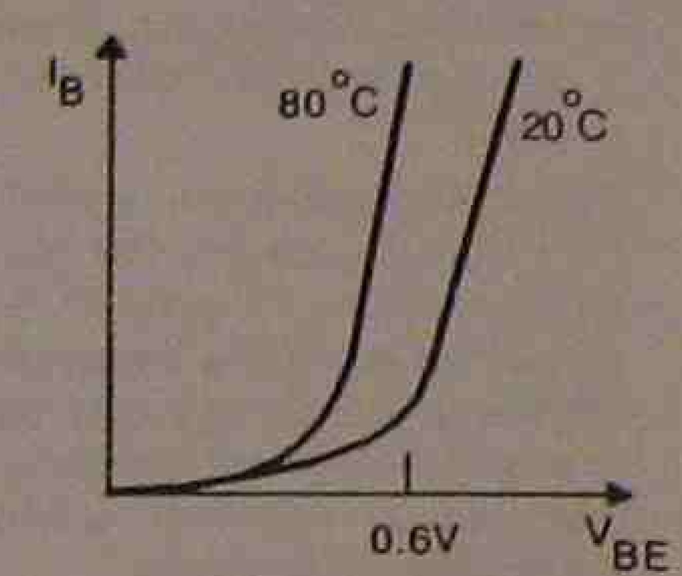


Fig.5: Effects of temperature on I_B.

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	2N408
AC127	TO-1	NG	18	32	500			105	50	1.5	10	340	Audio O/P	AC187
AC128	TO-1	PG	16	32	1000	0.C	1A	80-175	300	1	10	1W	Audio O/P	AC188
AC132	TO-1	PG	12	32	200	0.35	200	115	50	1.3	10	218	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	20	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	20	25	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	0.6	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	BC109C, 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	80	80	4A	2.5	1.5A	750	1.5A	7	1.5A	36W	High Gain Darl. O/P	BD266
BD263	TO-126	NS	80	80	4A	2.5	1.5A	750	1.5A	7	1.5A	36W	High Gain Darl. O/P	BD267
BD266A	TO-220	PS	80	100	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	20	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	30	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			20-80	30	130		3W	Video Amp	
BF337	TO-39	NS	200	300	100			20-80	30	130		3W	Video Amp	
BF338	TO-39	NS	225	250	100			20-80	30	130		3W	Video Amp	
BF463	TC-202	PS	35	250	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	50	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	

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 ELECTRONIC DEVICES

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 βI_B. 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 R_B supplies the bias voltage for the base-emitter and R_C converts the collector current variations to a voltage variation.

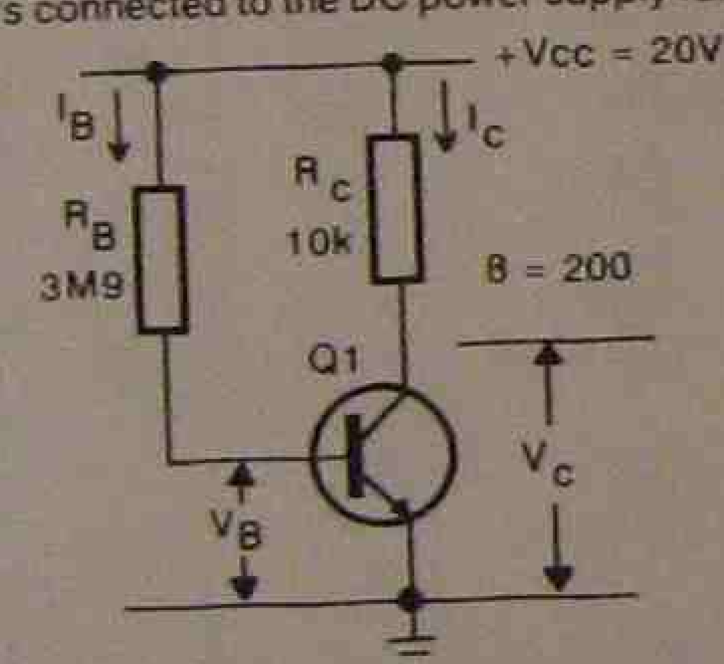


Fig.1: Basic transistor amplifier circuit

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:

- I_B = base current in Q1
- I_C = collector current (equals βI_B)
- V_B = base voltage (measured from base terminal to ground)
- V_C = collector voltage (measured from collector to ground)
- V_{CC} = 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) $I_B = \dfrac{V_{CC} - V_{BE}}{R_B} = \dfrac{20 - 0.6}{3M9} = \underline{5\mu A}$

(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 R_B, and the collector current that results is simply the base current multiplied by the DC current gain (β) of the transistor. The collector voltage (V_C) is determined by subtracting the voltage drop across the collector resistor R_C 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:

- <u>Temperature change</u>: 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.
- <u>Change in β</u>: A change in the current gain (β) 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 I_B 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 β 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 β 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 β 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:

- Ic = collector current (also equalsIE)
- VB = base voltage (from base to ground)
- Vbe = voltage between base & emitter (0.6V)
- Vc = collector voltage (collector to ground)
- VE = emitter voltage (emitter to ground)
- Vcc = value of the supply voltage

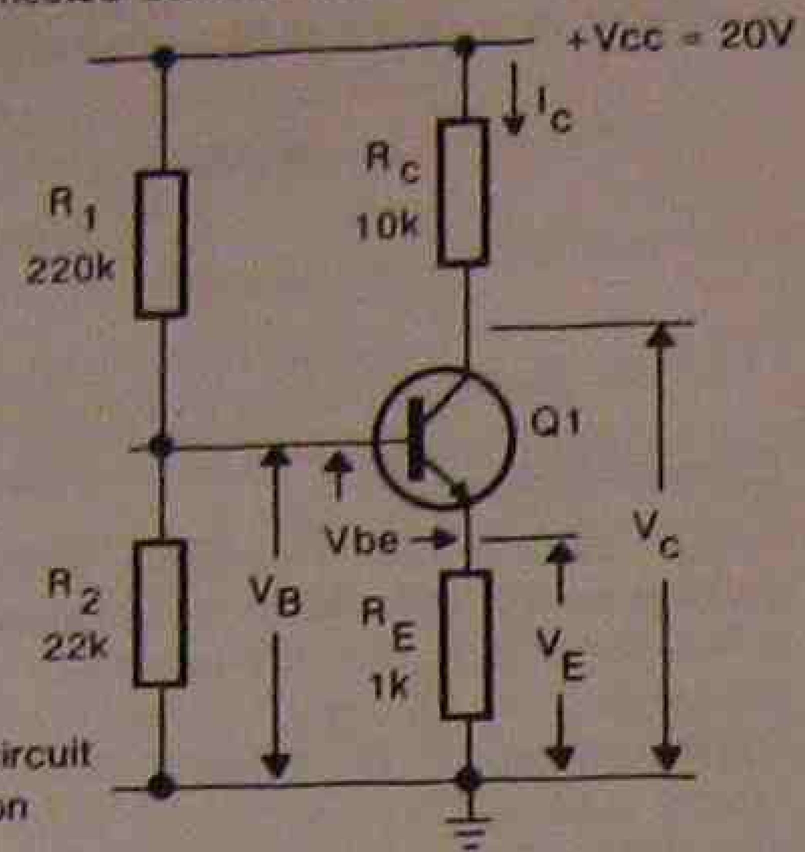


Fig.2: Potential divider bias circuit with emitter stabilisation

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 (VB) is determined by the potential divider formed by R1 and R2.

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

(b) The emitter voltage (VE) equals VB - Vbe

> □ $V_E = V_B - V_{be} = V_B - 0.6V$

(c) The emitter current equals the collector current which equals the emitter voltage (VE) divided by RE:

> □ $I_E = I_C = \frac{V_E}{R_E}$

(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 VB and on the feedback supplied by RE.

8. EFFECT OF EMITTER RESISTOR

In Fig.2, resistor RE 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 VB 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 RE rises because the emitter current has increased.

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

That is, RE 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 RE. The higher the value of RE the greater the stability, but the greater the circuit losses. The lower the value of RE, 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

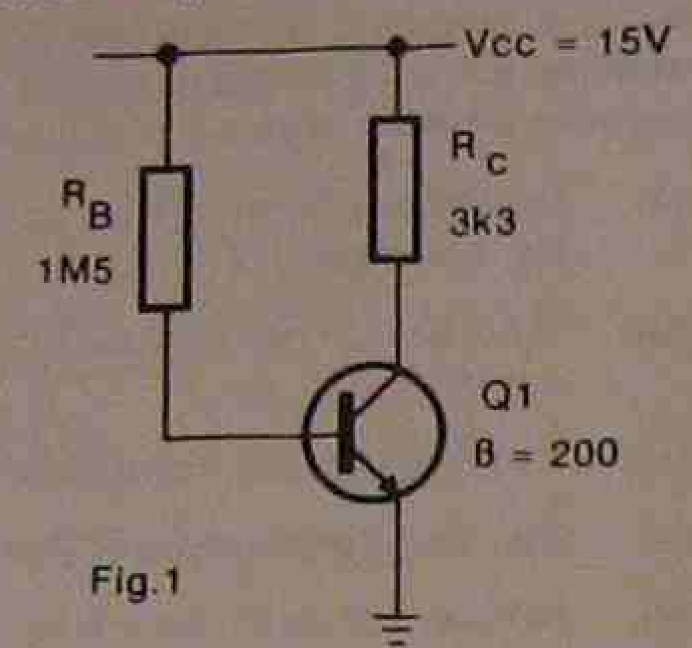


Fig.1

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.

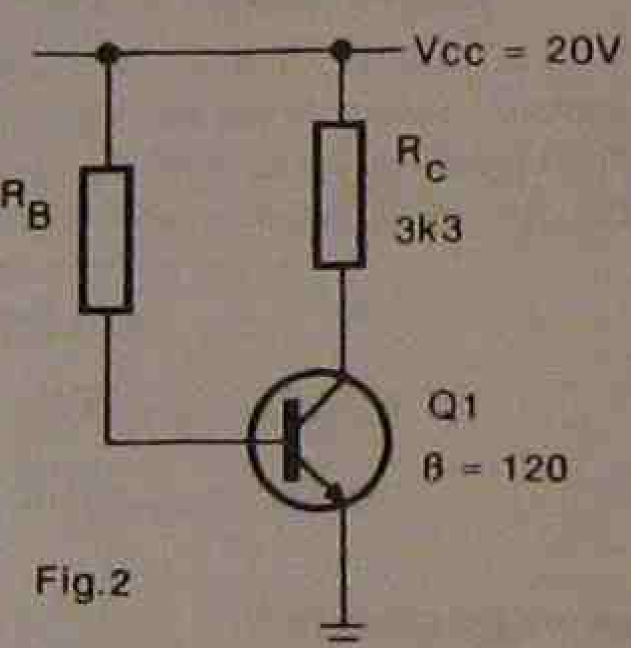


Fig.2

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

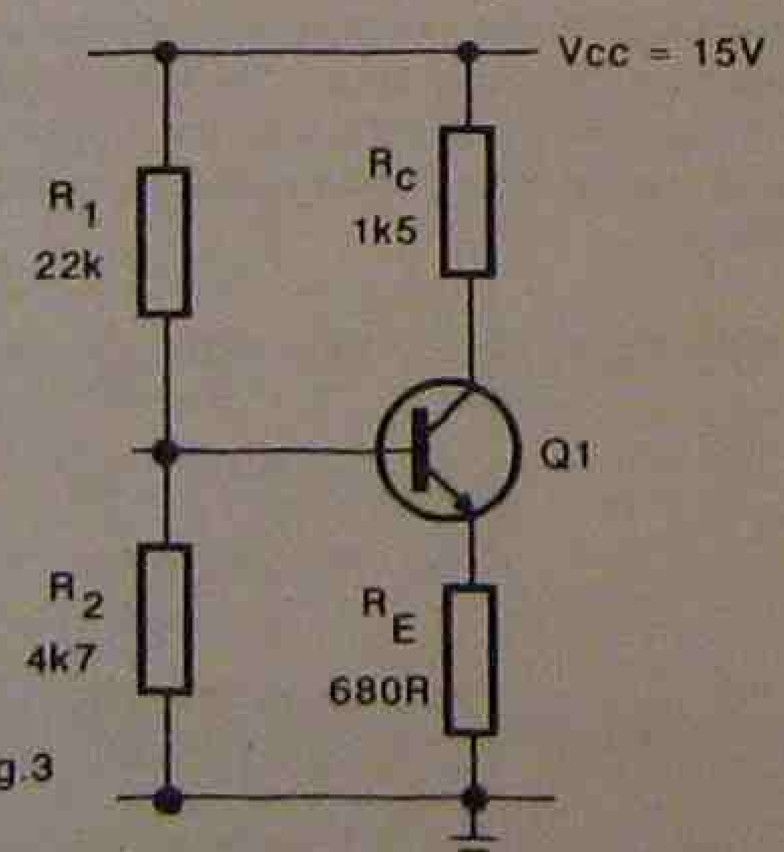


Fig.3

4. Give two reasons why the circuit of Fig.3 is more stable against changes in temperature and B 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.

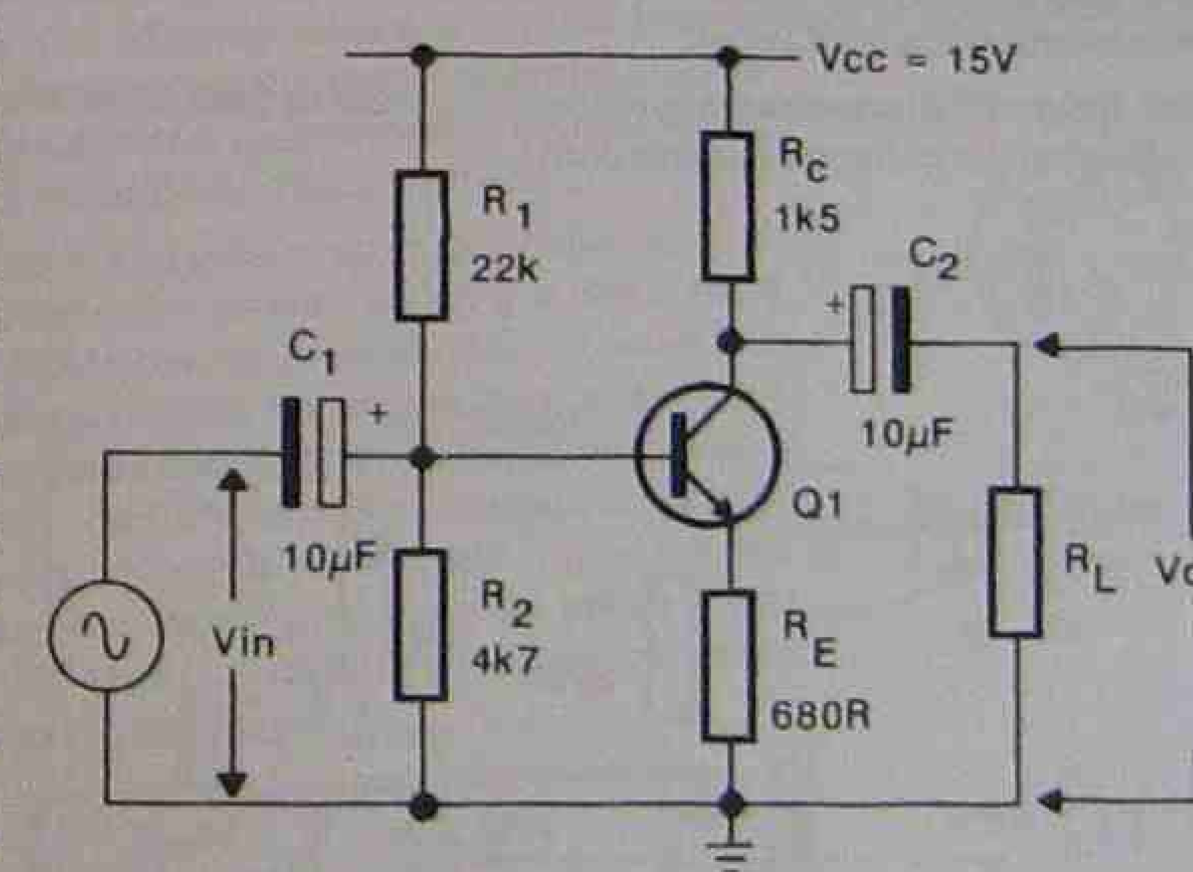


Fig.1: The common emitter amplifier

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).

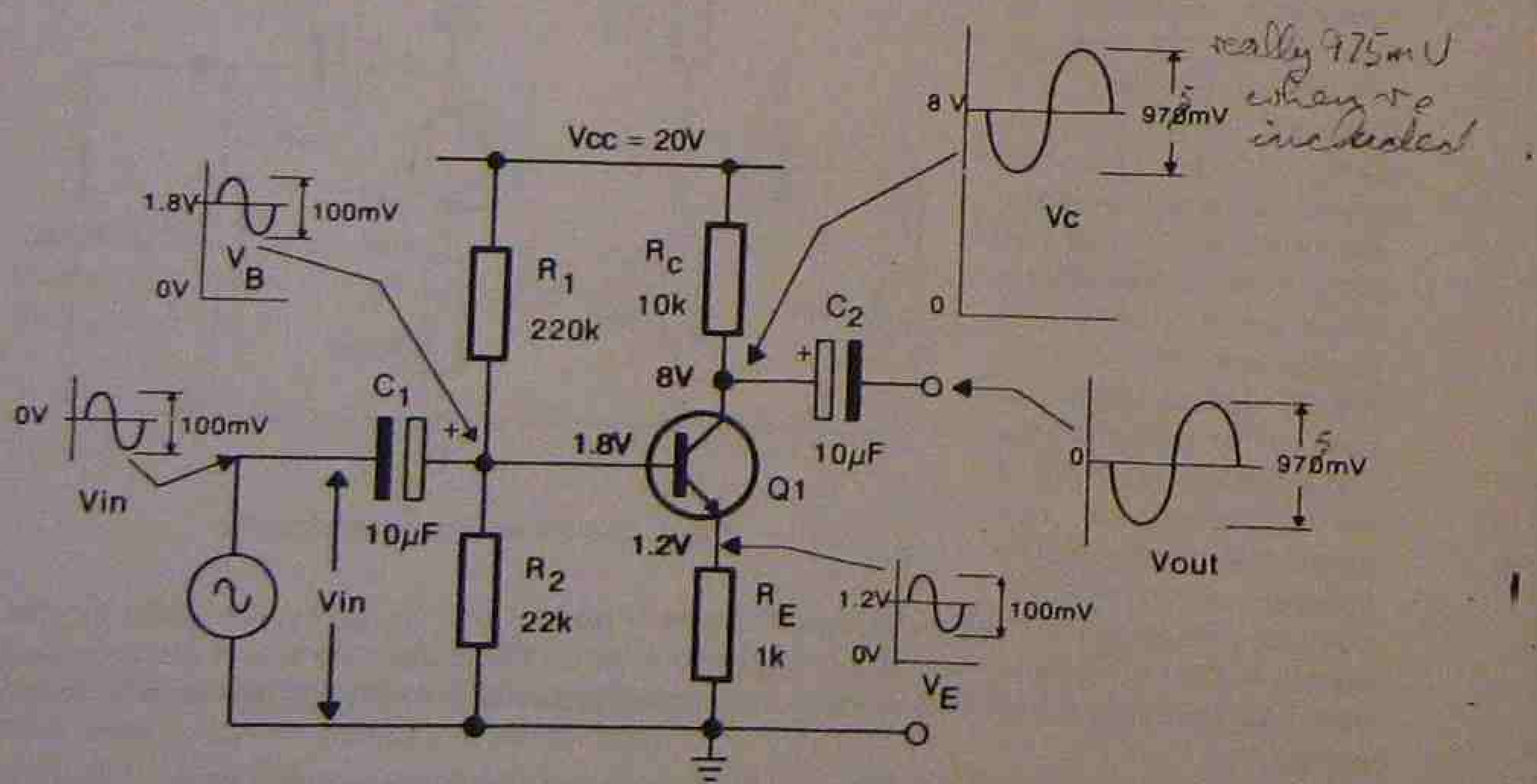


Fig.2: *Waveforms for the common emitter (CE) amplifier*

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):

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

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.

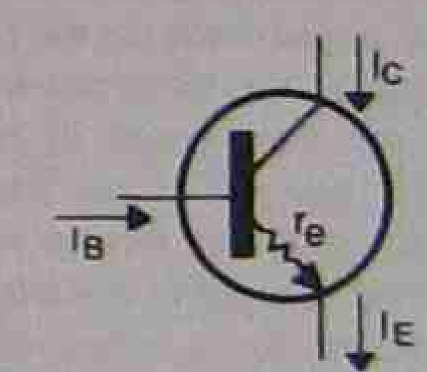


Fig.3: dynamic resistance re

$$\square \quad re = \frac{30mV}{Ic\ mA}$$

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:

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

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 R_L is in parallel with the collector resistor (R_C) so the gain equation that takes into account r_e and R_L is:

$$\square \quad A_V = \frac{R_C // R_L}{R_E + r_e}$$

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 R_L will therefore appear as an open-circuit (infinite resistance), leaving R_C 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 (B) of the transistor. That is, the series connected emitter resistor (R_E) and the dynamic resistor r_e are effectively increased in resistance to a value equal to their sum multiplied by the B of the transistor. The equation to calculate input resistance (r_{in}) is shown below:

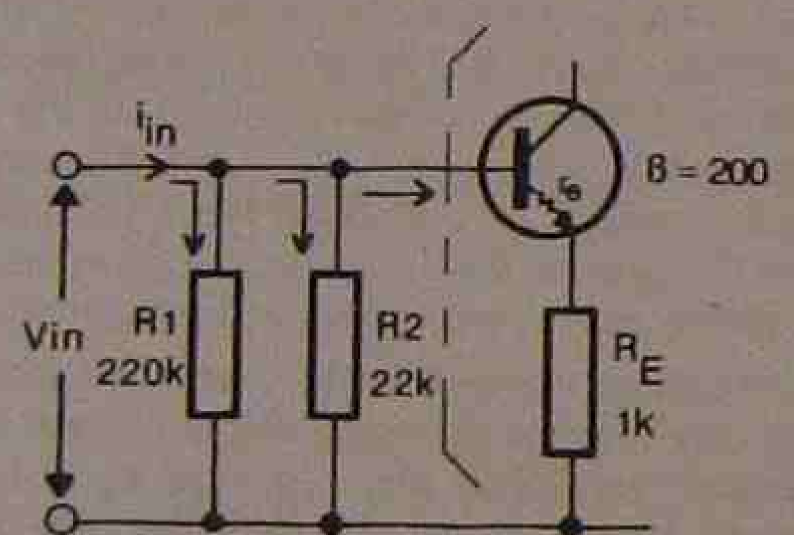


Fig.4: equivalent circuit for input resistance.

$$\square \quad r_{in} = R1//R2//[B(r_e + R_E)]$$

For the equivalent circuit of Fig.4, r_{in} = 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 ($A_V \times v_{in}$). (See notes for week 3 for the equivalent circuit.) Thus, the equation to find output resistance r_o is:

$$\square \quad r_o = R_C$$

7. ADDING AN EMITTER BYPASS CAPACITOR

The equation for gain shows that the value of the emitter resistor (R_E in Fig.2) affects the voltage gain of the circuit. For optimum DC stability, R_E 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 (C_E) connected across one of these as shown in Fig.5. The bypass capacitor can also be connected across a single emitter resistor (R_E in Fig.2) or across either emitter resistor (R_{E1} or R_{E2} 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.

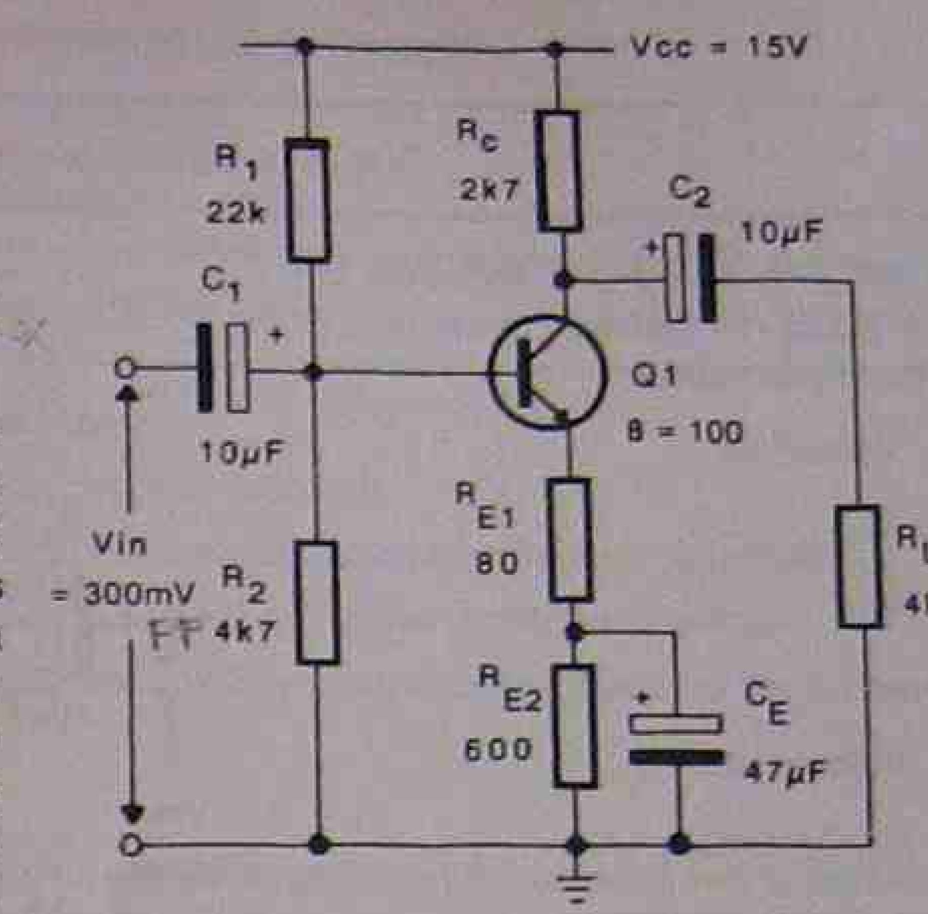


Fig.5: Adding a bypass capacitor

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 C_E is disconnected.

(a) $V_B = \frac{V_{CC} R2}{R1 + R2} = \frac{15 \times 4k7}{22k + 4k7} = \underline{2.6V}$

$V_E = V_B - V_{be} = 2.6 - 0.6 = \underline{2V}$

$I_E = I_C = \frac{V_E}{R_E} = \frac{2}{680} = 3mA$

$V_C = V_{CC} - I_C R_C = 15 - (2k7 \times 3mA) = \underline{6.9V}$

$r_e = \frac{30mV}{I_C} = \frac{30mV}{3mA} = 10$ ohms

(b) $A_V = \frac{R_C//R_L}{r_e + R_{E1}} = \frac{2k7//4k7}{10 + 80} = \frac{1715}{90} = \underline{19}$

(c) $r_{in} = R1//R2//[B(r_e + R_{E1})]$ = 22k//4k7//[100(10 + 80)] = 3k87//9k = **2k71 ohms**

(d) $r_o = R_C$ = **2k7 ohms**

(e) $A_V = \frac{R_C//R_L}{r_e + R_{E1} + R_{E2}} = \frac{2k7//4k7}{10 + 80 + 600} = \frac{1715}{690} = \underline{2.5}$ (C_E disconnected)

$r_{in} = R1//R2//[B(r_e + R_{E1} + R_{E2})]$ = 22k//4k7//[100(10 + 80 + 600)]
= 3k87//69k = **3k7ohms** (C_E disconnected)

****

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

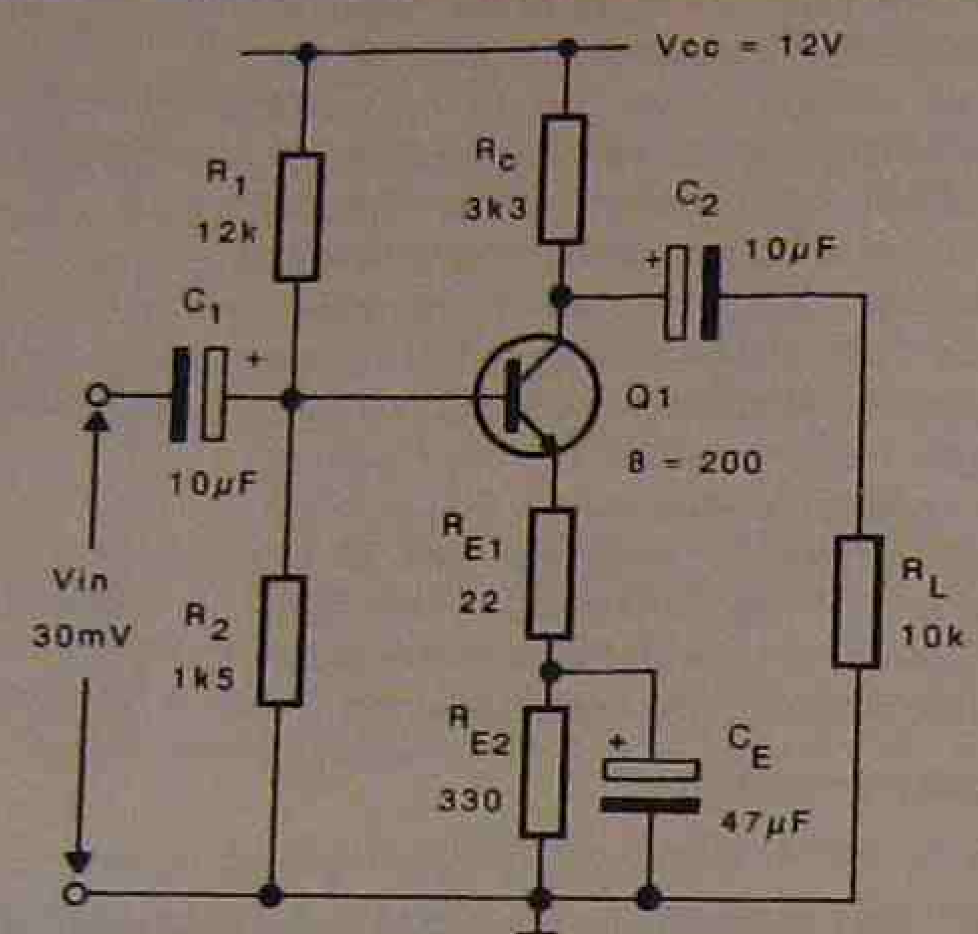
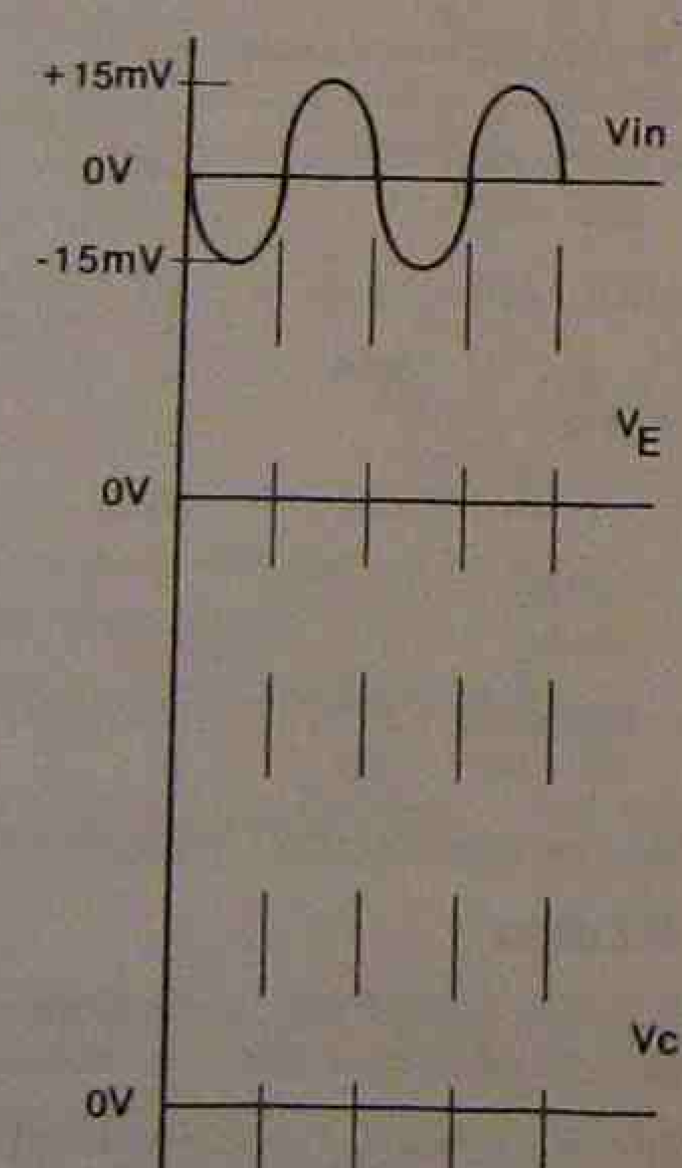


Fig.1

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.

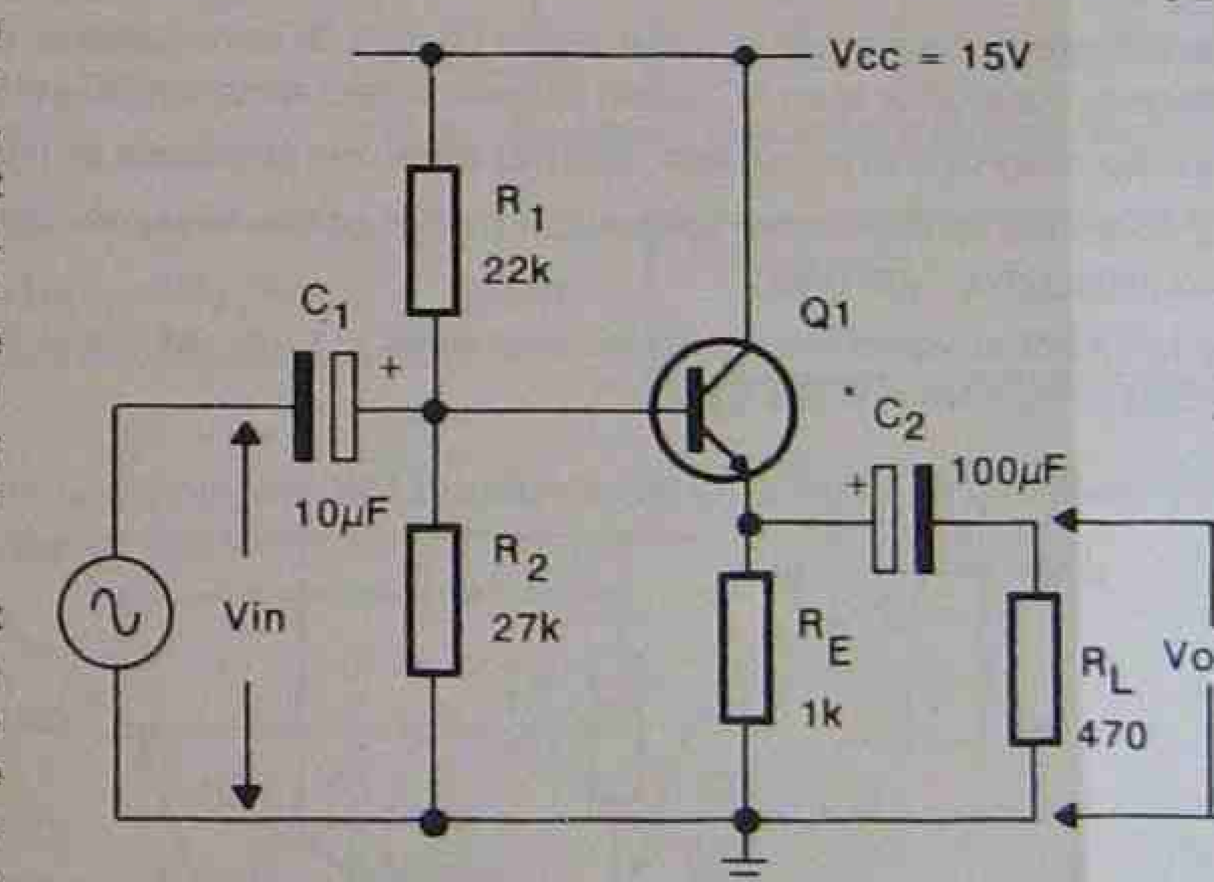


Fig.1: The common collector amplifier

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) $V_B = \dfrac{V_{CC} R2}{R1 + R2} = \dfrac{15 \times 27k}{22k + 27k} = \underline{8.3V}$

(b) $V_E = V_B - V_{be} = 8.3 - 0.6 = \underline{7.7V}$

(c) $I_E = I_C = \dfrac{V_E}{R_E} = \dfrac{7.7}{1k} = \underline{7.7mA}$

(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).

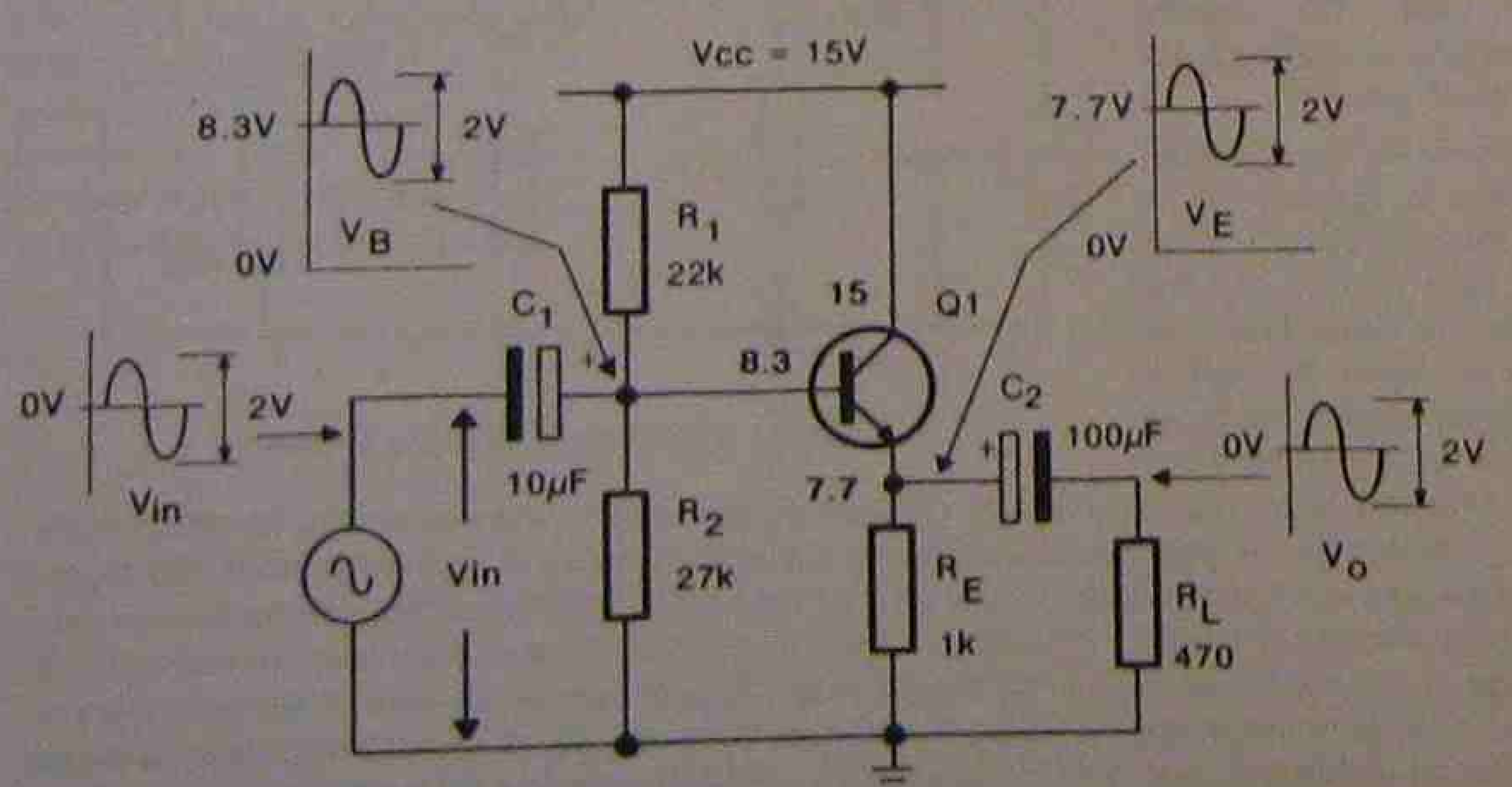


Fig.2: Waveforms for the common collector (CC) amplifier

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:

$$A_v = \text{unity}$$

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:

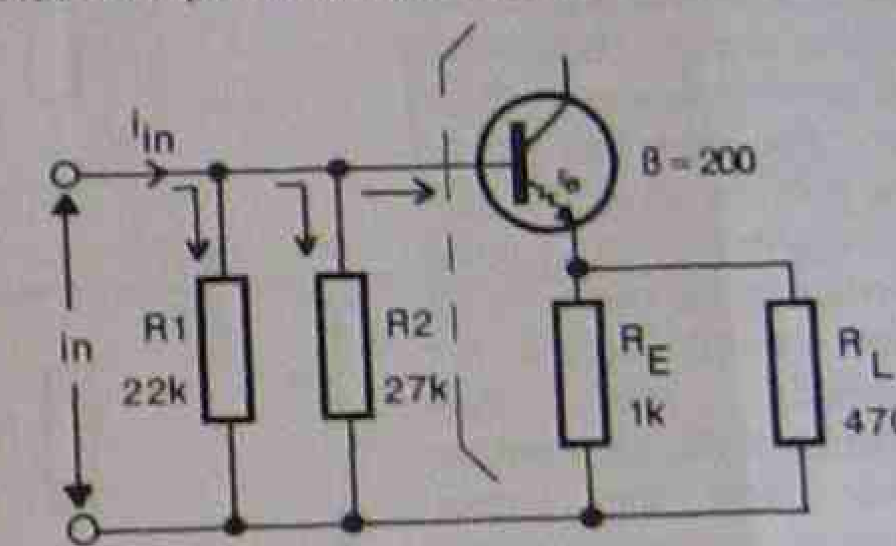


Fig.3: equivalent circuit for input resistance

$$r_{in} = R1//R2//[ß(r_e + R_E//R_L)]$$

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) $r_e = \dfrac{30mV}{I_C} = \dfrac{30mV}{7.7mA} = \underline{3.9\ ohms}$

(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.

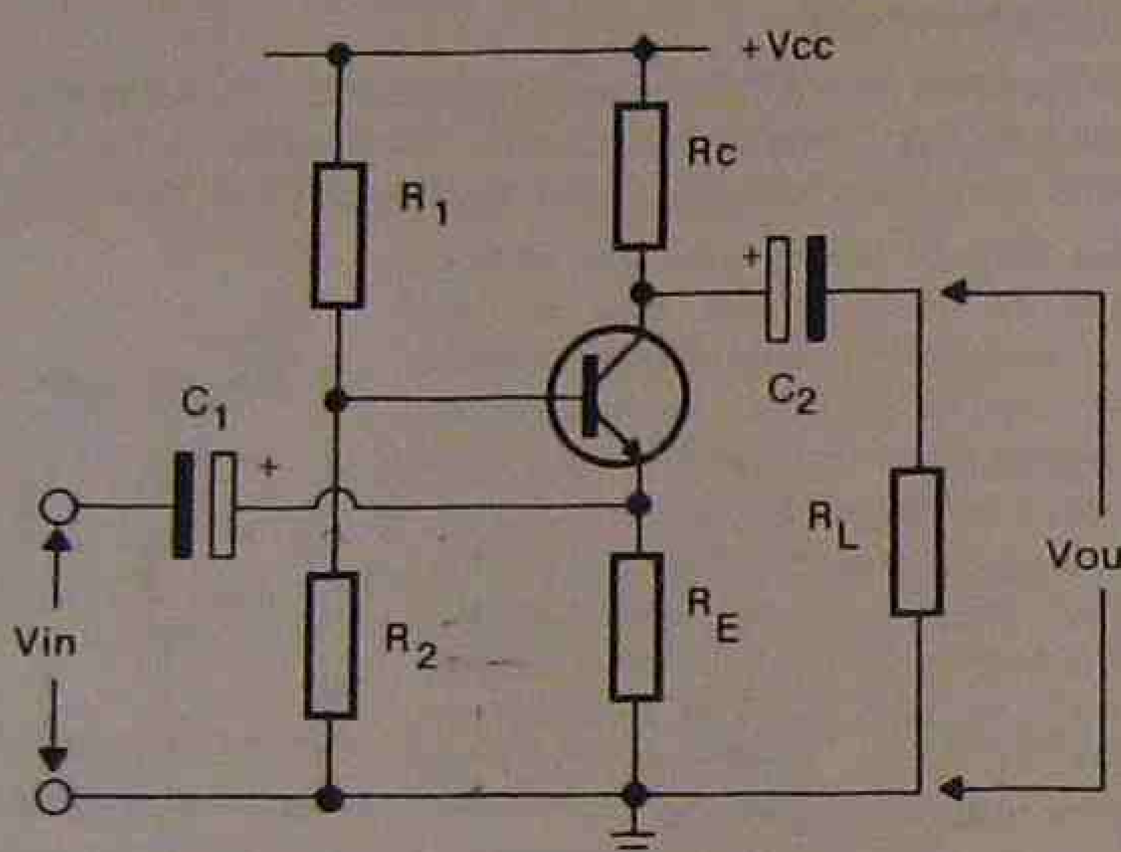


Fig.4: The common base amplifier

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.

****

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

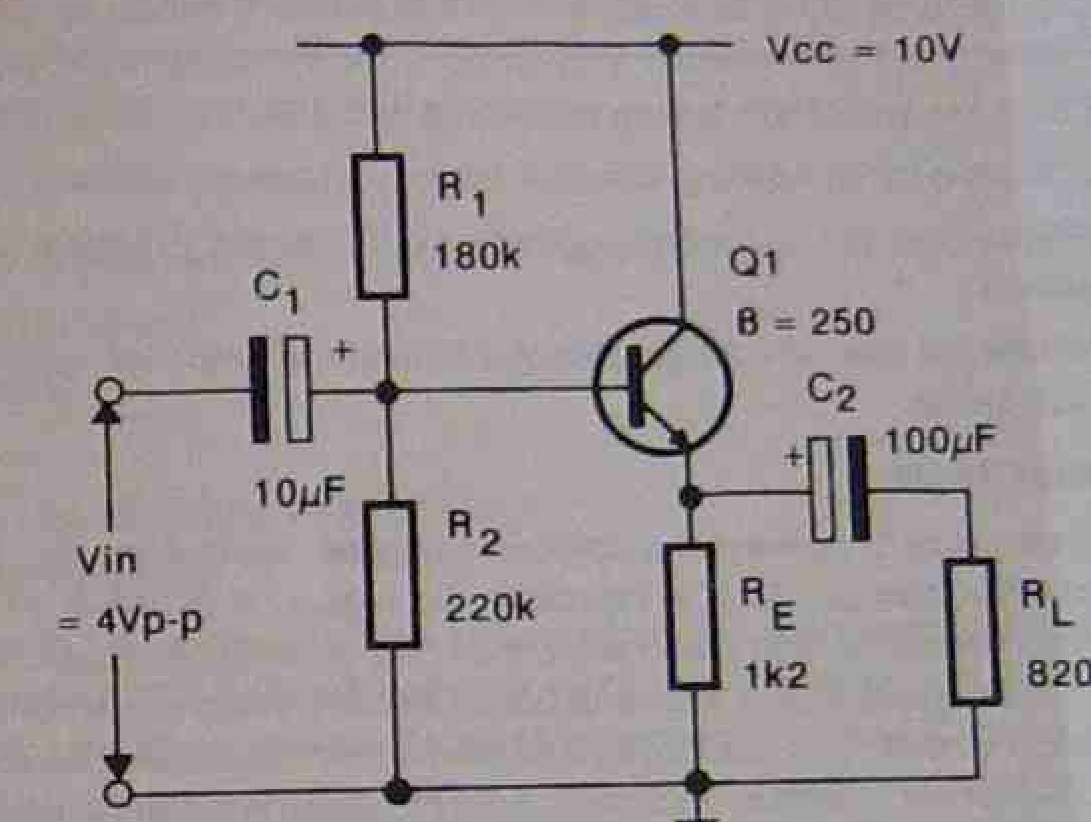
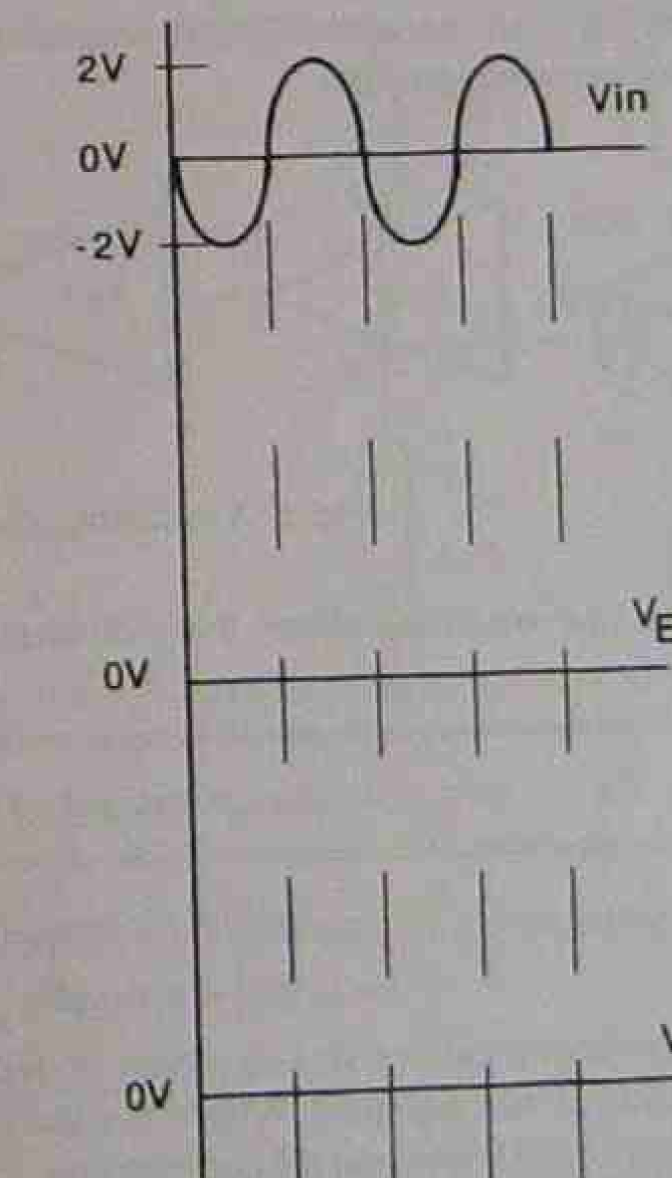


Fig.1

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 vin. 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.

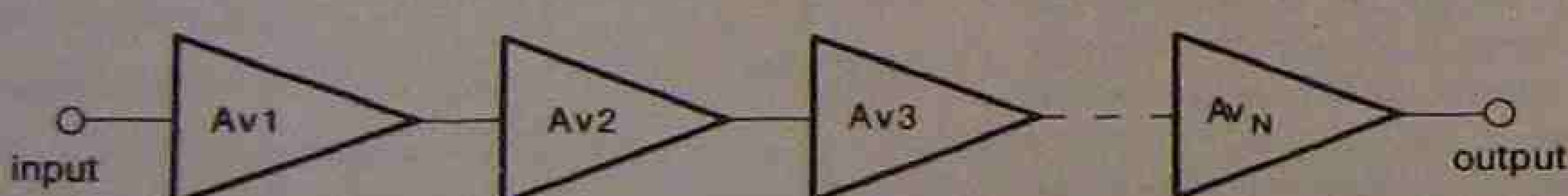


Fig.1: A multistage amplifier

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.

$$A_{VT}\text{(in dB)} = dB1 + dB2 + dB3 + \ldots dB_N$$

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.

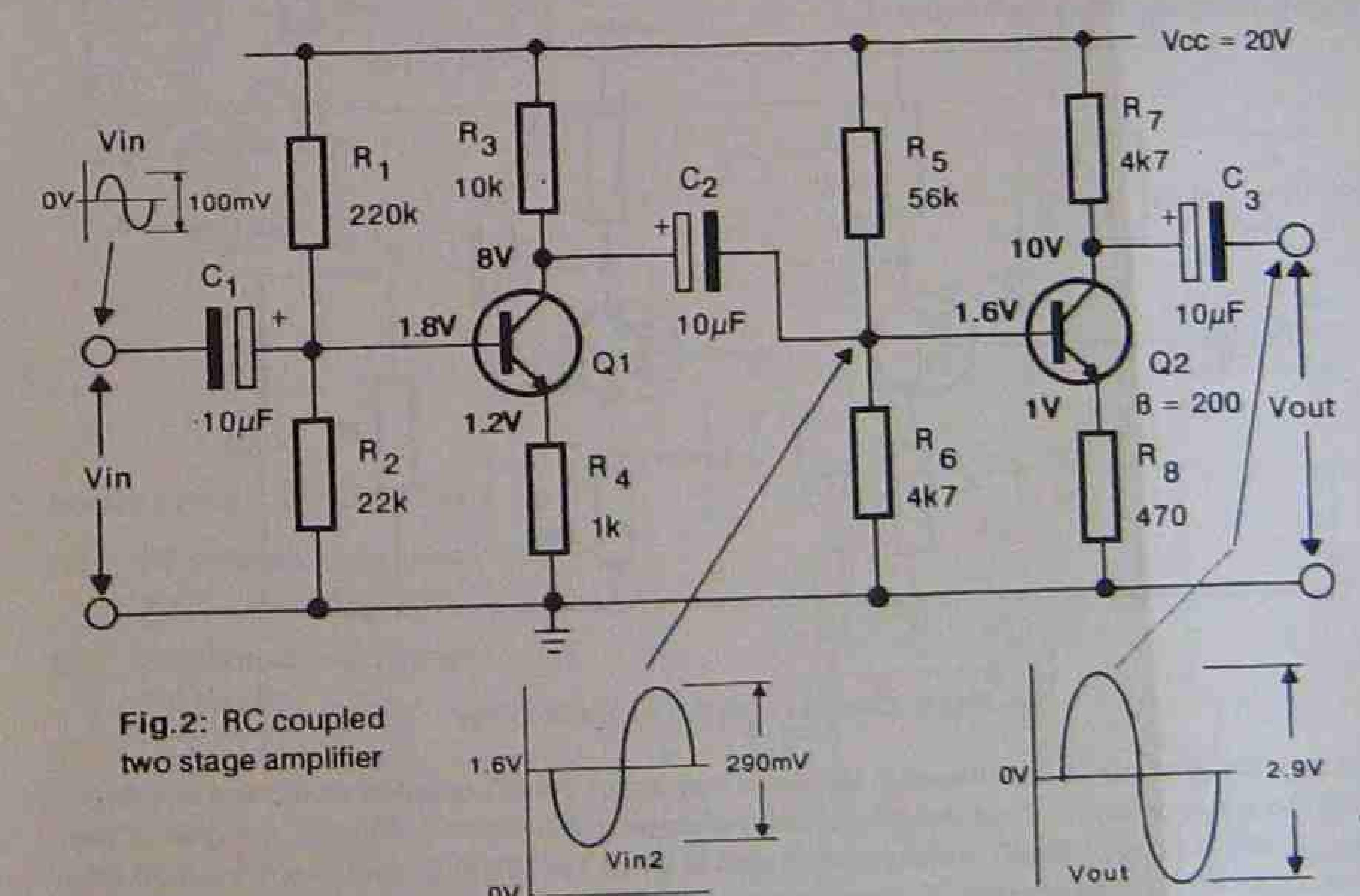


Fig.2: RC coupled two stage amplifier

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 $R7/r_{e2}$, 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 $A_v = v_o/v_{in}$ 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.

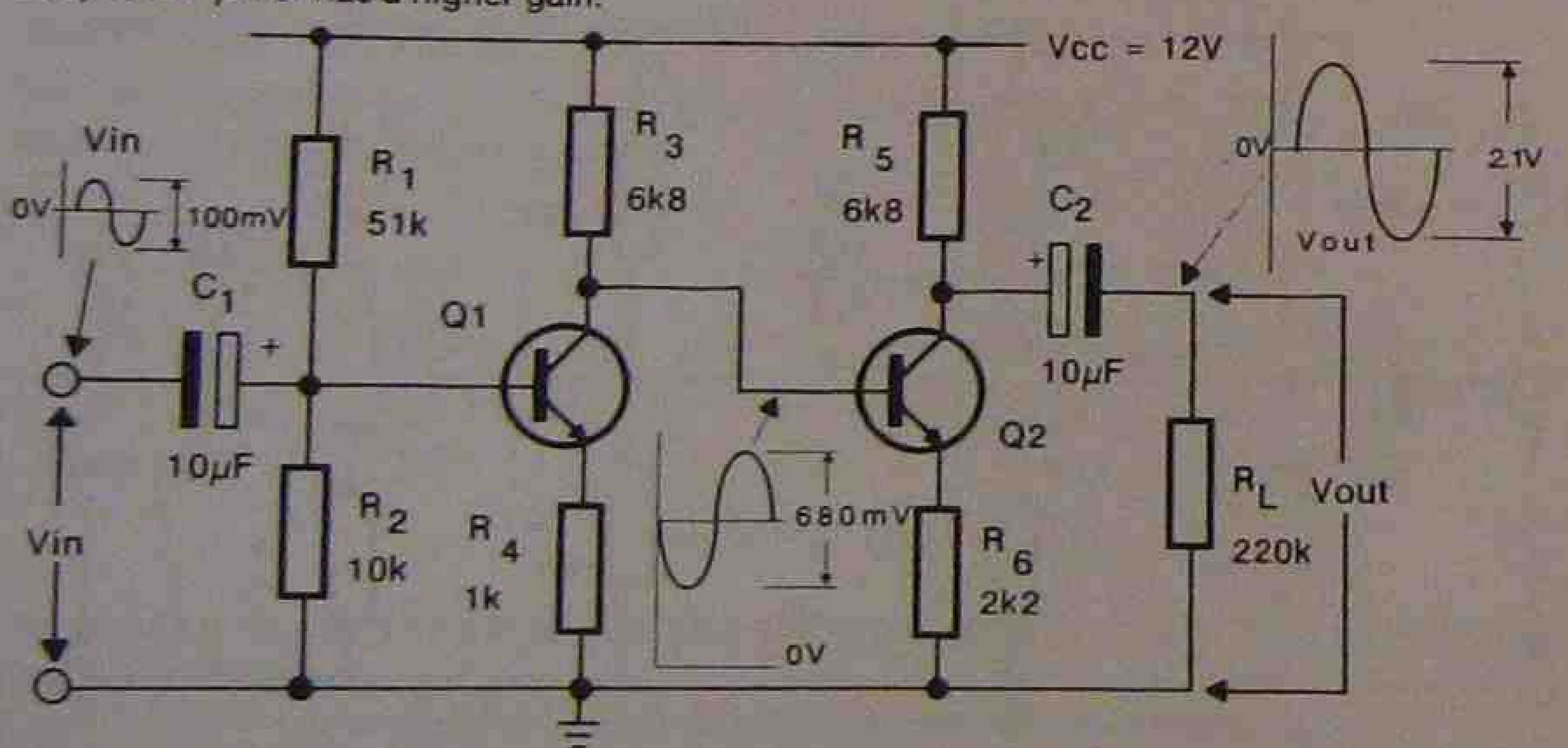


Fig.3: Direct coupled, 2 stage amplifier

The gain of the circuit is determined in the same way as for the RC coupled multistage amplifier using the equation v_o/v_{in}. 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.

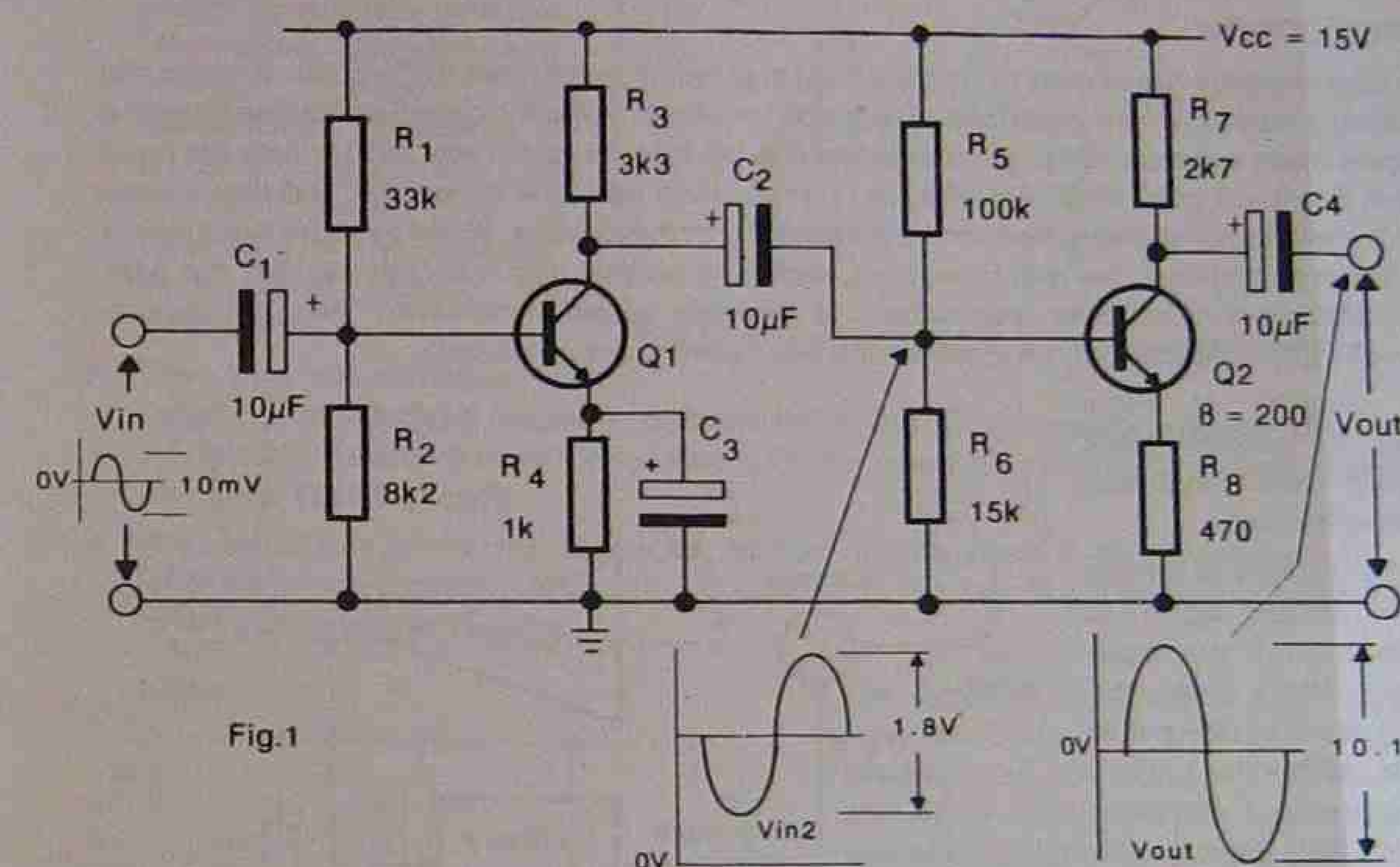


Fig.1

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.

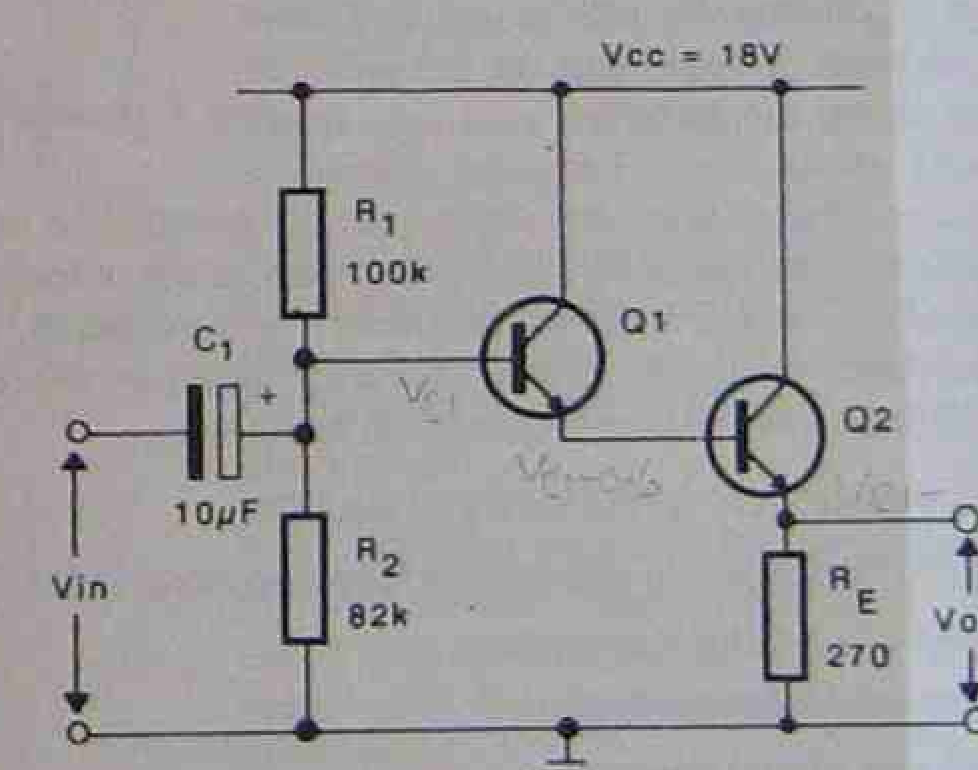


Fig.2

****

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.

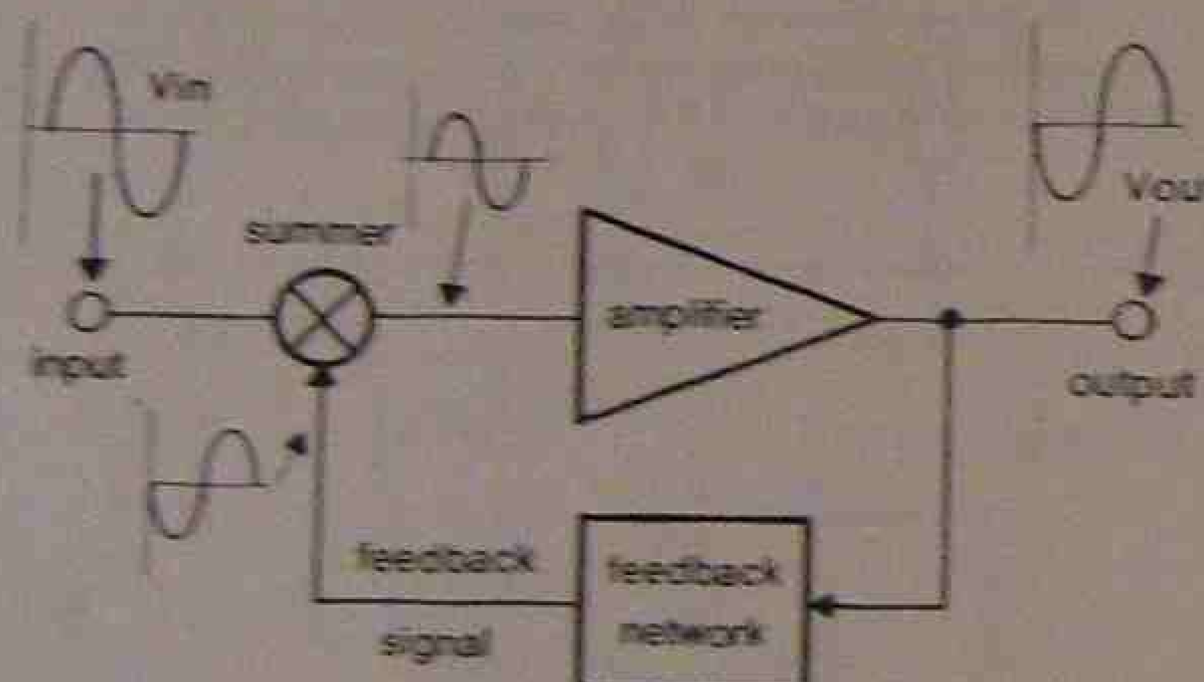


Fig.1: general form of negative feedback

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.

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 R_E 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.

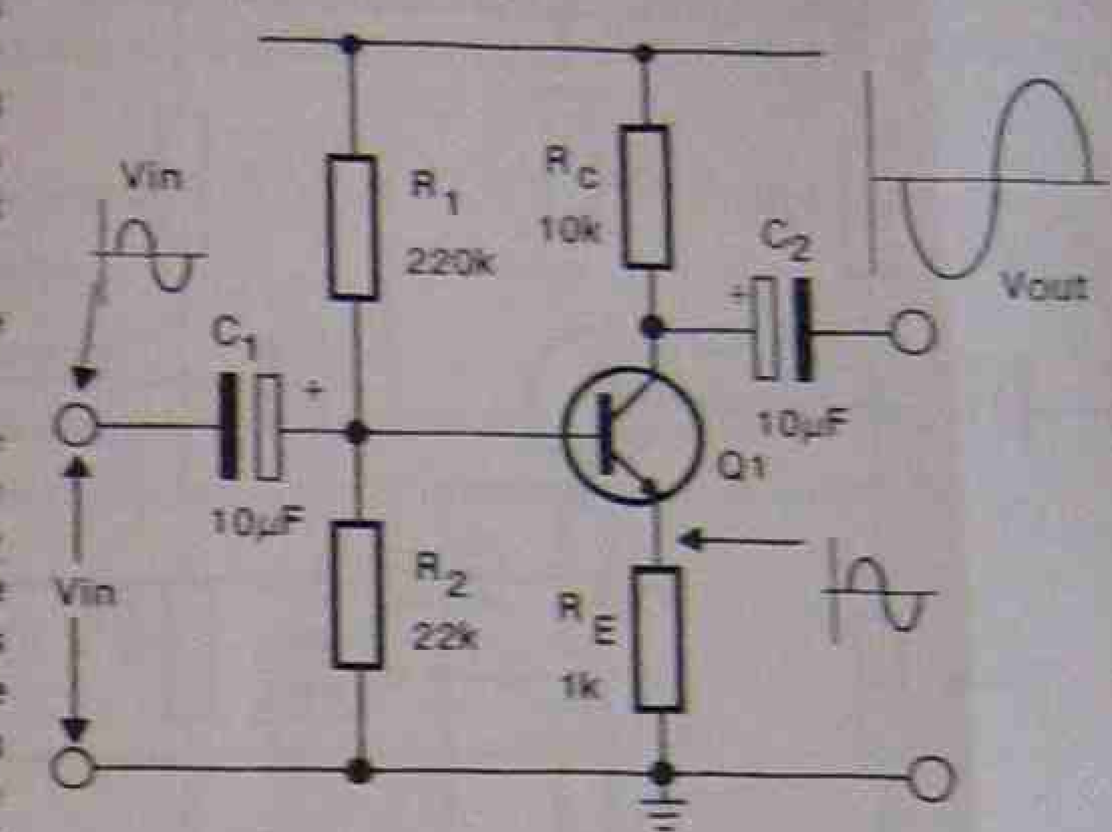


Fig.2: AC and DC feedback with an emitter resistor

- 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.

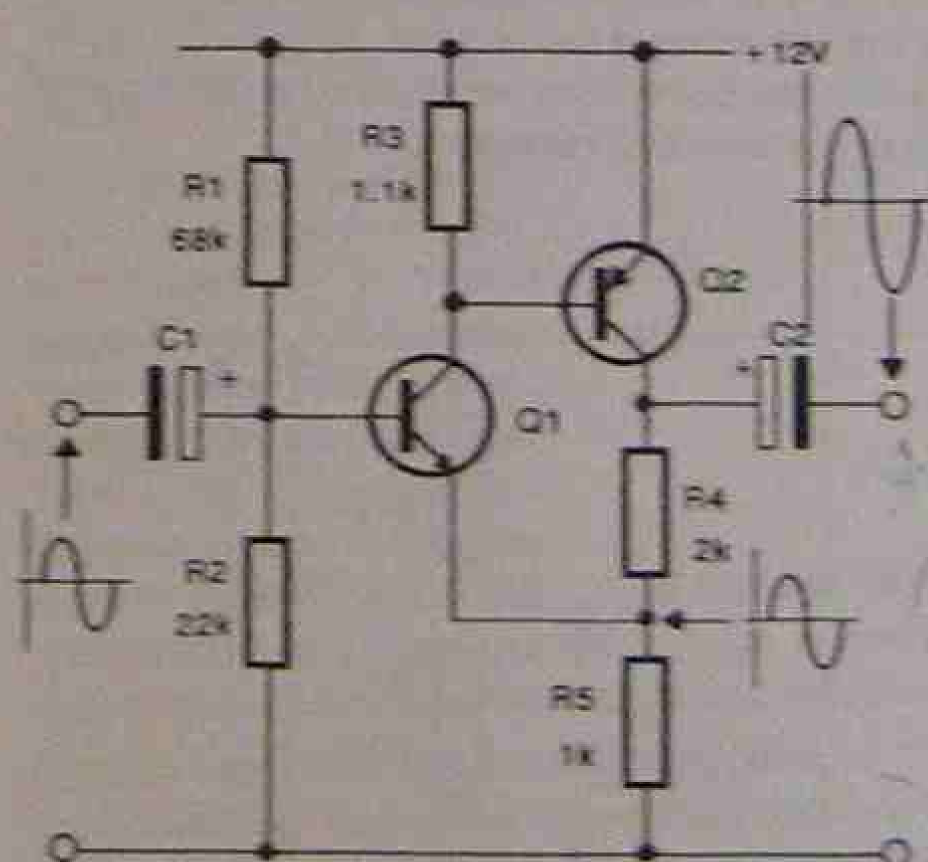


Fig.3: DC and AC feedback

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.

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.

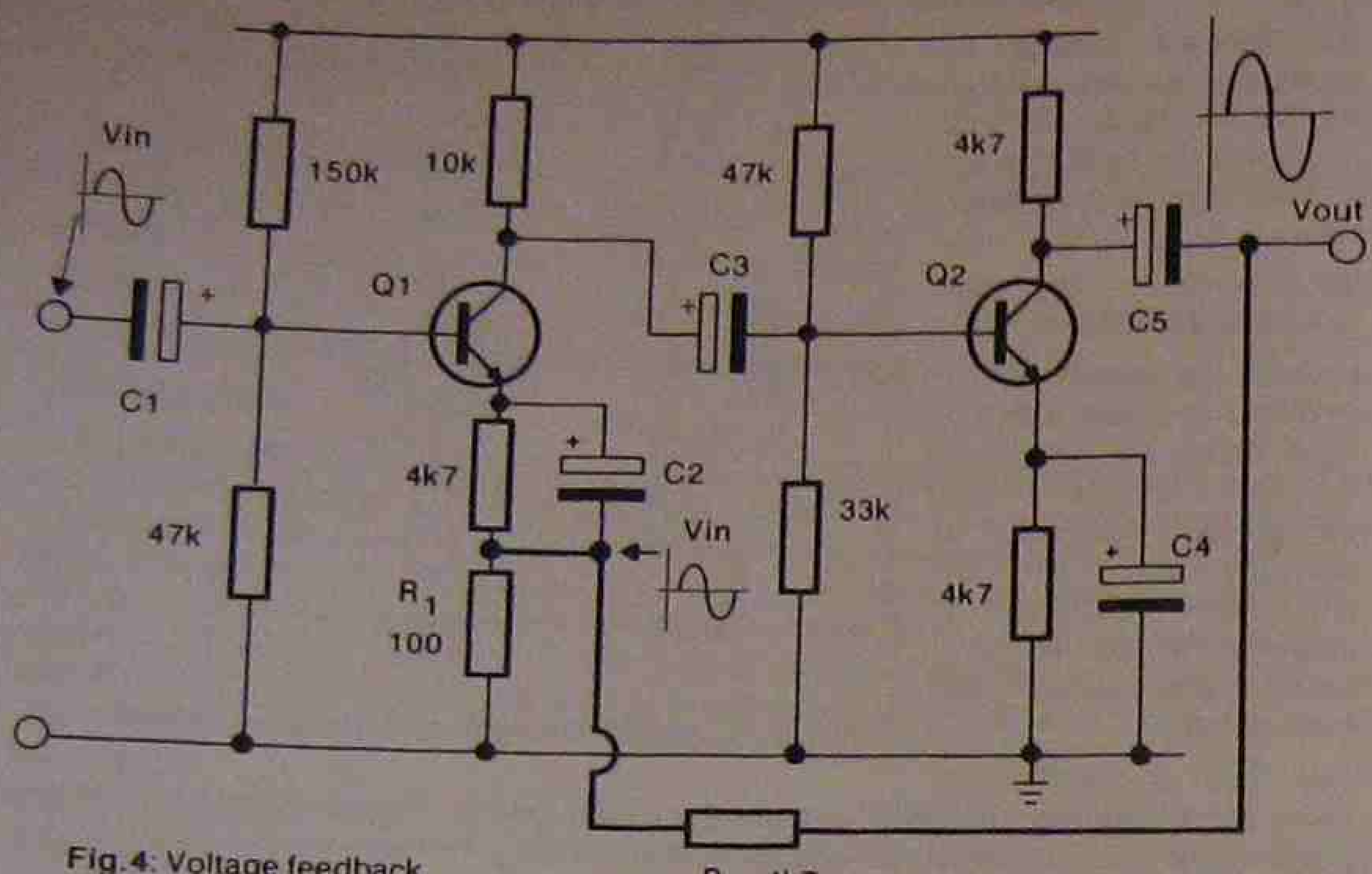


Fig.4: Voltage feedback

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:

$$\square \quad \frac{Vout}{Vin} = \frac{Rf}{R1} + 1 = \text{gain}$$

$$\text{Gain of Fig.4} = \frac{4k7}{100} + 1 = \underline{48}$$

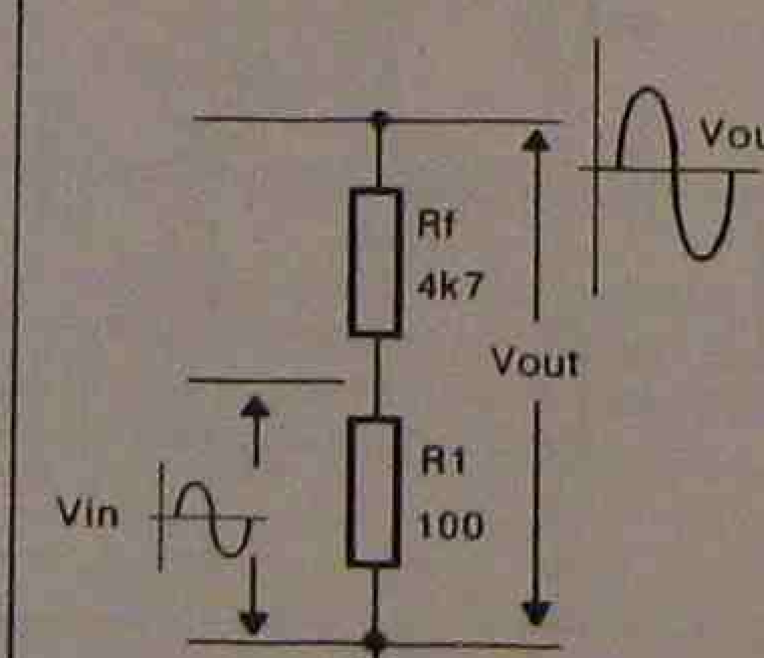


Fig.5: Equivalent circuit of feedback network of Fig.4.

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.

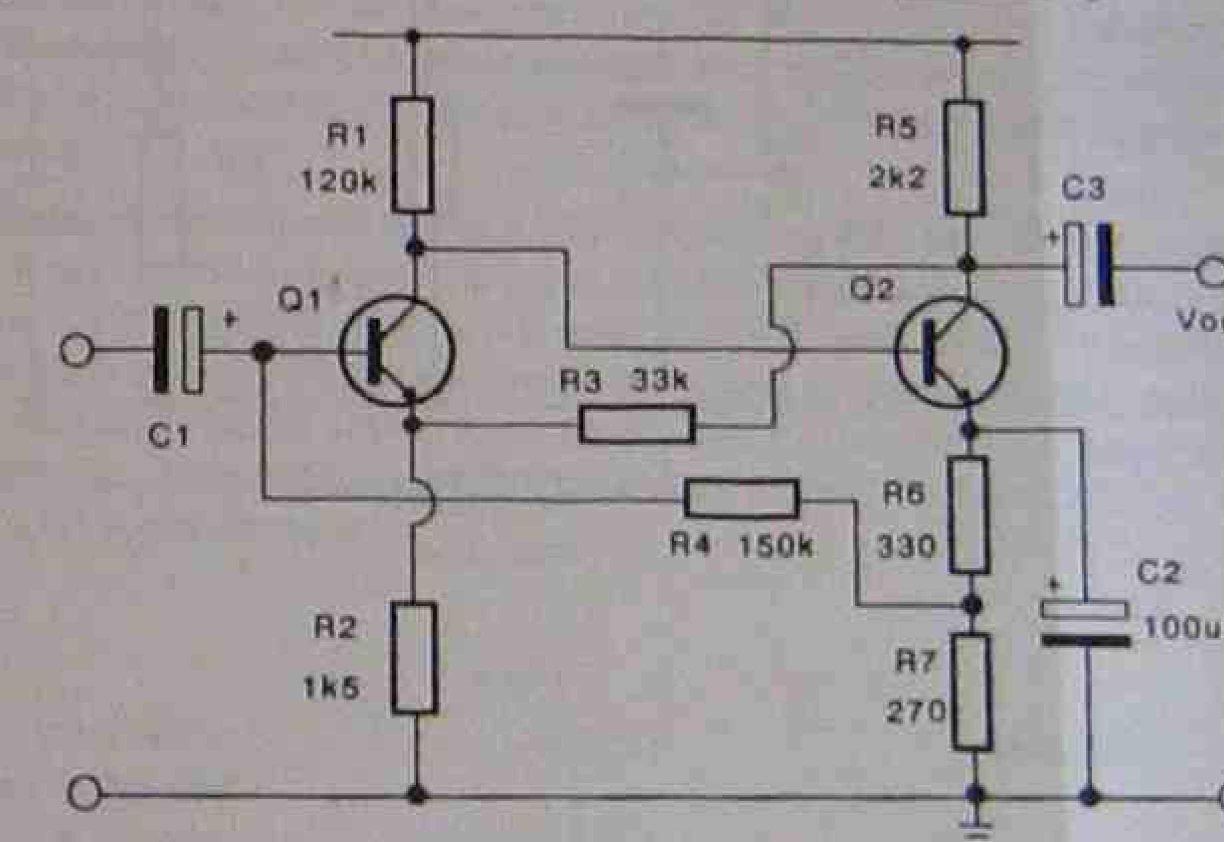


Fig.6: example of AC and DC feedback

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.

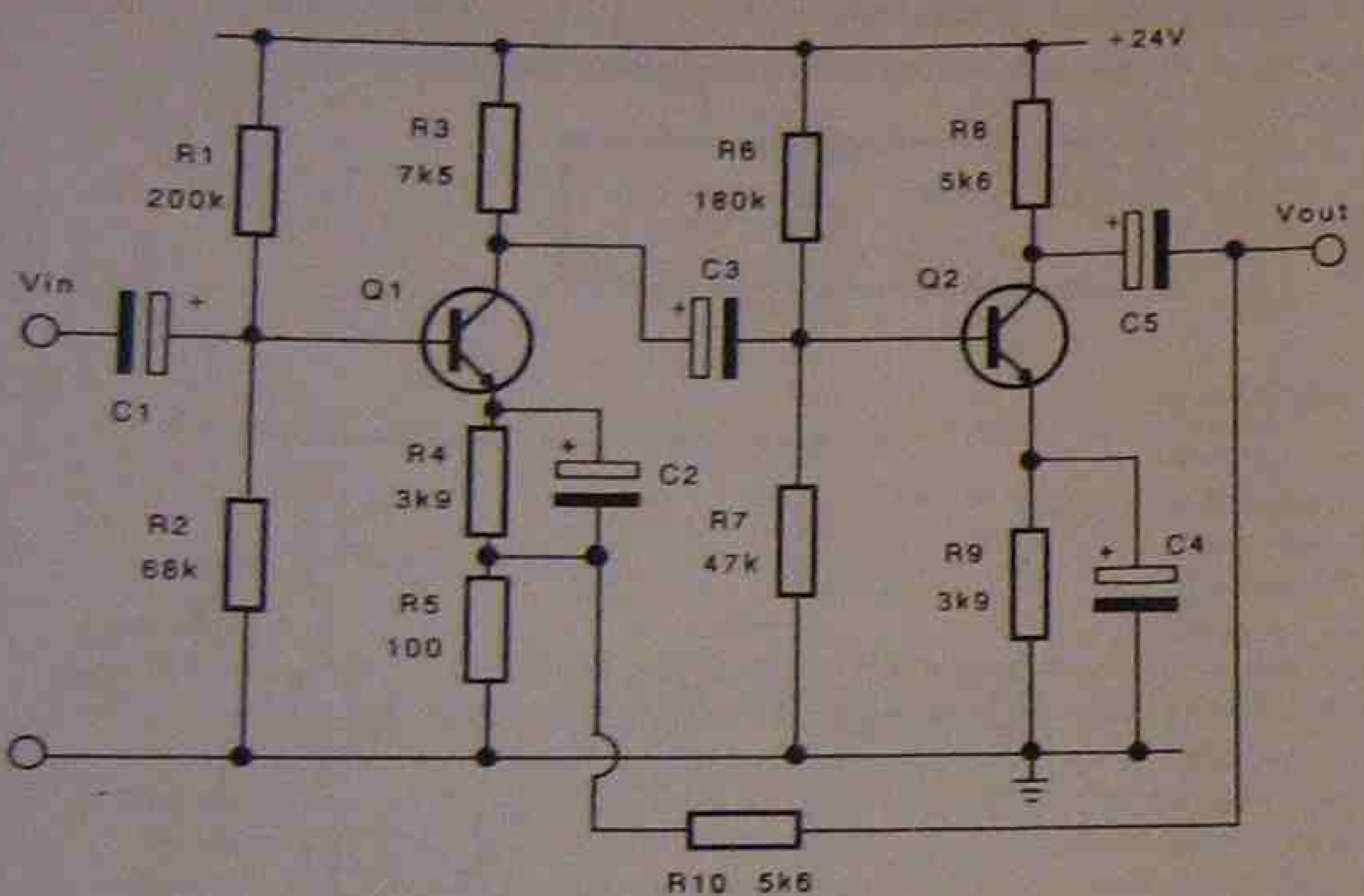


Fig.1

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.

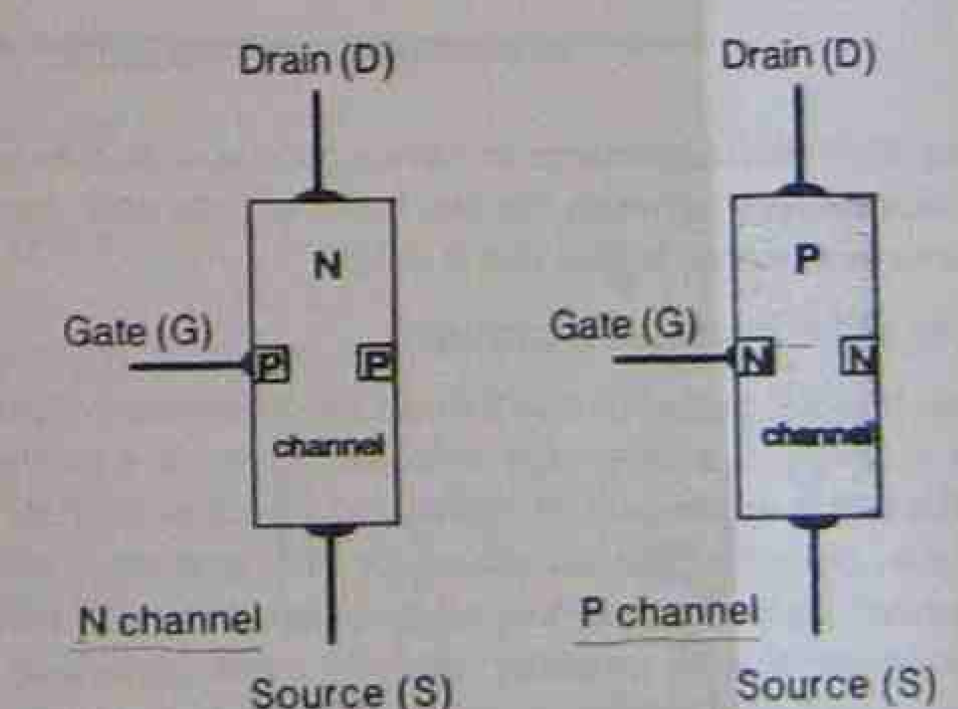


Fig.1: structure of the JFET

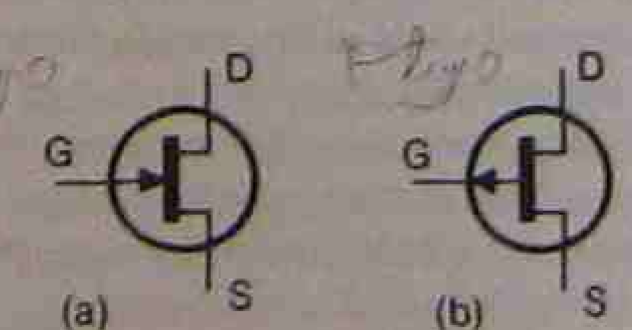


Fig.2 (a) N channel, (b) P channel

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.

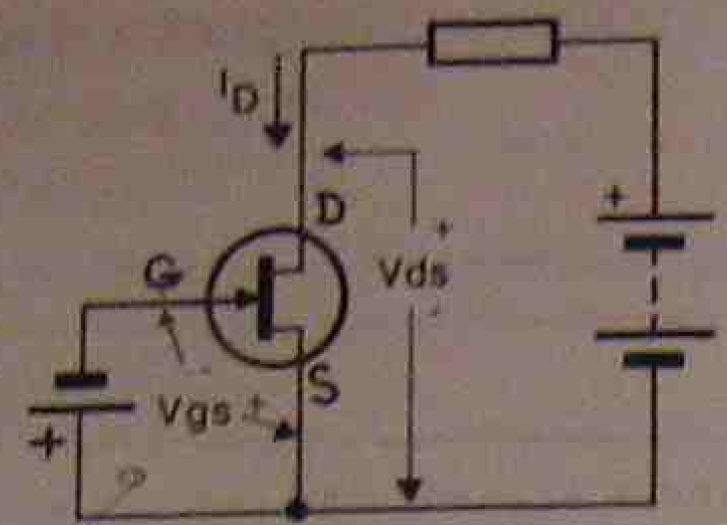


Fig.3: Voltage polarities for the N channel JFET

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**.

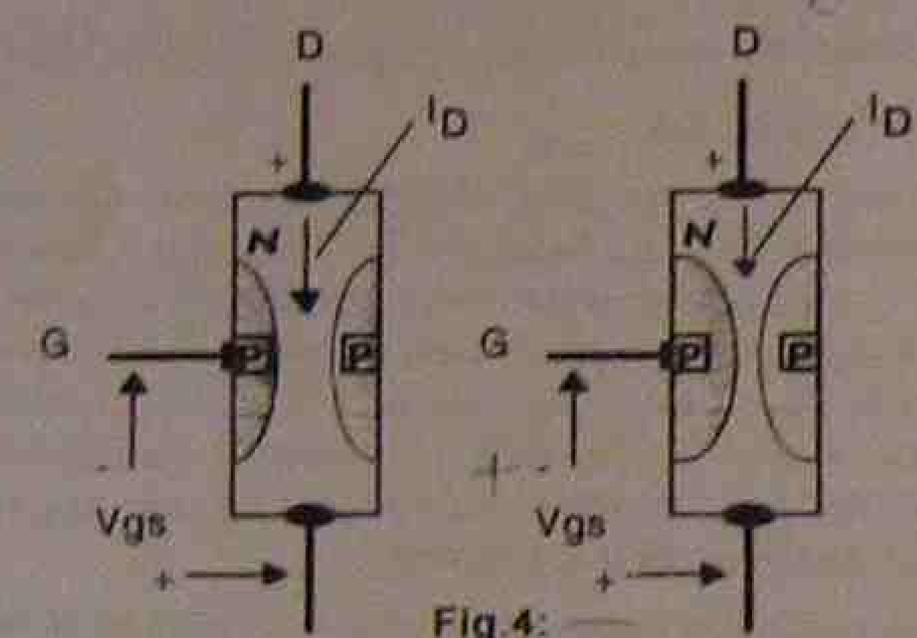


Fig.4: (a) small Vgs, large I_D (b) large Vgs, small I_D

- 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.

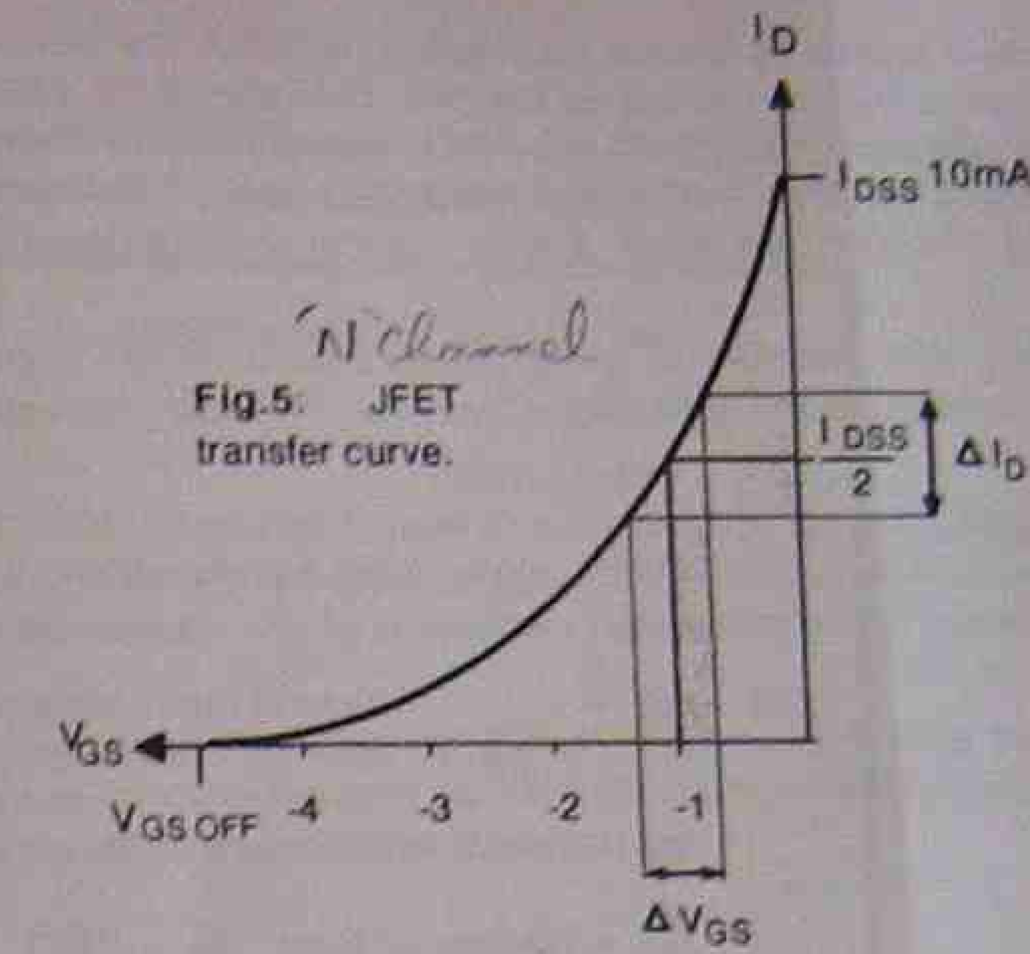


Fig.5: JFET transfer curve.

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:

$$gm = \frac{I_D}{V_{GS}} \text{ siemens or amps/volt}$$

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.

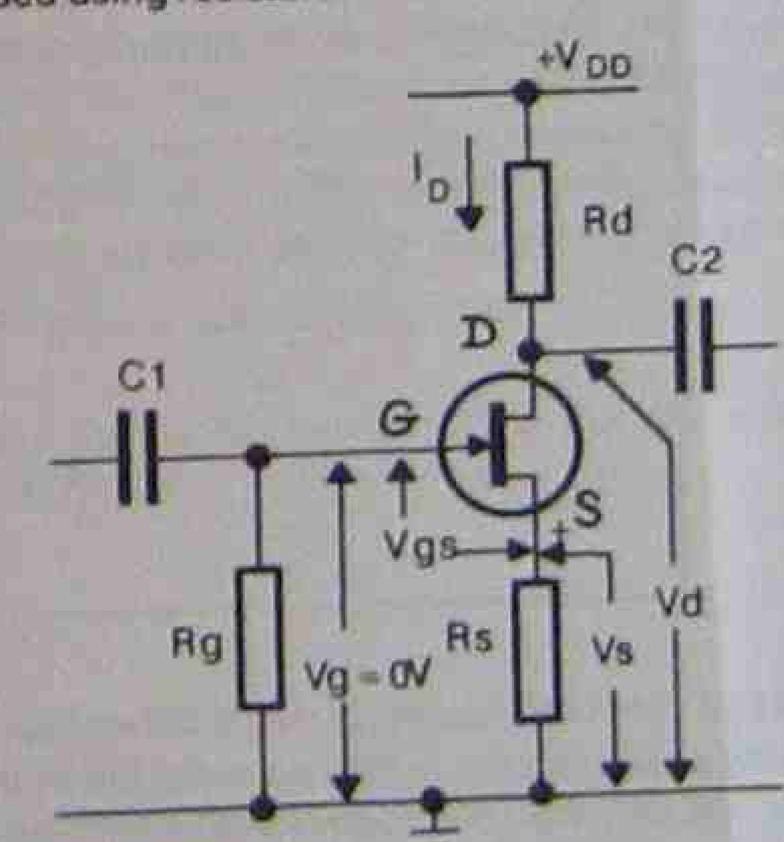


Fig.6

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.

<u>EXAMPLE:</u> 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) $I_D = \frac{I_{DSS}}{2} = \underline{5mA}$

$V_{GS} = -1V$ (from transfer curve, Fig.5)

$R_S = \frac{V_{GS}}{I_D} = 1/5mA = 200$ ohm (chose NPV of **<u>220 ohm</u>**)

(b) $V_D = \frac{V_{DD}}{2} = 10/2 = \underline{5V}$

$R_D = \frac{V_{DD} - V_D}{I_D} = (10 - 5)/5mA = \underline{1k\ ohm}$

(c) $g_m = \frac{I_D}{V_{GS}} = 5mA/1V = \underline{5mA/V}$

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.

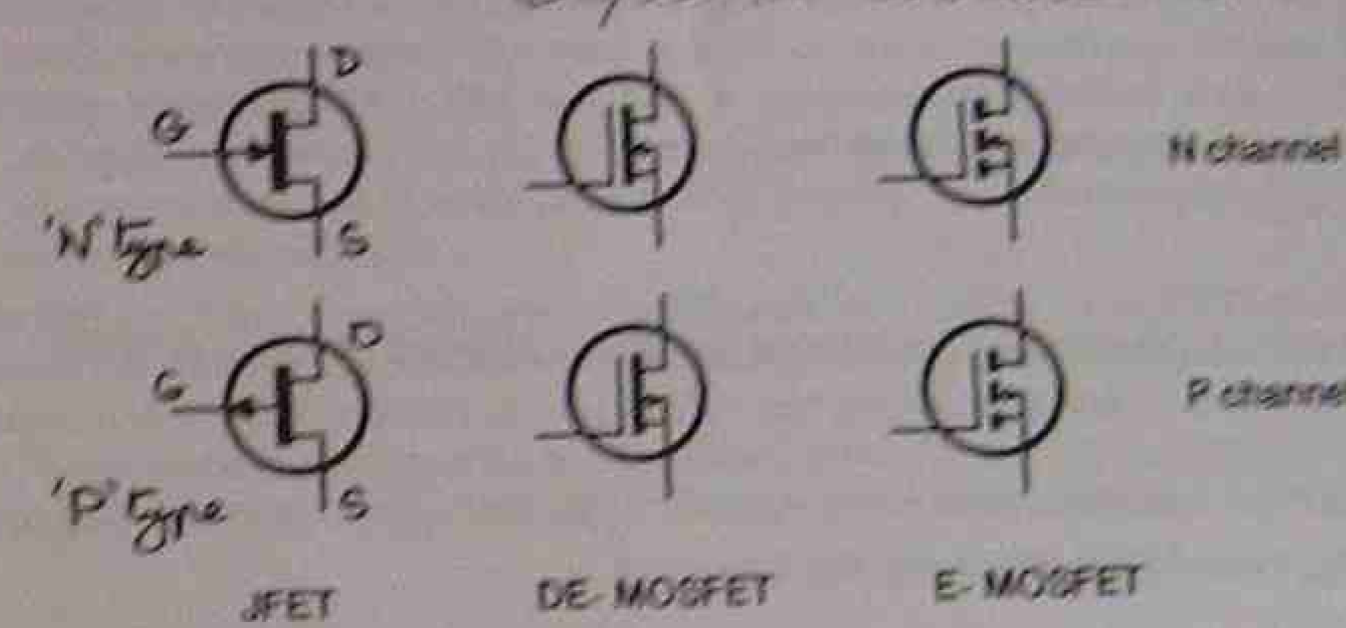


Fig.7: FET symbols

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.

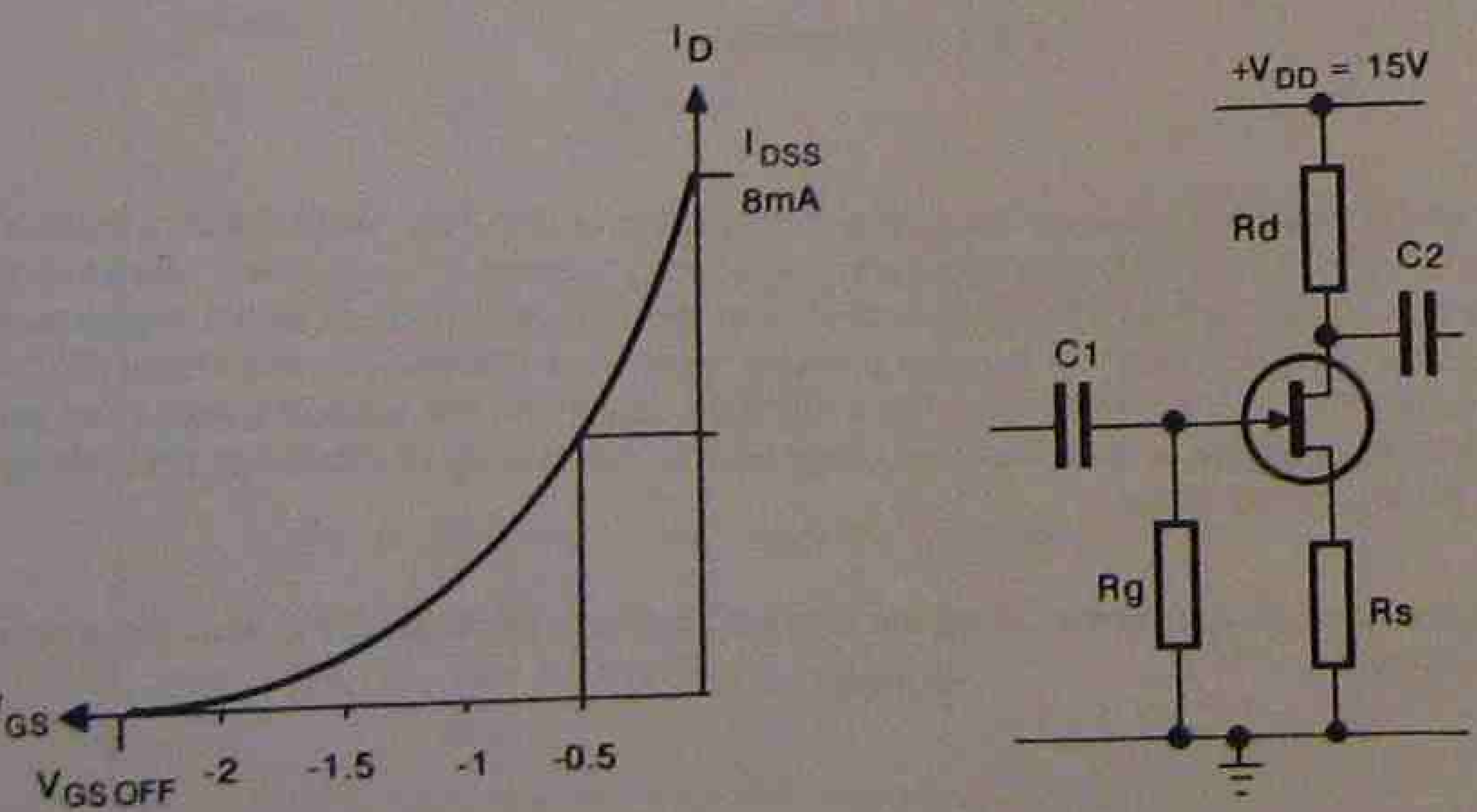


Fig.1

****

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.

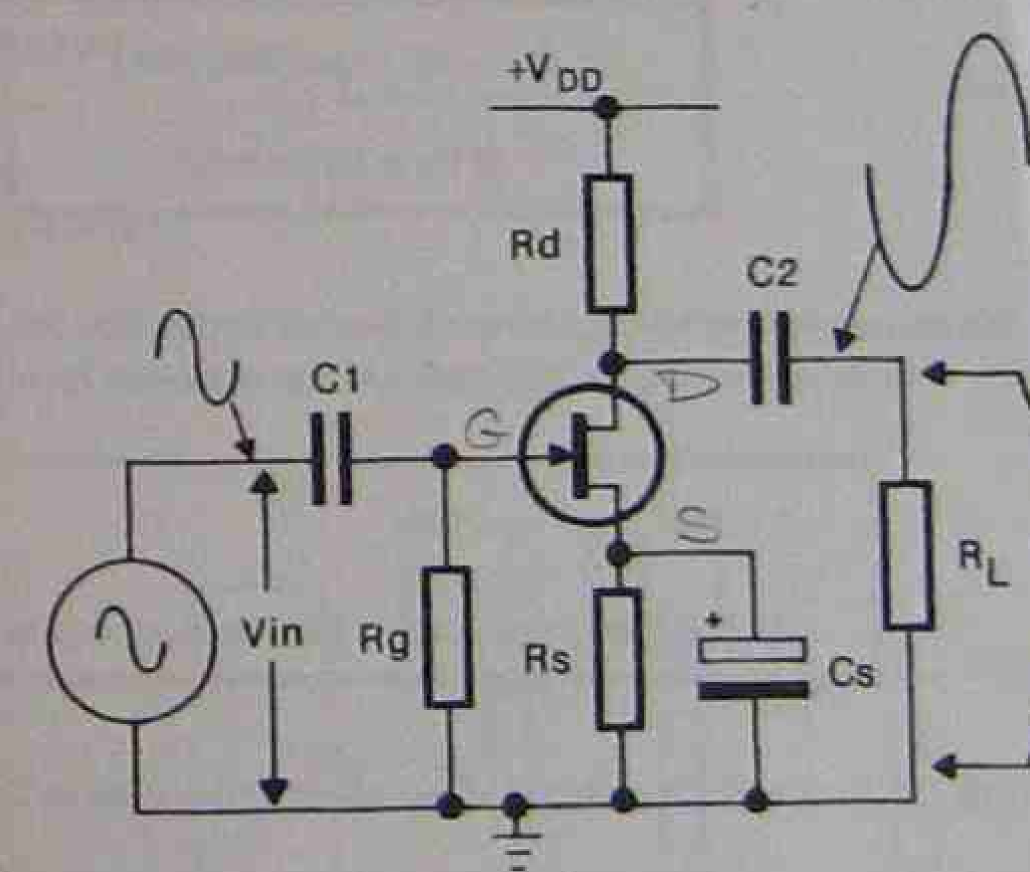


Fig.1: The JFET CS amplifier

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 ID) 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:

$gm = \frac{I_D}{V_{GS}}$

☐ $A_v = \dfrac{g_m (R_d // R_L)}{1 + g_m R_s}$

where: gm = mutual conductance in A/V Siemens
Rd = value of drain resistor
RL = 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:

☐ $A_v = g_m (R_d // R_L)$

(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:

☐ $A_v = g_m R_d$

(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:

☐ r_{in} = Rg ohms

The output resistance (r_{out}) for the circuit is the value of the drain resistor Rd: That is:

☐ r_{out} = Rd 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

with no Cs or RL
$A_v = \frac{3e^{-3}}{1} \times 3.9k = 11.7$

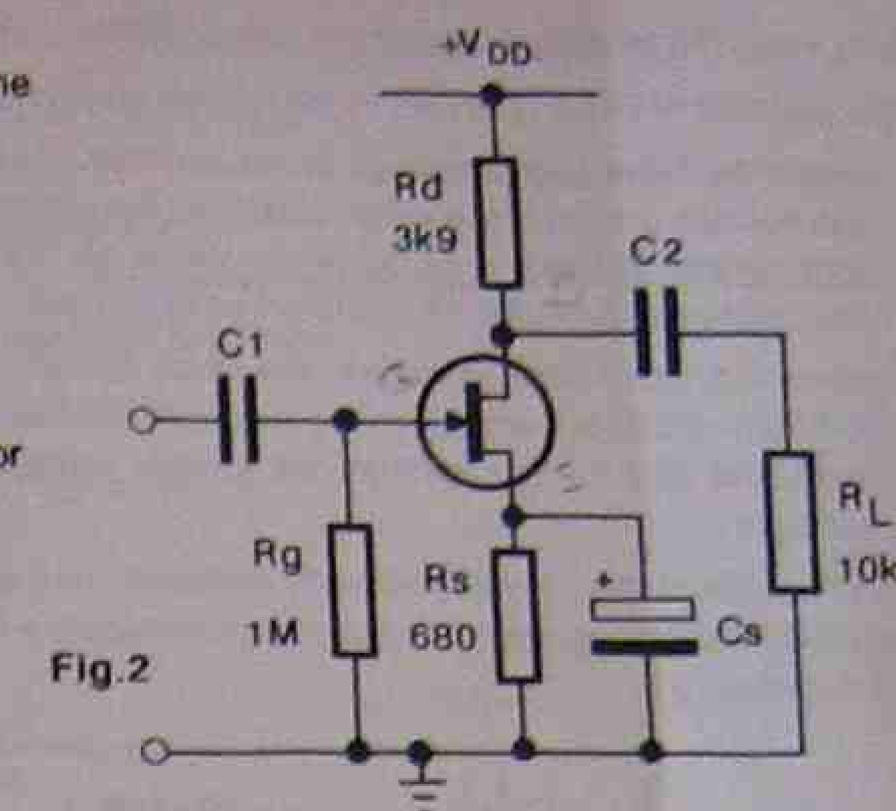


Fig.2

Solution:

(a) $A_v = \dfrac{g_m(R_d//R_L)}{1 + g_m R_s} = 3 \times 10^{-3}(3k9//10k) = 3 \times 10^{-3} \times 2k8 =$ **8.4**

(b) r_{in} = Rg ohms = **1M ohm**

(c) r_{out} = Rd ohms = **3k9 ohm**

(d) $A_v = \dfrac{g_m(R_d//R_L)}{1 + g_m R_s}$

$= \dfrac{3 \times 10^{-3}(3k9//10k)}{1 + (3 \times 10^{-3} \times 680)} = 8.4/3.04 =$ **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.

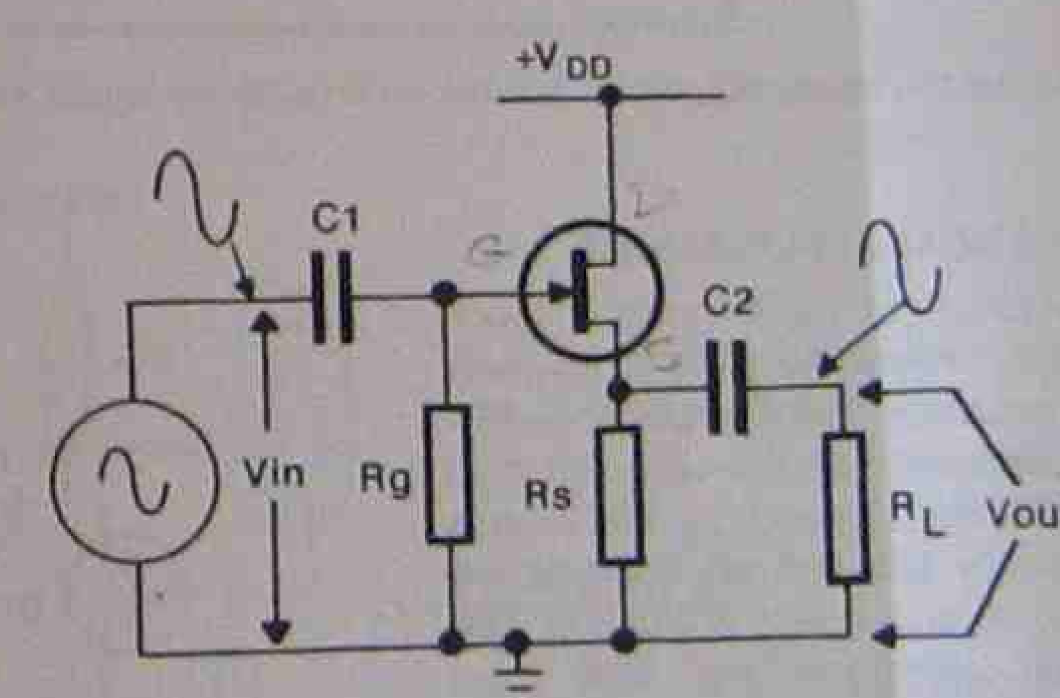


Fig.3: The common drain JFET amplifier

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:

□ $A_v = \dfrac{g_m (R_s // R_L)}{1 + g_m R_s}$

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:

□ $r_{in} = R_g$ ohms

The output resistance of the common drain amplifier is determined by:

□ $r_{out} = \dfrac{R_s}{1 + g_m R_s}$ 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 QI, 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.

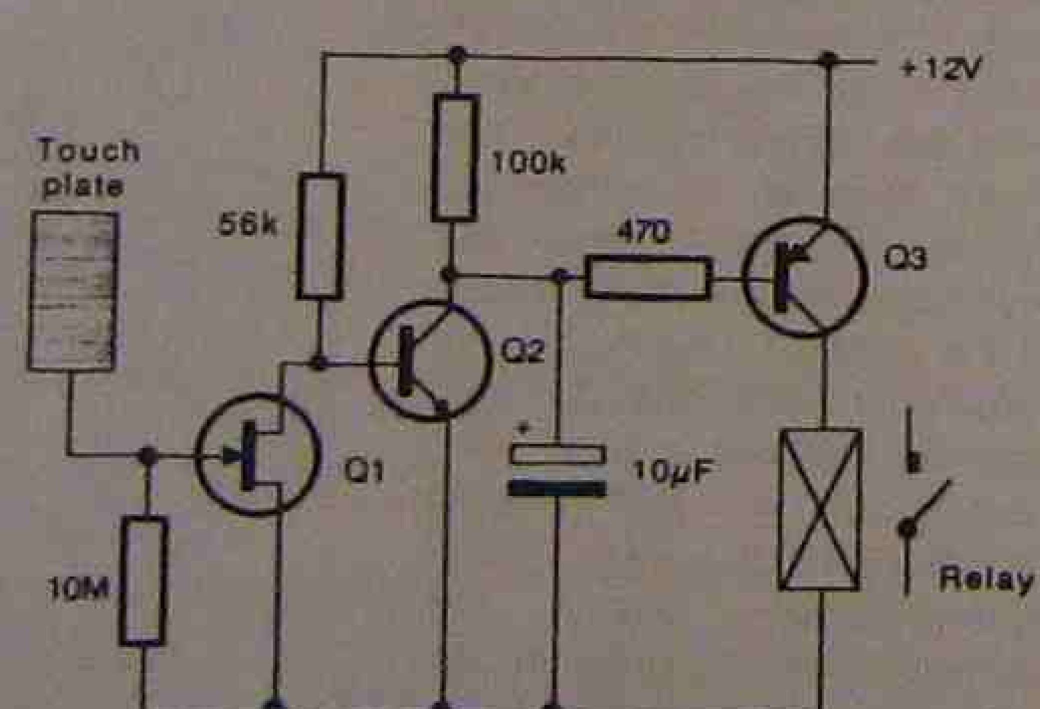


Fig.4: Touch switch with FET input stage

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR1 ELECTRONIC DEVICES

THEORY ASSIGNMENT 11

1. Forthe 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

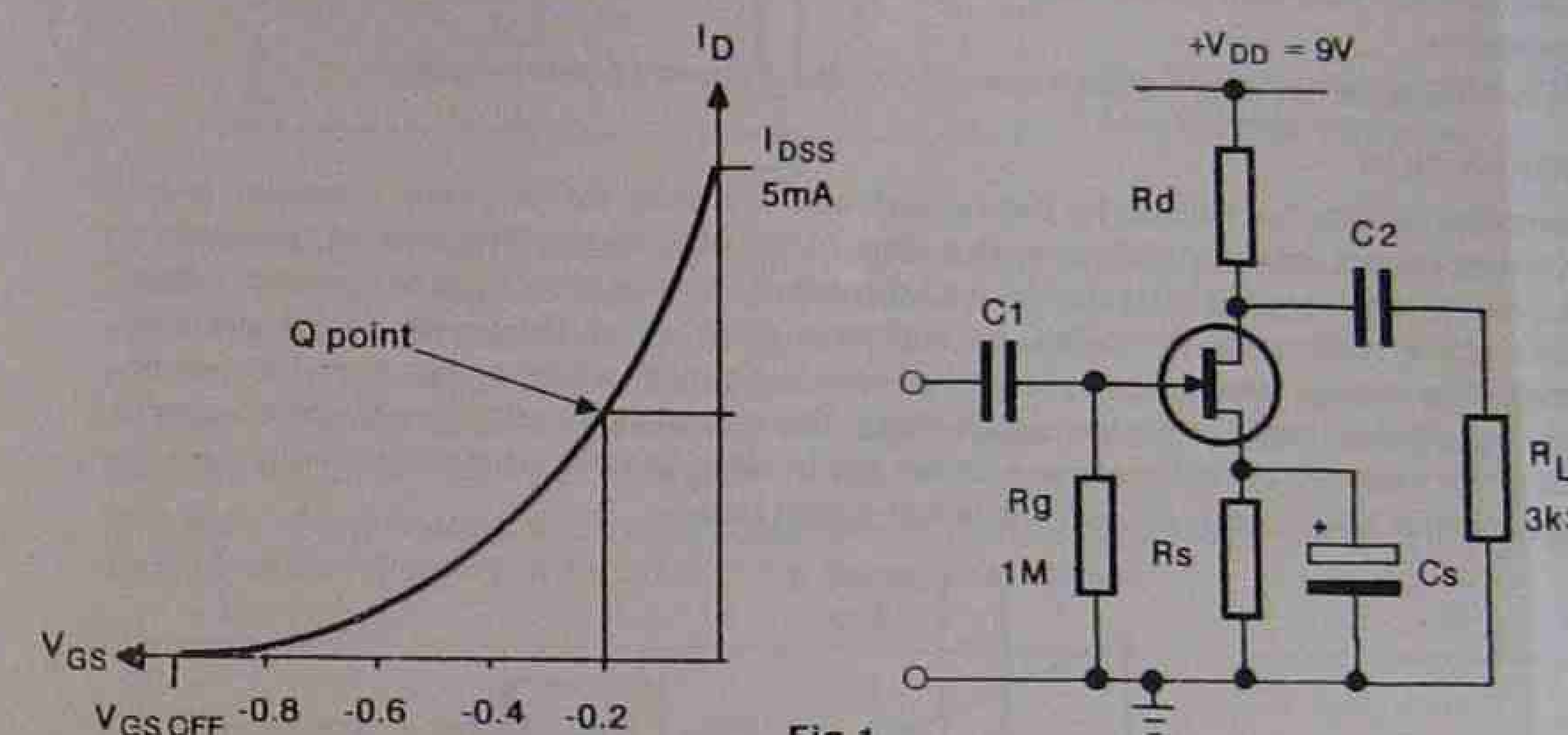


Fig.1

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.

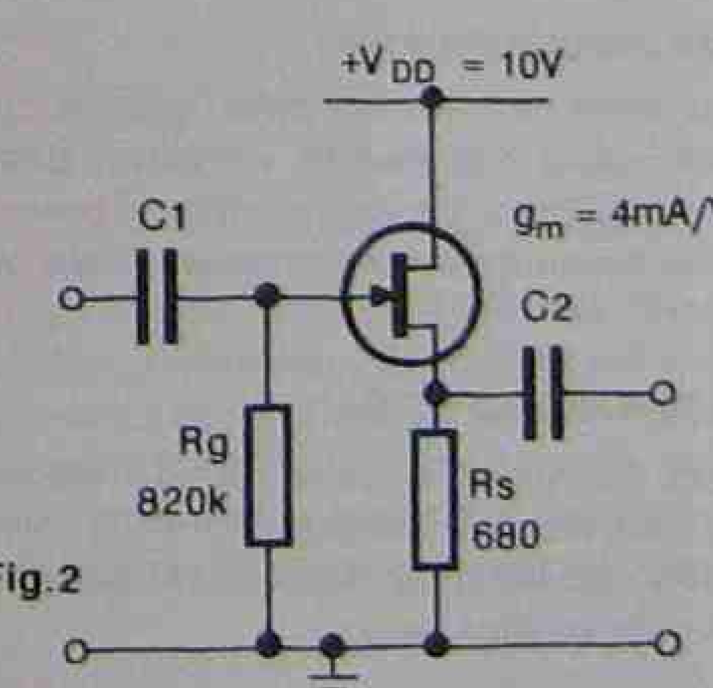


Fig.2

****

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.

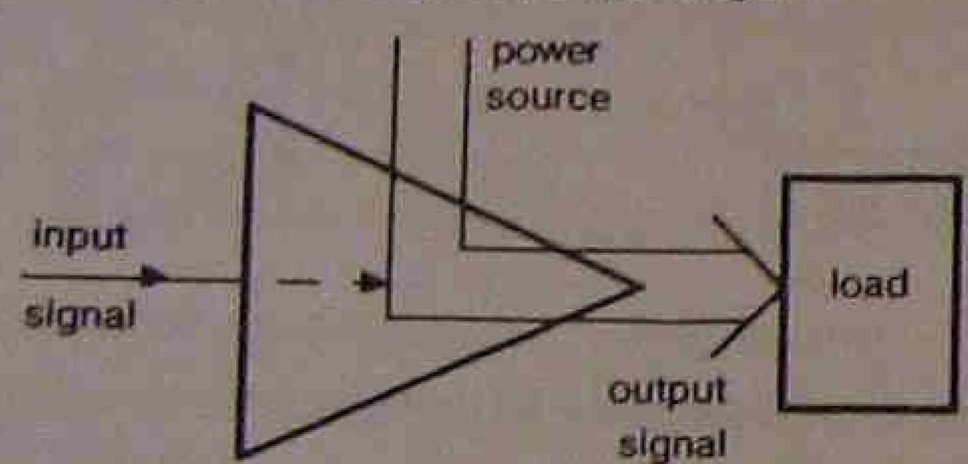


Fig.1: General form of a power output stage

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.

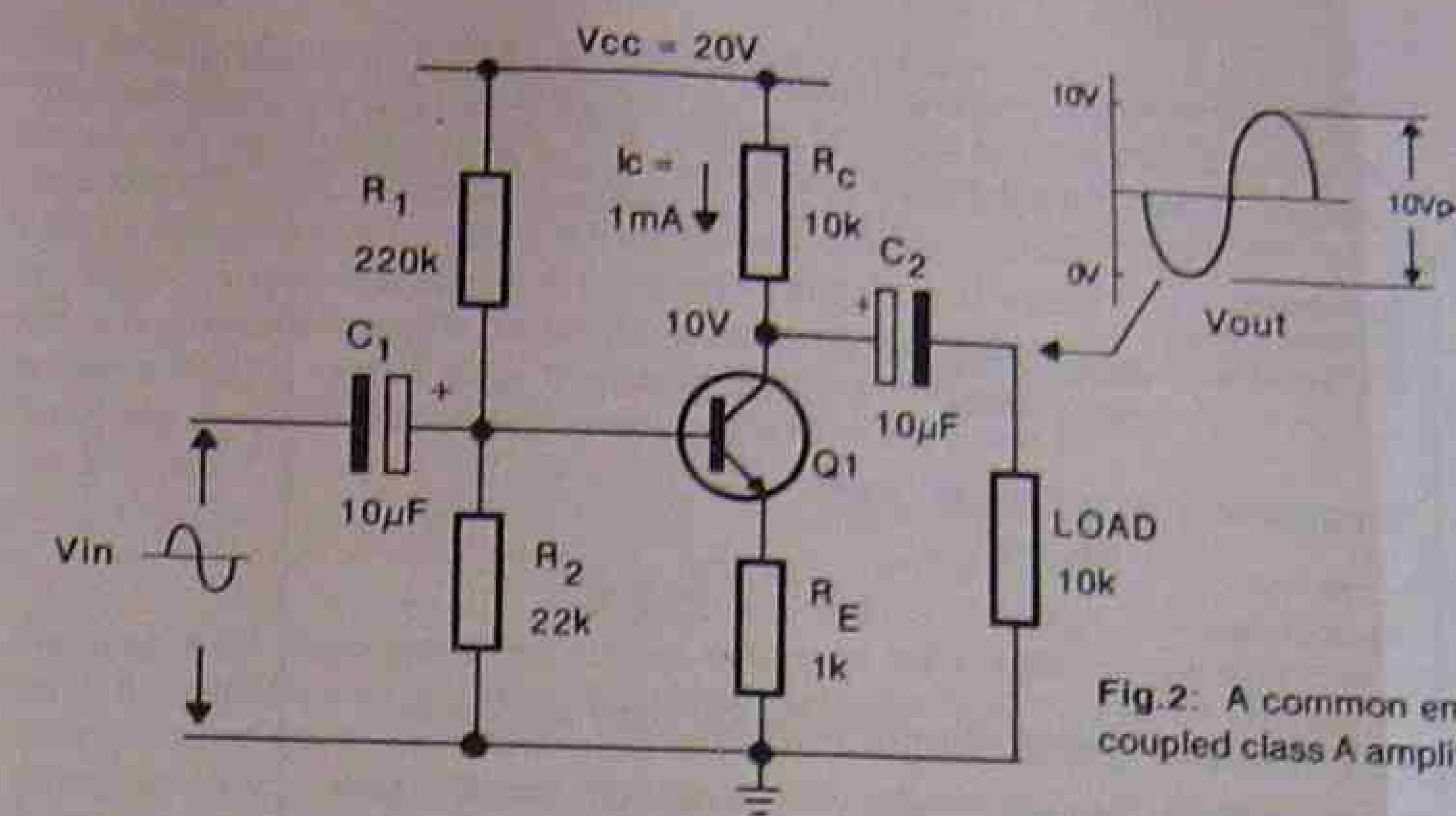


Fig.2: A common emitter, RC coupled class A amplifier

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:

$$\text{Efficiency\%} = \frac{\text{output power}}{\text{input power}} \times 100$$

$$\text{input power} = \frac{\text{output power}}{\text{efficiency}} \times 100$$

$$\text{input power} = \frac{50\text{kW}}{20} \times 100 = 250\text{kW}$$

If the system is supplied with a single phase, 240V AC supply,

$$\text{the line current (I) equals } \frac{\text{Pin}}{\text{V}} = \frac{250\text{kW}}{240\text{V}} = 1041.67\text{A}$$

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.

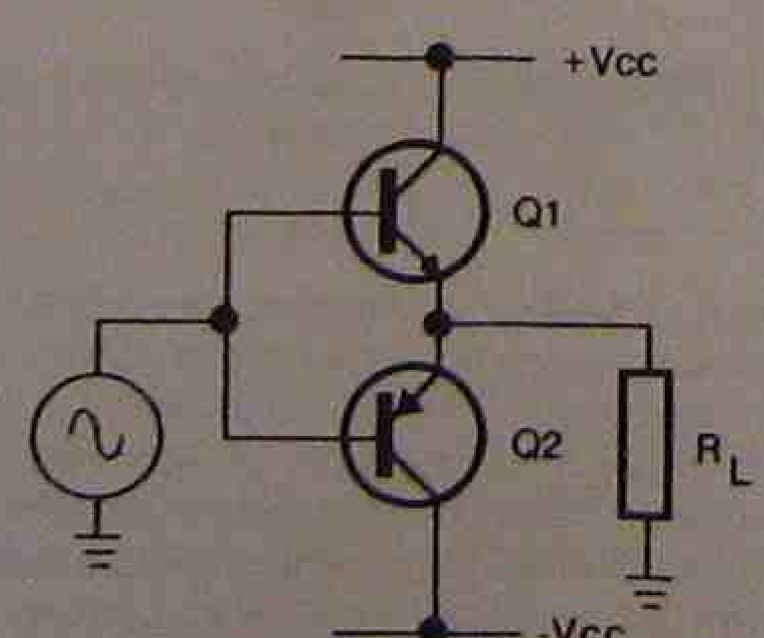


Fig.3: Class B output stage

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.

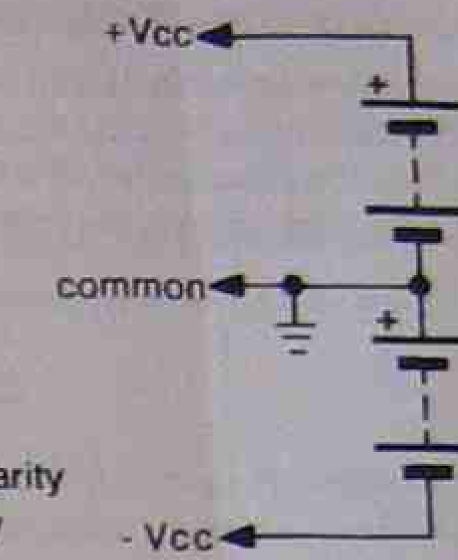


Fig.4: A dual polarity power supply

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.

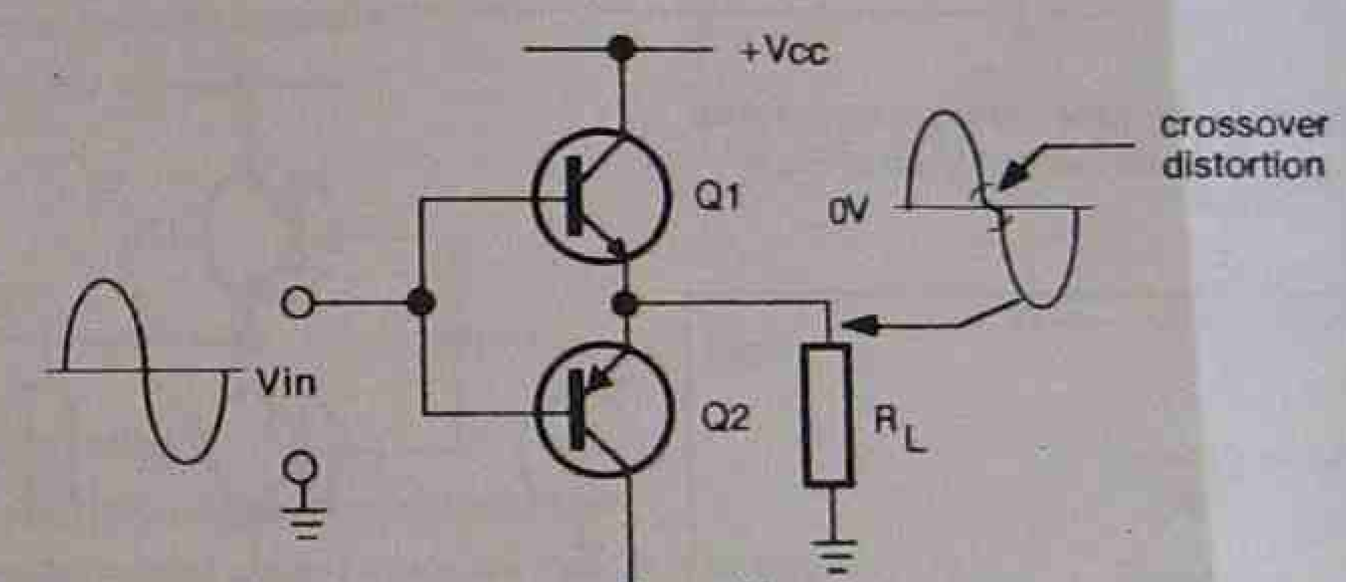


Fig.5: Crossover distortion

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.

Power output equation

$$P_{out} = \frac{(V_{orms})^2}{R_{load}}$$

$$(V_{orms})^2 = \left(\frac{V_{op\text{-}p}}{2\sqrt{2}}\right)^2 = \frac{V_{op\text{-}p}^2}{8}$$

□ $$P_{out} = \frac{(V_{op\text{-}p})^2}{8R_L}$$

EXAMPLE 2: Calculate the maximum output power for the circuit of Fig.6.

Solution:

$$P_{omax} = \frac{(V_{op\text{-}p})^2}{8R_L}$$

$$= \frac{(40)^2}{8 \times 10} = \frac{1600}{80} = \underline{20W}$$

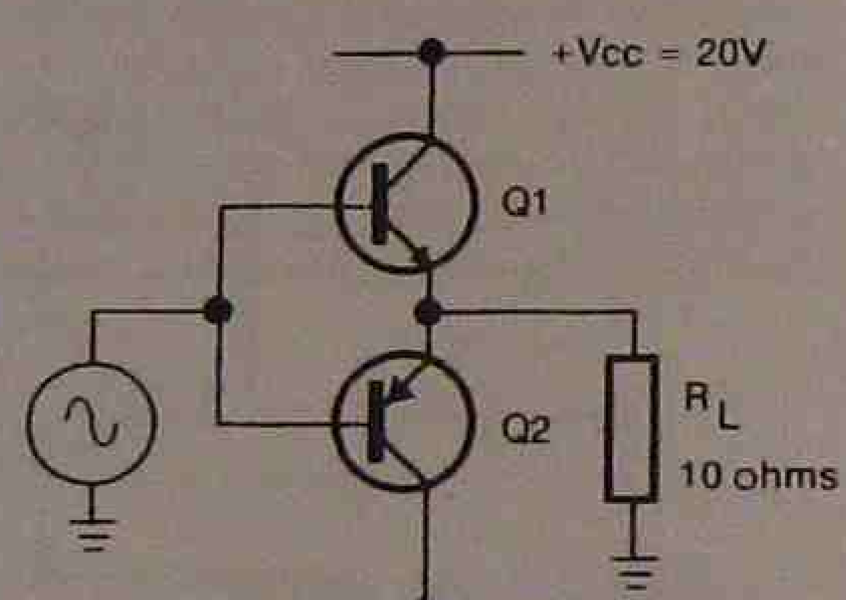


Fig.6

5. **VOLTAGE GAIN AND POWER GAIN OF THE CLASS B AMPLIFIER**

The circuit of Fig.6 is referred to as a Complementary Symmetry 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.

□ A_v = unity

□ A_p = current gain of transistors (β)

****

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.

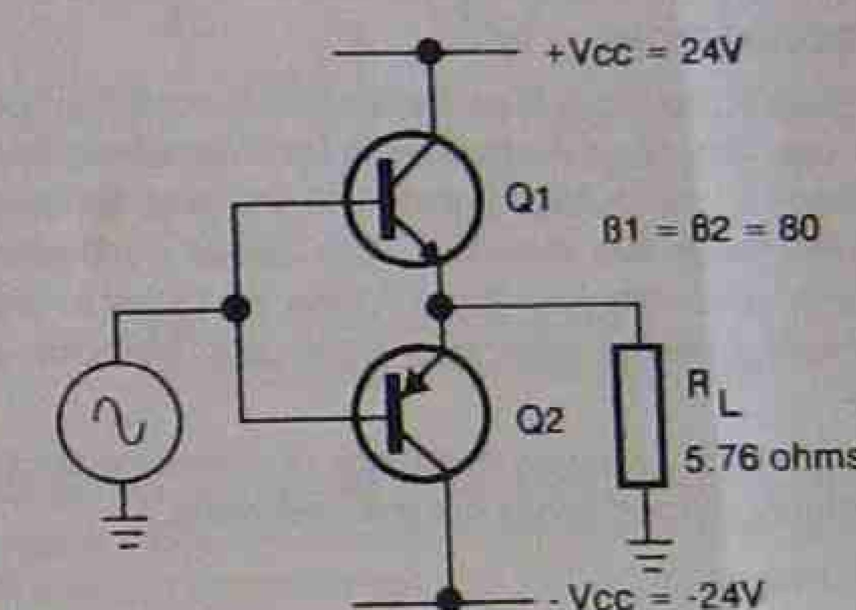


Fig.1

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.

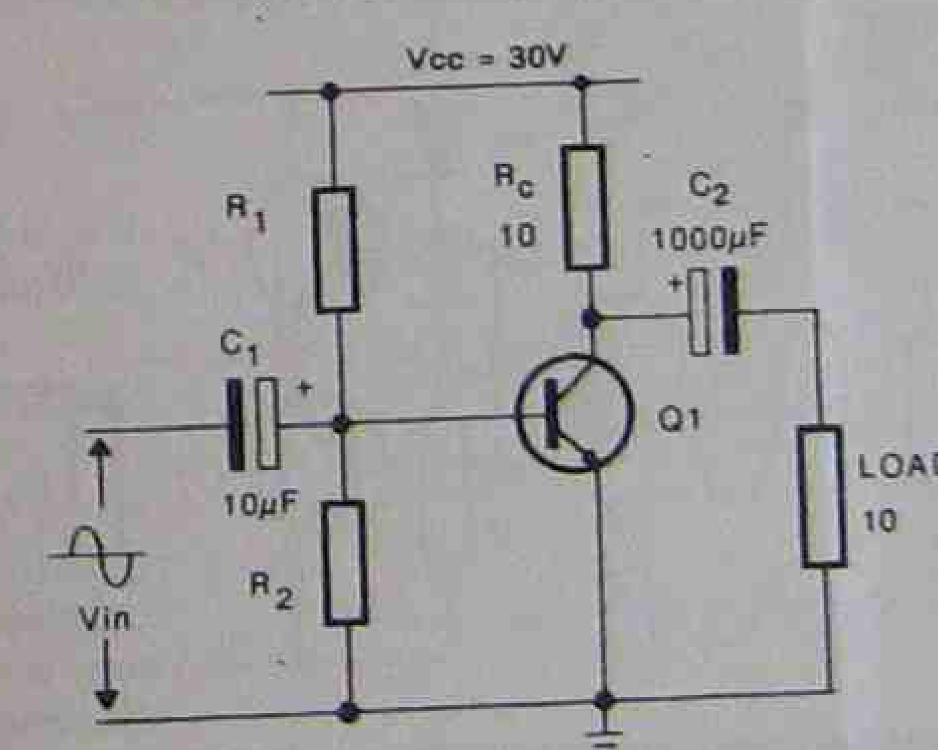


Fig.2

****

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

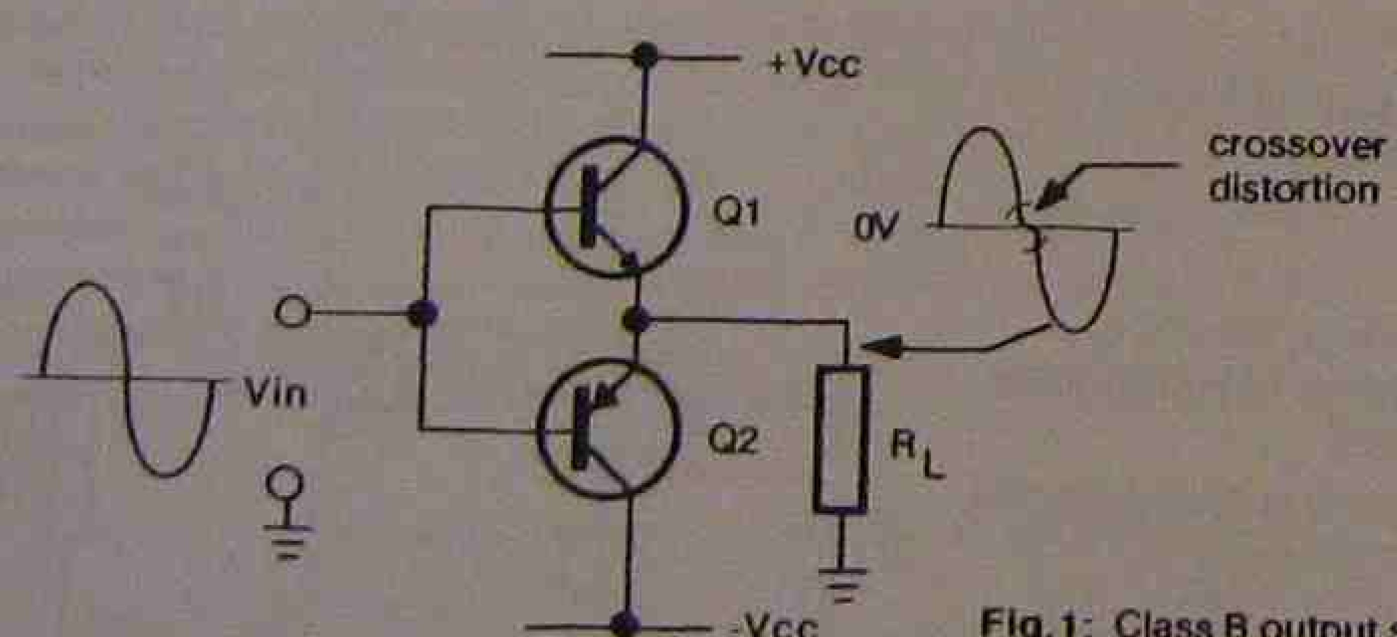


Fig.1: Class B output stage

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.

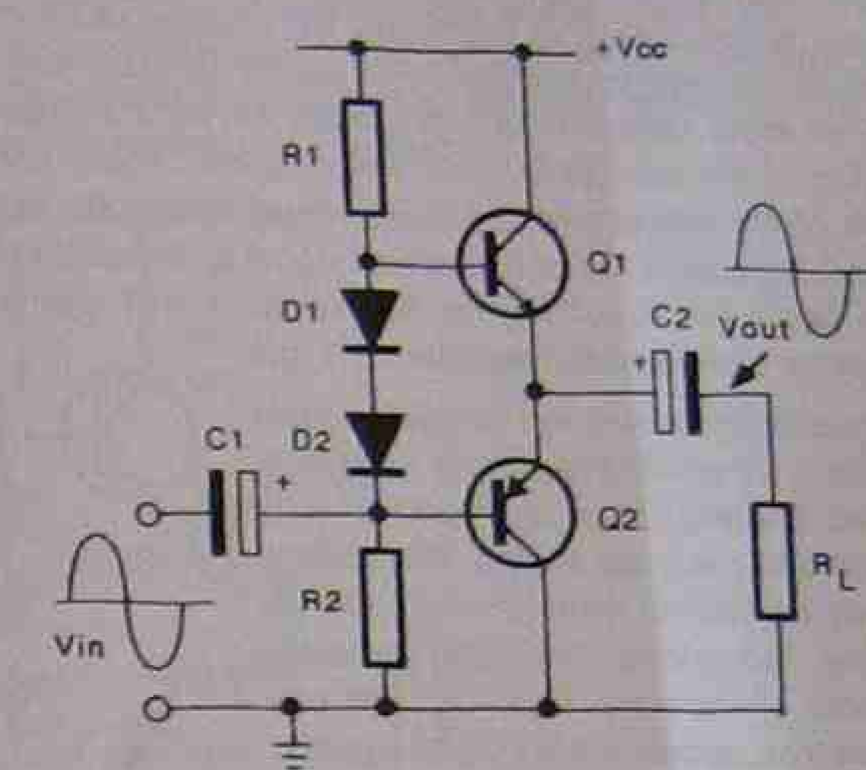


Fig.2: The basic class AB, complementary symmetry amplifier

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.

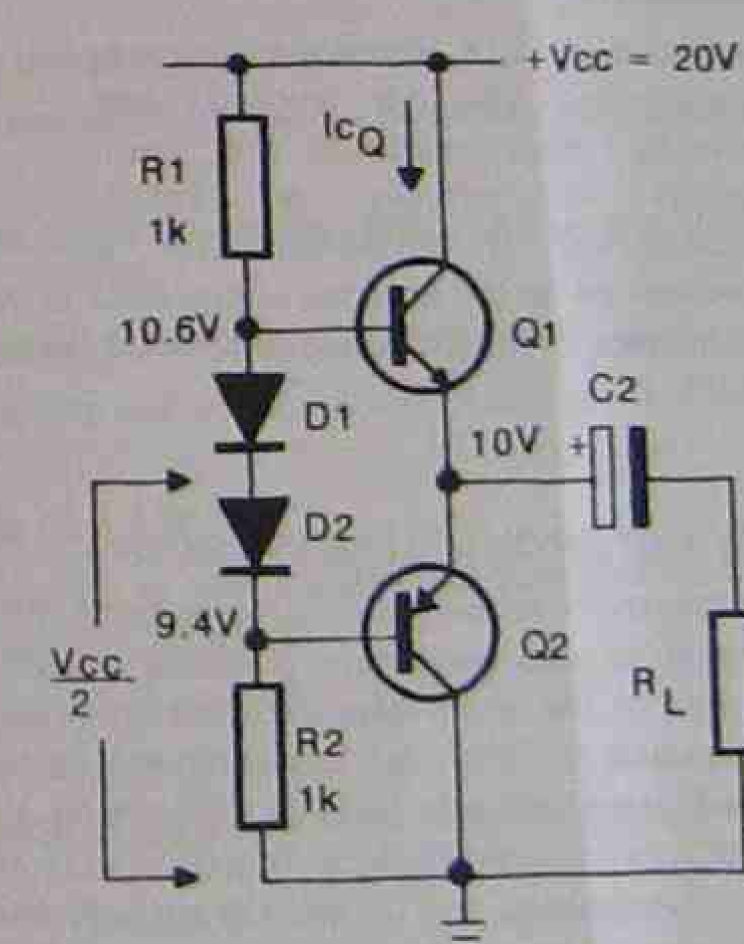


Fig.3: DC conditions

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!)

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.

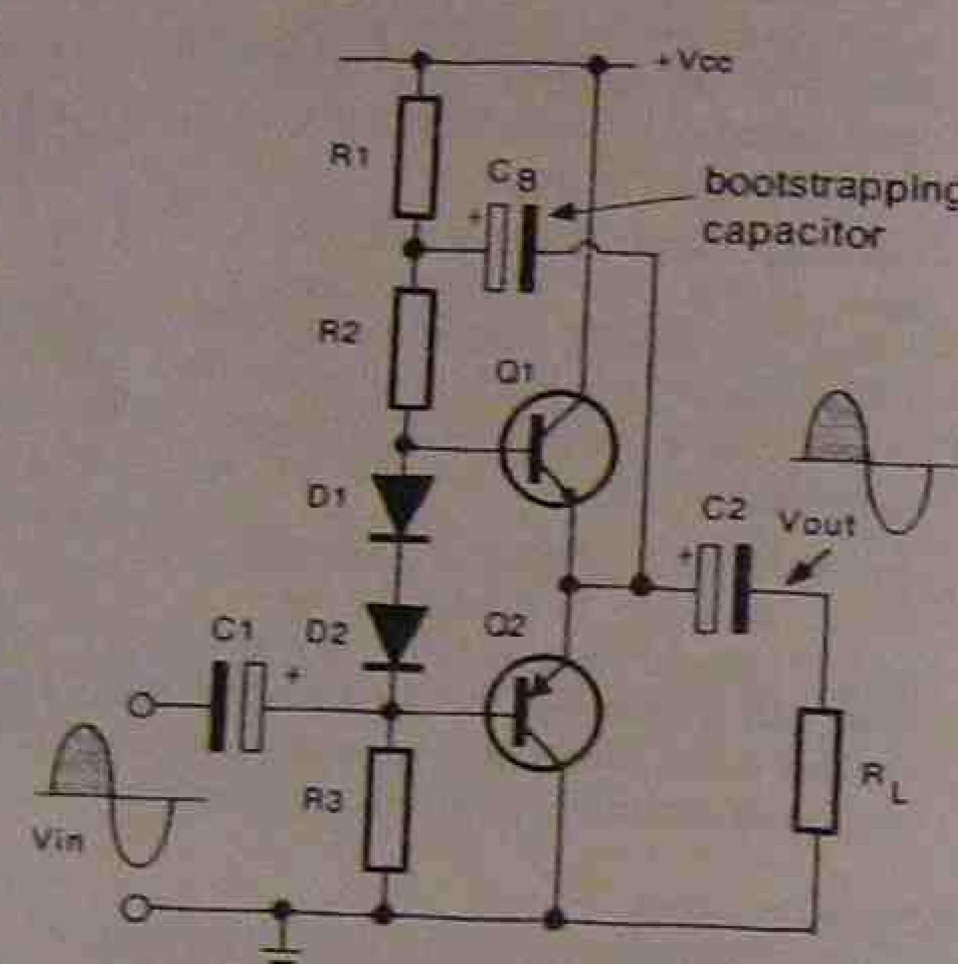


Fig.4: Adding bootstrapping to obtain a greater output signal swing

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.

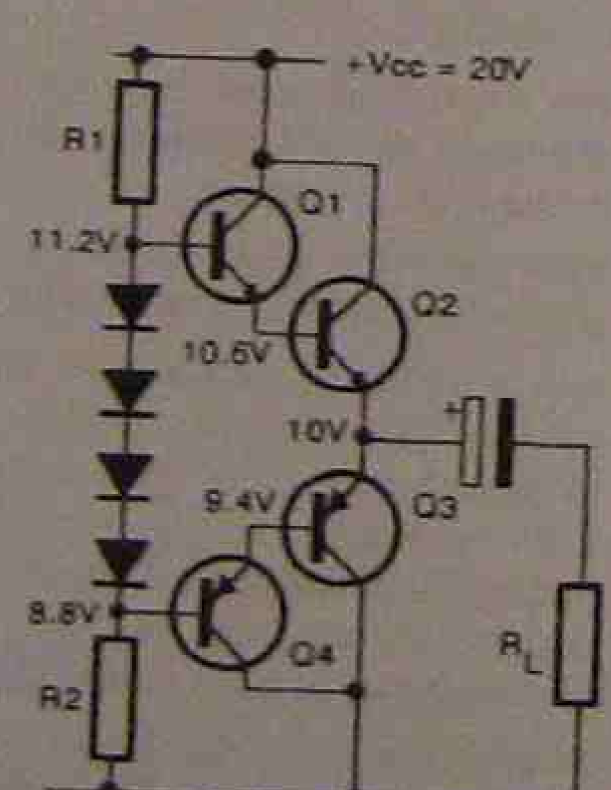


Fig.5: Darlington output

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.

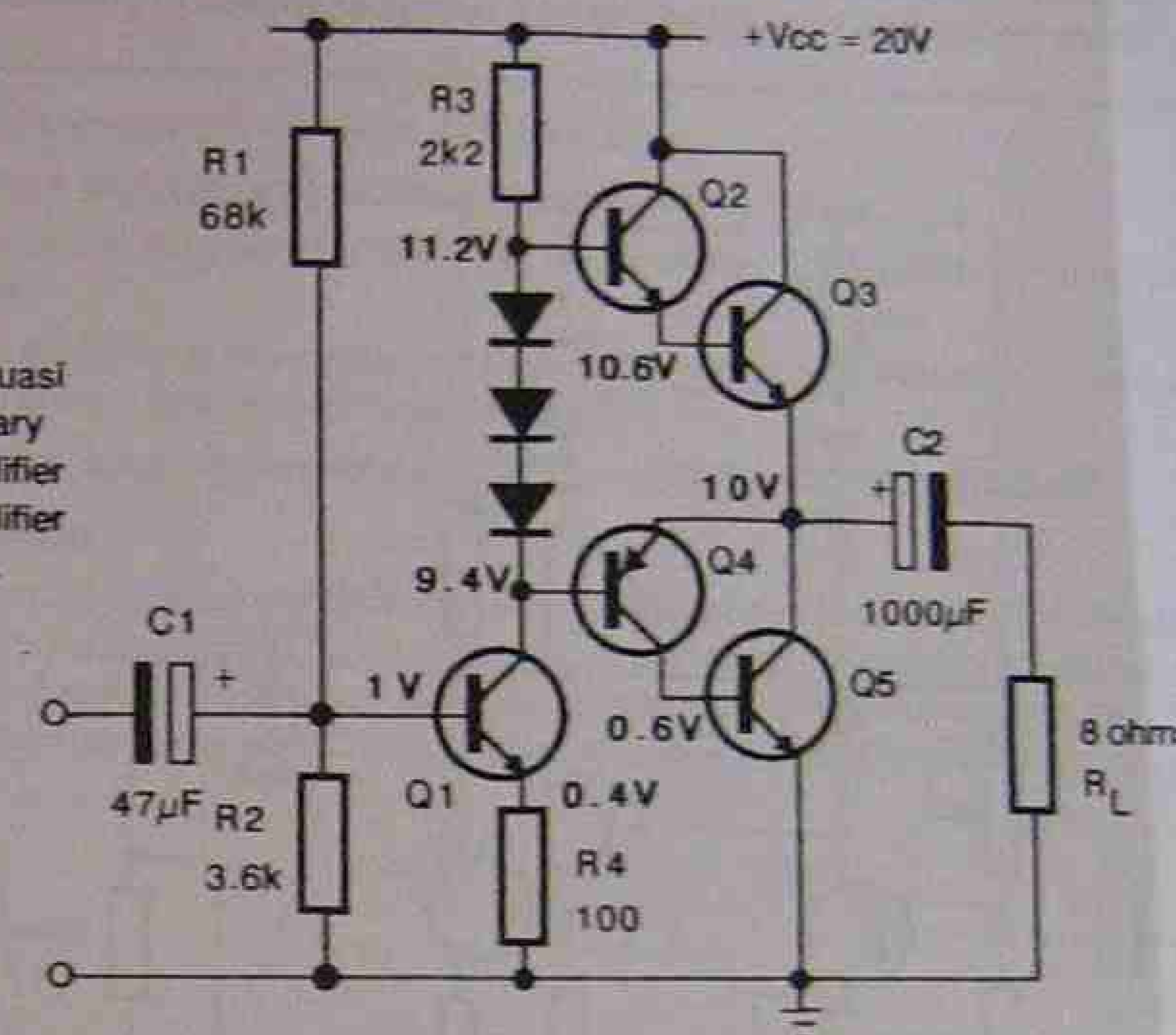


Fig.6: The quasi complementary power amplifier with an amplifier -driver stage.

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.

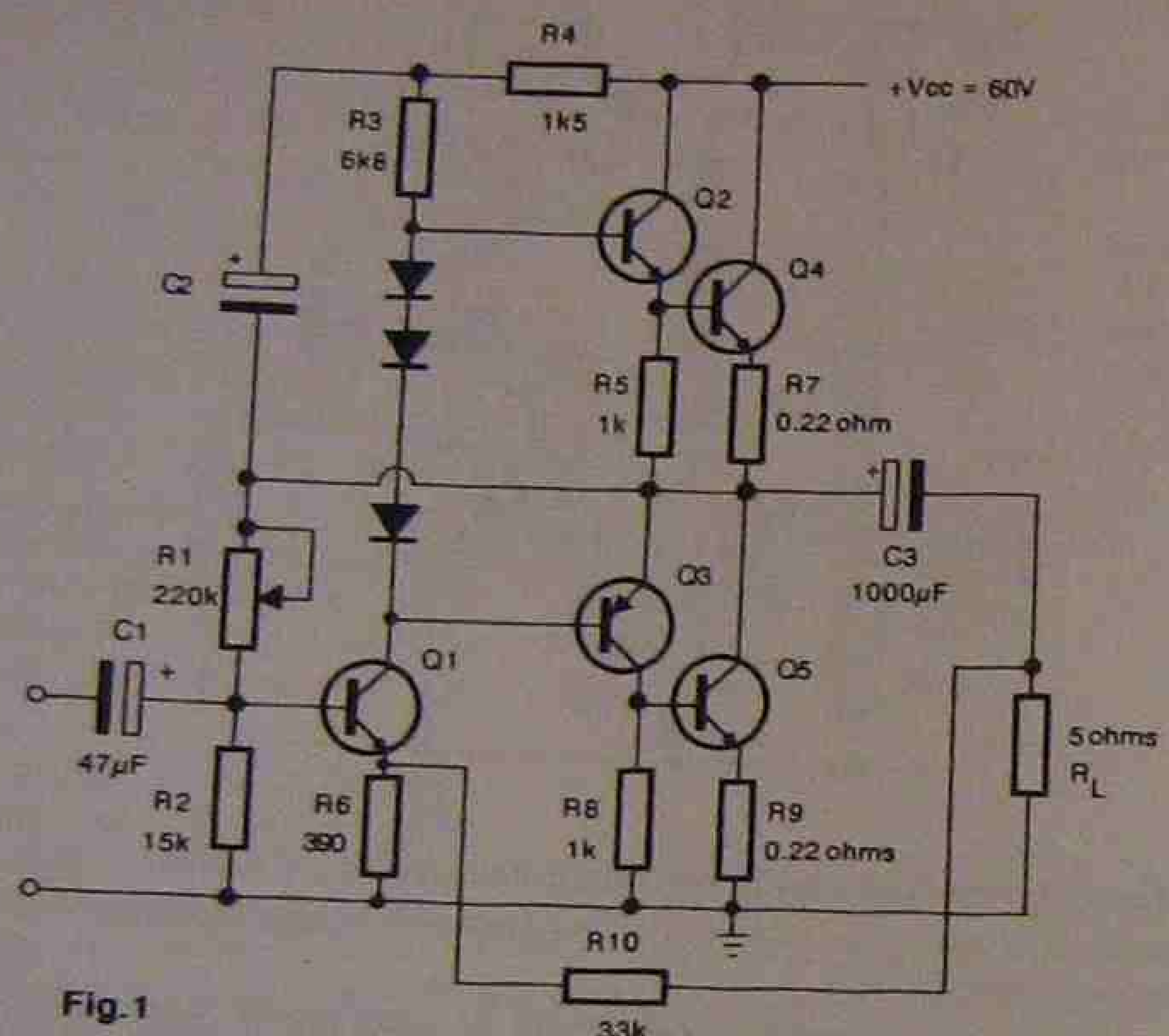


Fig.1

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

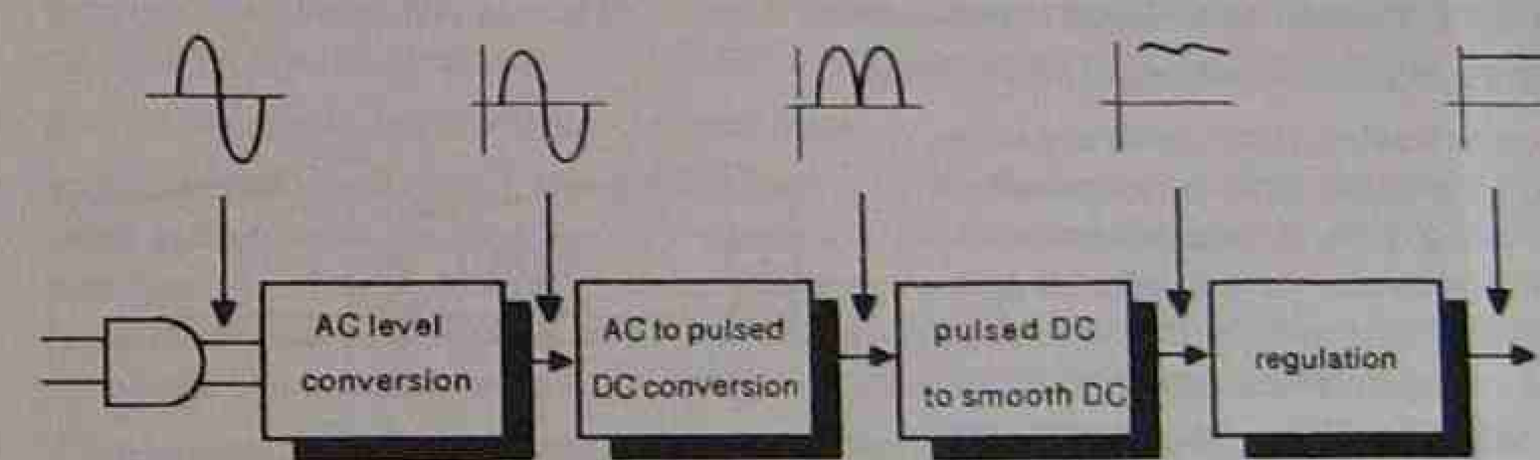


Fig.1: Block diagram of a regulated power supply

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

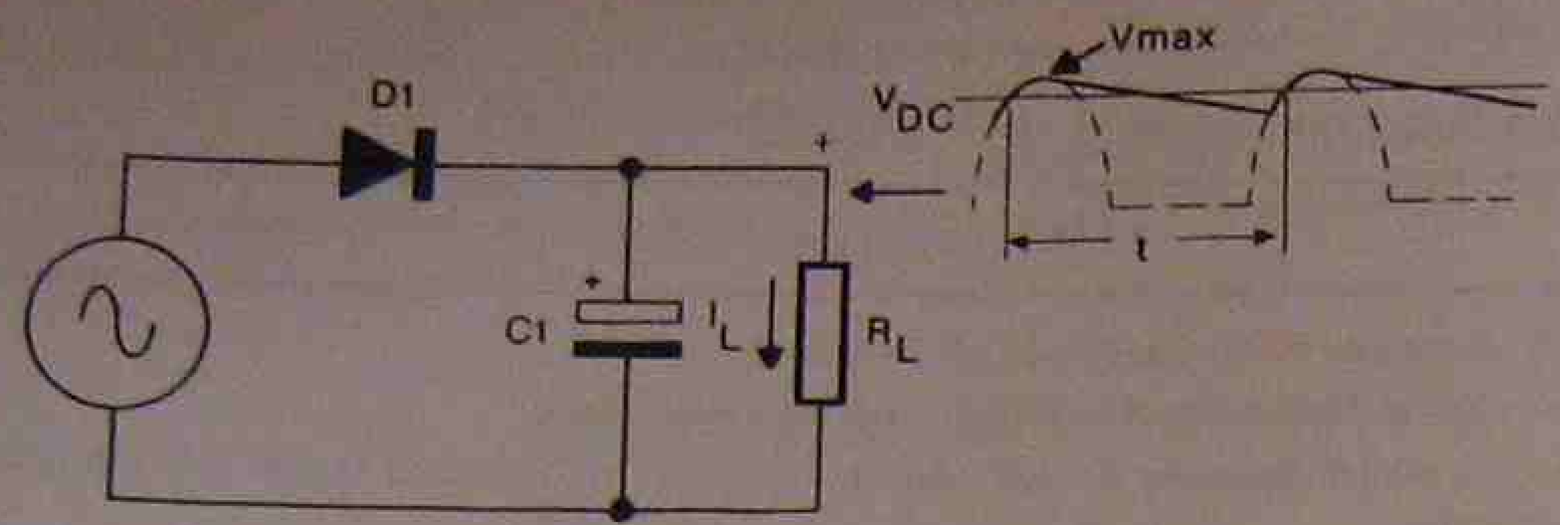


Fig.2: Half wave rectifier with filter capacitor

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.

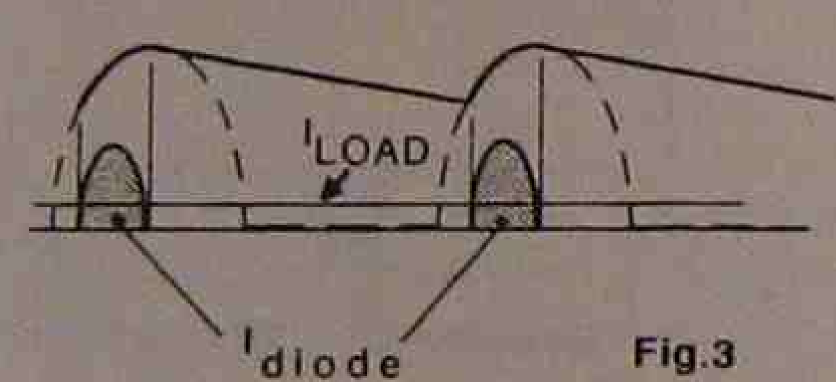


Fig.3

- 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

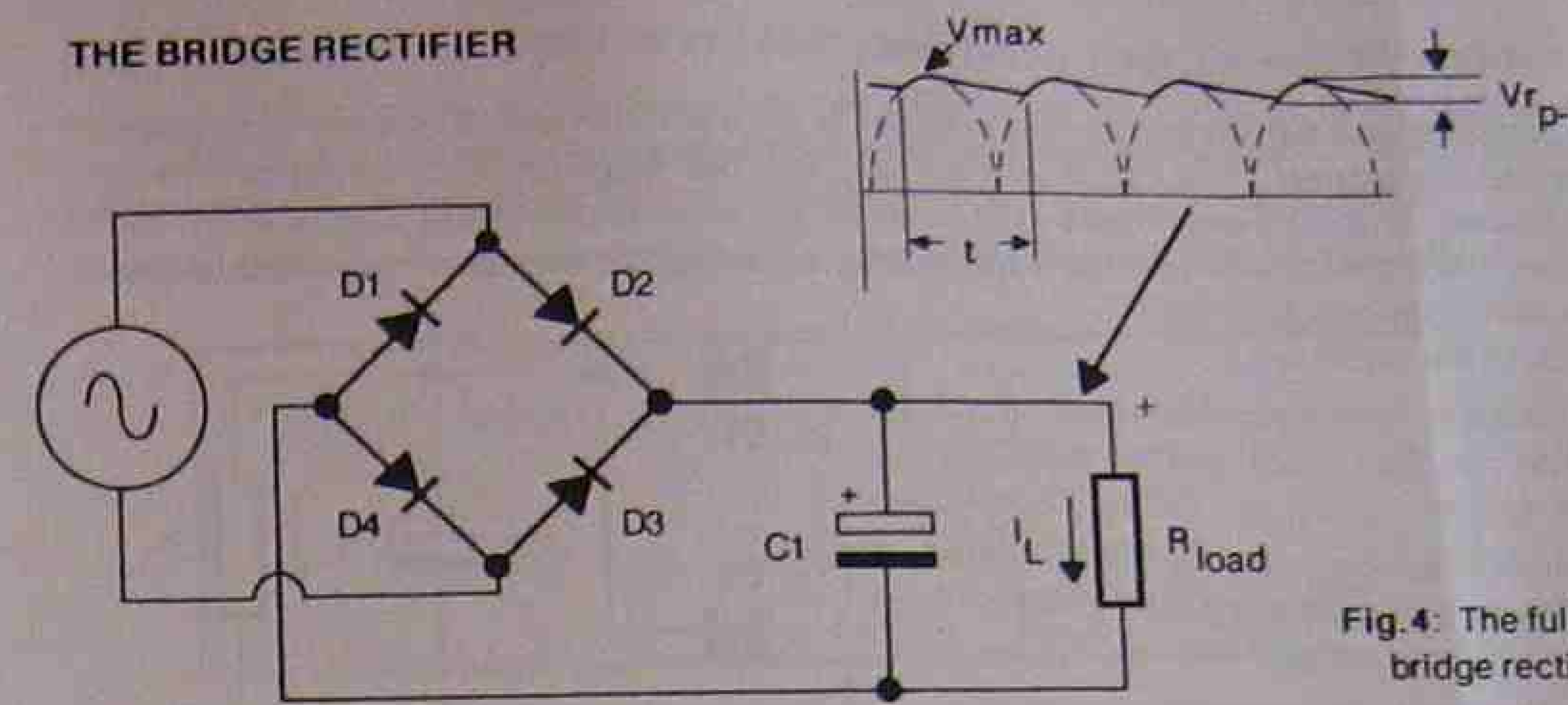


Fig.4: The full wave bridge rectifier

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:

- 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.

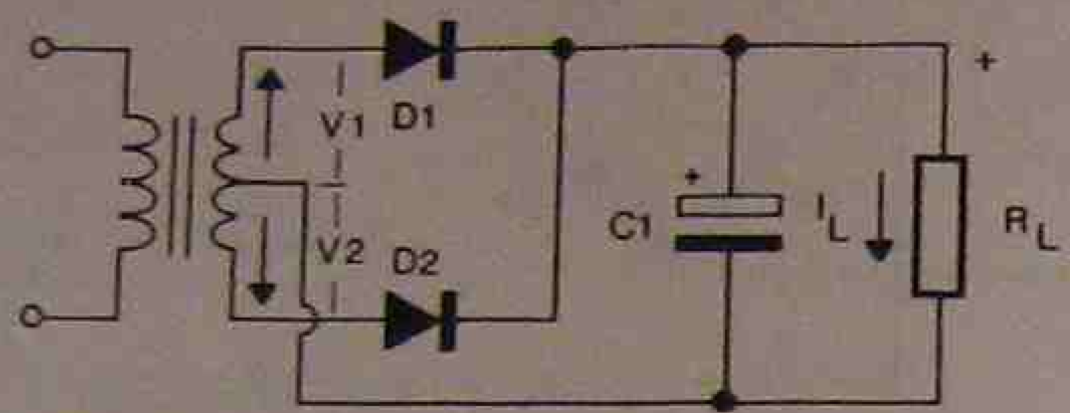


Fig.5: Centre-tapped transformer, full wave rectifier

  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 (V_r) 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:

$$R_L = \frac{V_O}{I_L} = \frac{40}{0.25} = \underline{160\ ohms}$$

$$V_r = 40V \times \frac{2}{100} = \underline{0.8V}$$

$$V_{DC} = V_r f_r R_L C \quad or \quad C = \frac{V_{DC}}{V_r f_r R_L} = \frac{40}{0.8 \times 100 \times 160} = \underline{3125\mu F}$$

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:

□ **% load Regulation** $= \frac{V_{DC}(no\ load) - V_{DC}(full\ load)}{V_{DC}(full\ load)} \times \frac{100}{1}$

Thus, a power supply with a no load voltage of 20V and a full load output of 18V has a load regulation of: $\frac{20 - 18}{18} \times 100 = \underline{11.1\%}$

****

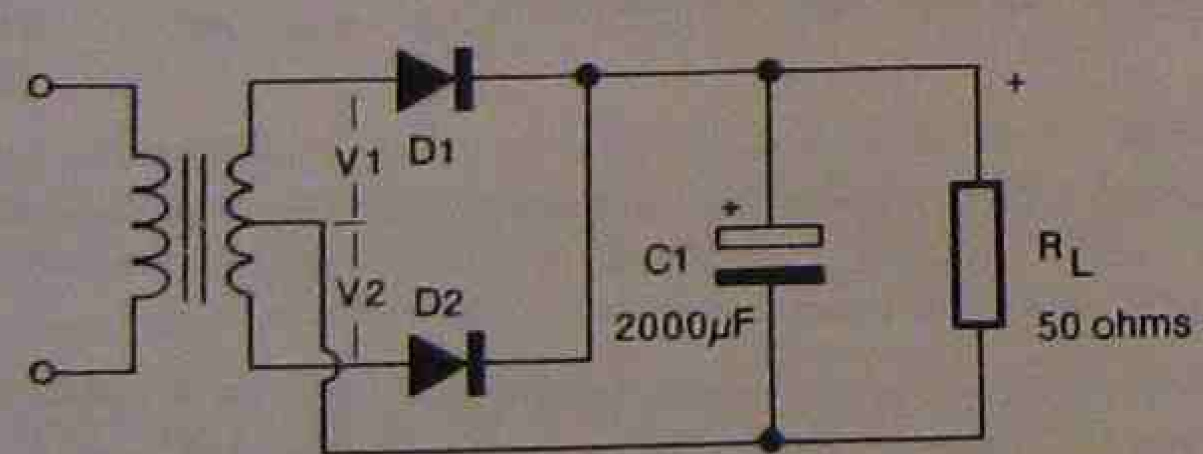
THEORY ASSIGNMENT WEEK 15

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.



$V1 = V2 = 10V_{RMS}$ Fig.1

****

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 (I_F) 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.

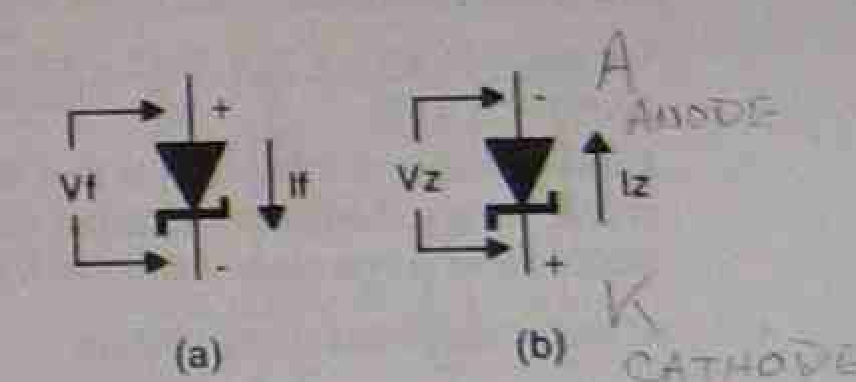


Fig.1: The zener diode

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.

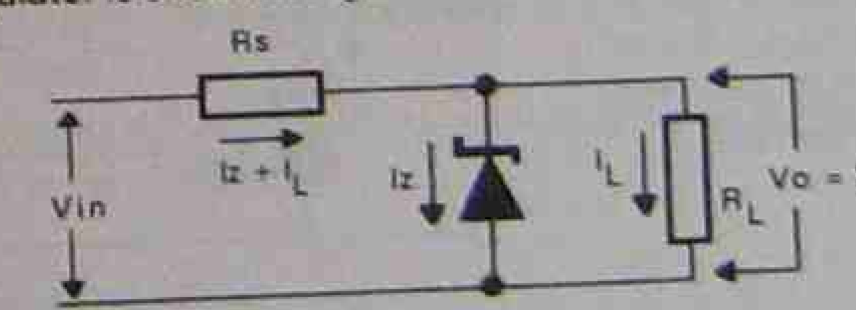
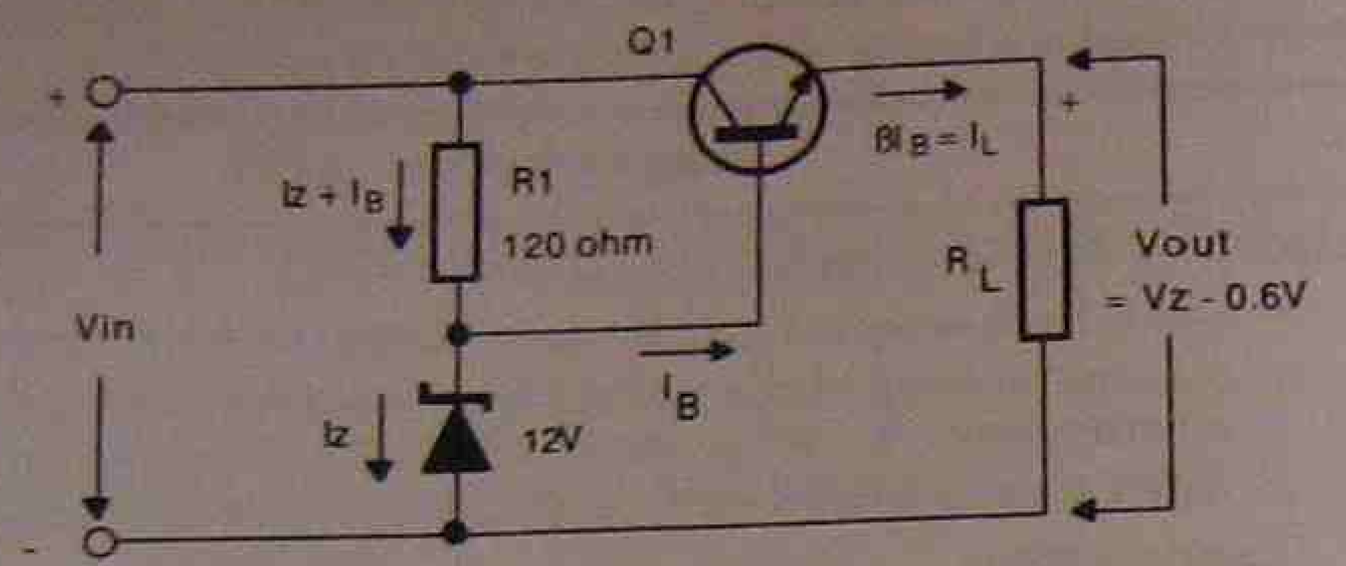


Fig.2: Basic series zener regulator

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.

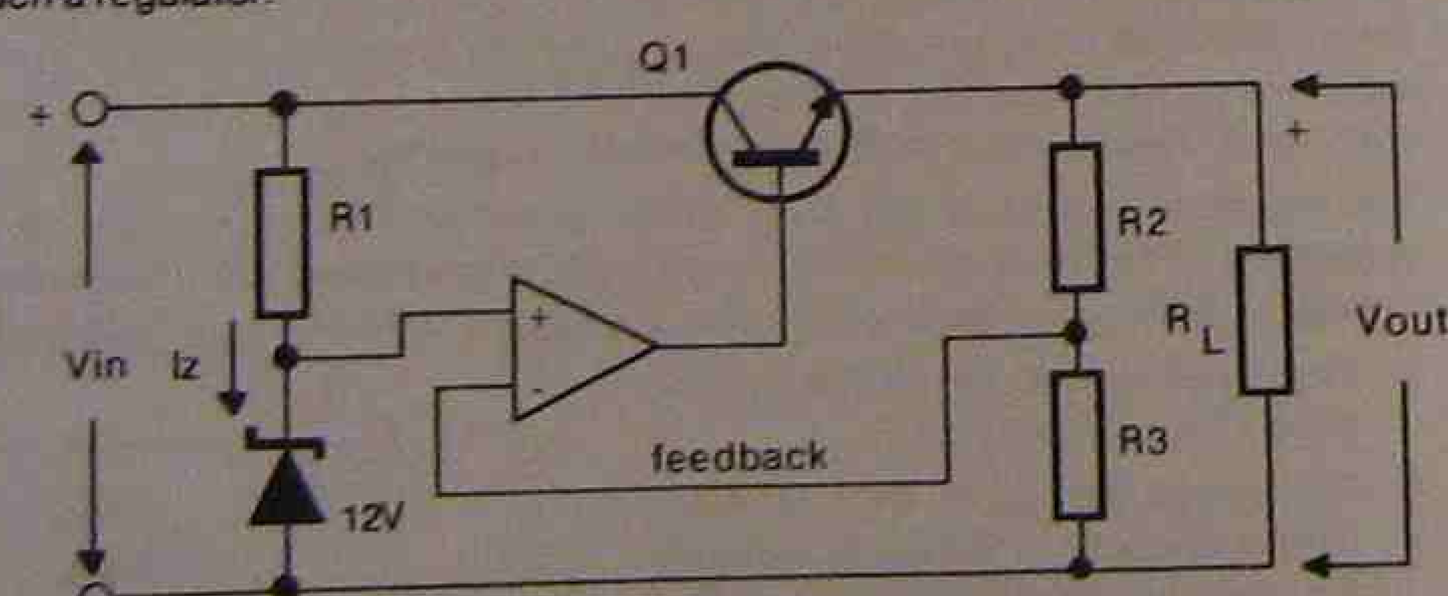
For example, if Vin = 20V, Vout = 11.4V and I_L = 1A, the power dissipated by Q1 will equal (20 - 11.4) x 1 = 8.6W.

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:

- $Vout = \left(\frac{R2}{R3} + 1\right) \times Vref$ where: Vref = Vz
- $Pd = \left(\frac{Vin - Vout}{I_L}\right) \times I_L$

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.

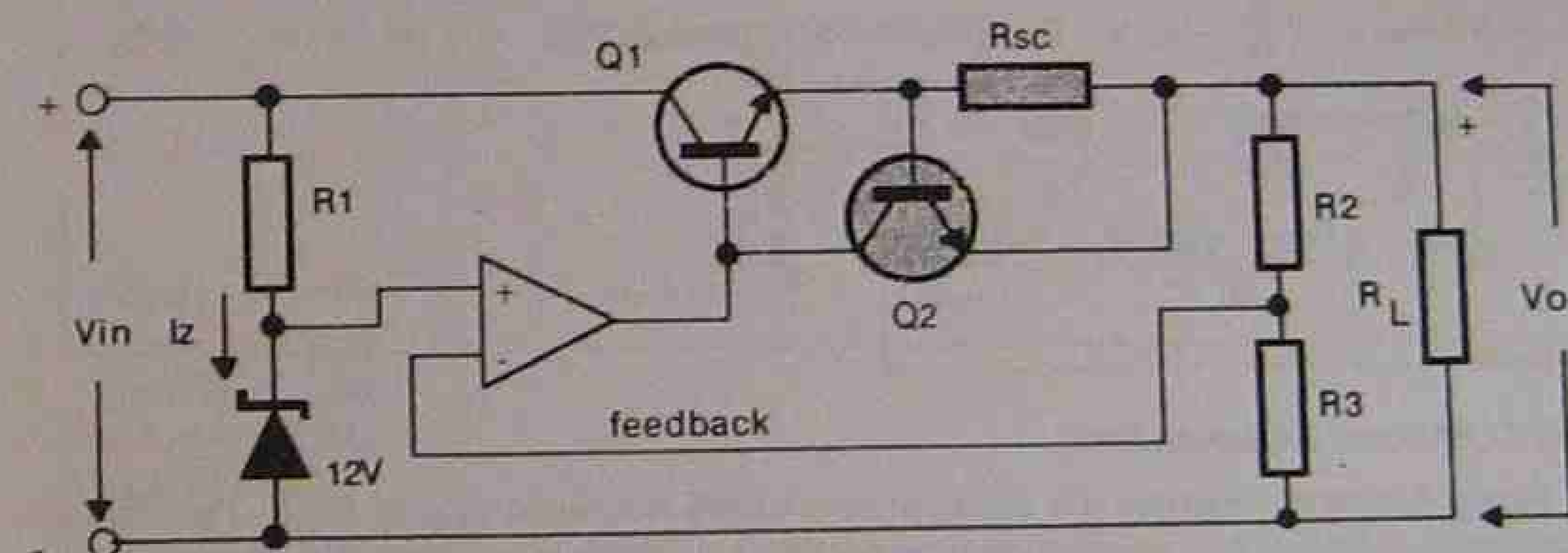


Fig.5: Series regulator with current limiting

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

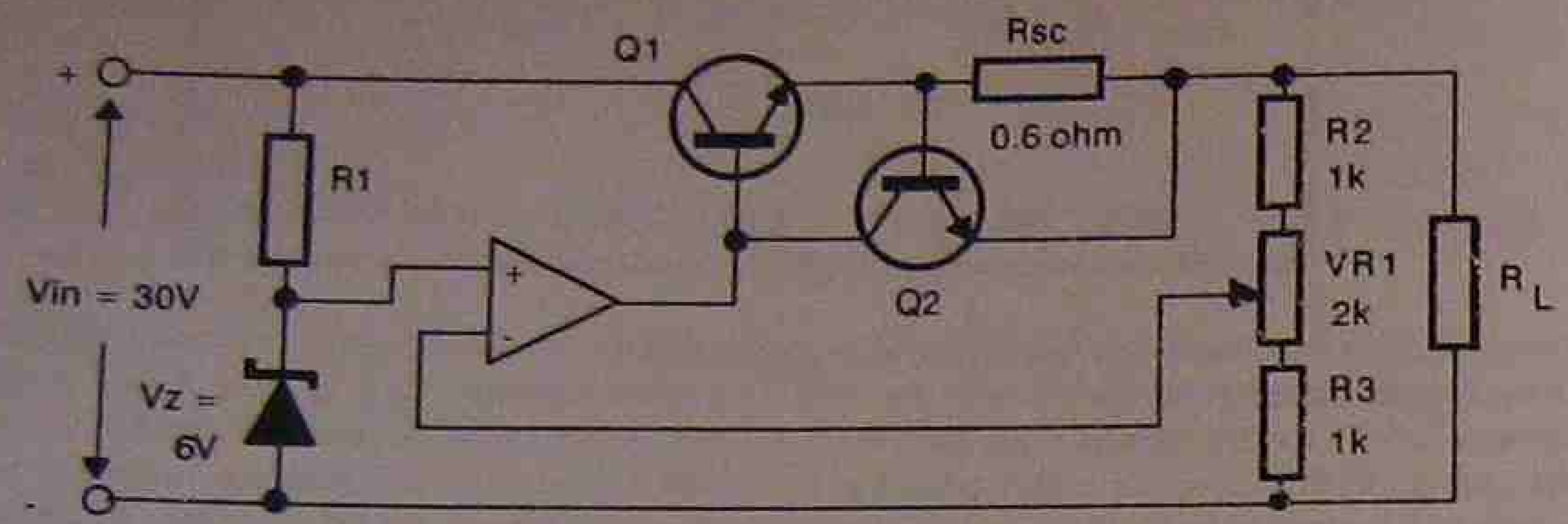


Fig.6

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) $I_{sc} = \frac{0.6V}{R_{sc}} = \frac{0.6V}{0.6} = \underline{1A}$

(c) $P_d = (V_{in} - V_{out}) I_L = 1(30 - 0) = \underline{30W}$

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	VIN (max)	IOUT (A) (nom)	PTOT (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	VOUT +40V	1.5	20		78G

Note: Replace XX by output voltage e.g. - 7812, 7905 etc

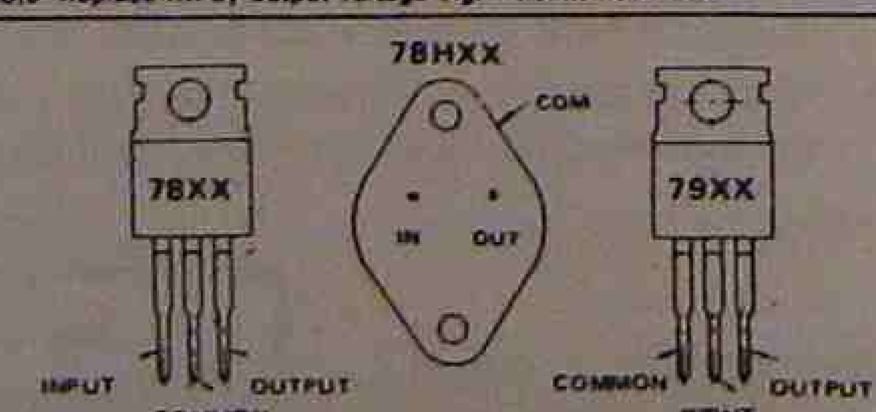


Fig.7: Three terminal, fixed voltage regulators

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.

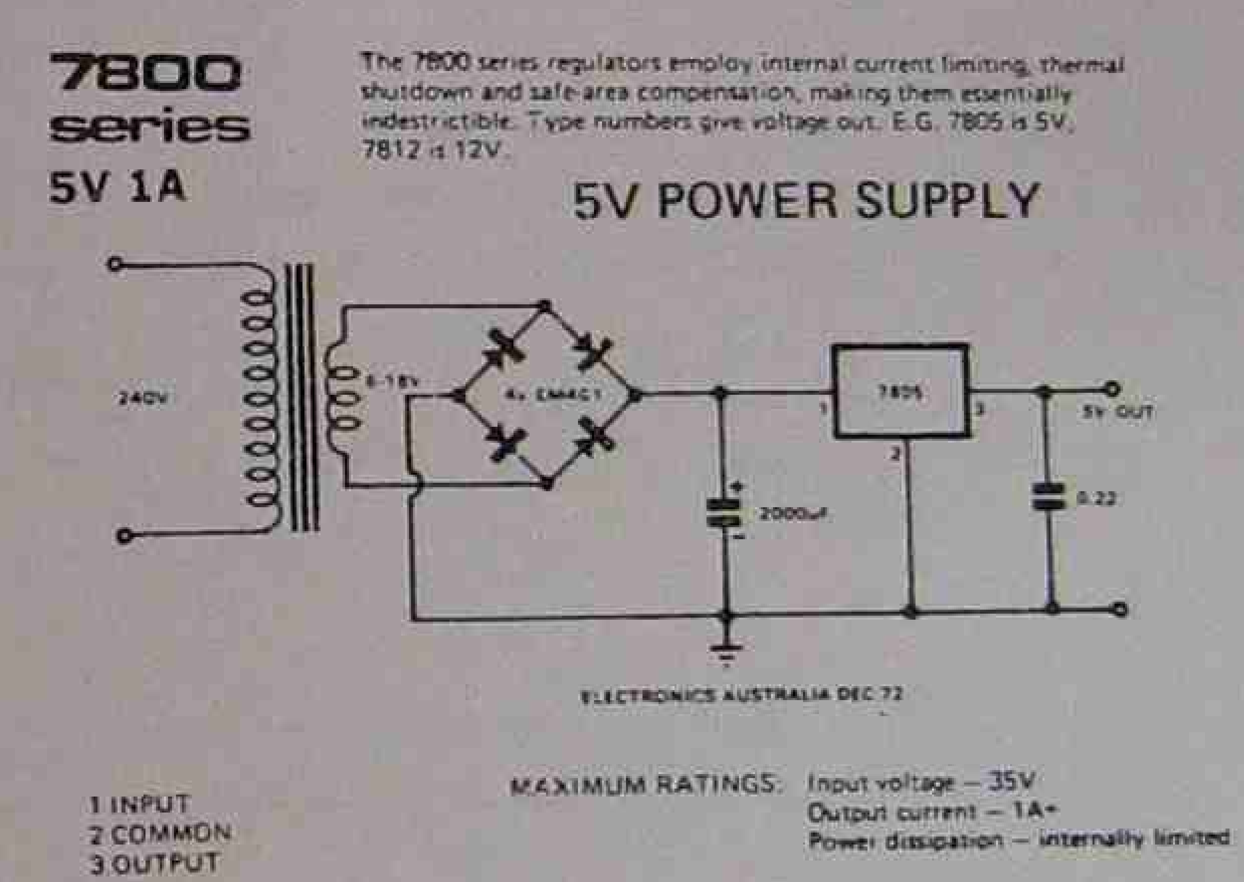


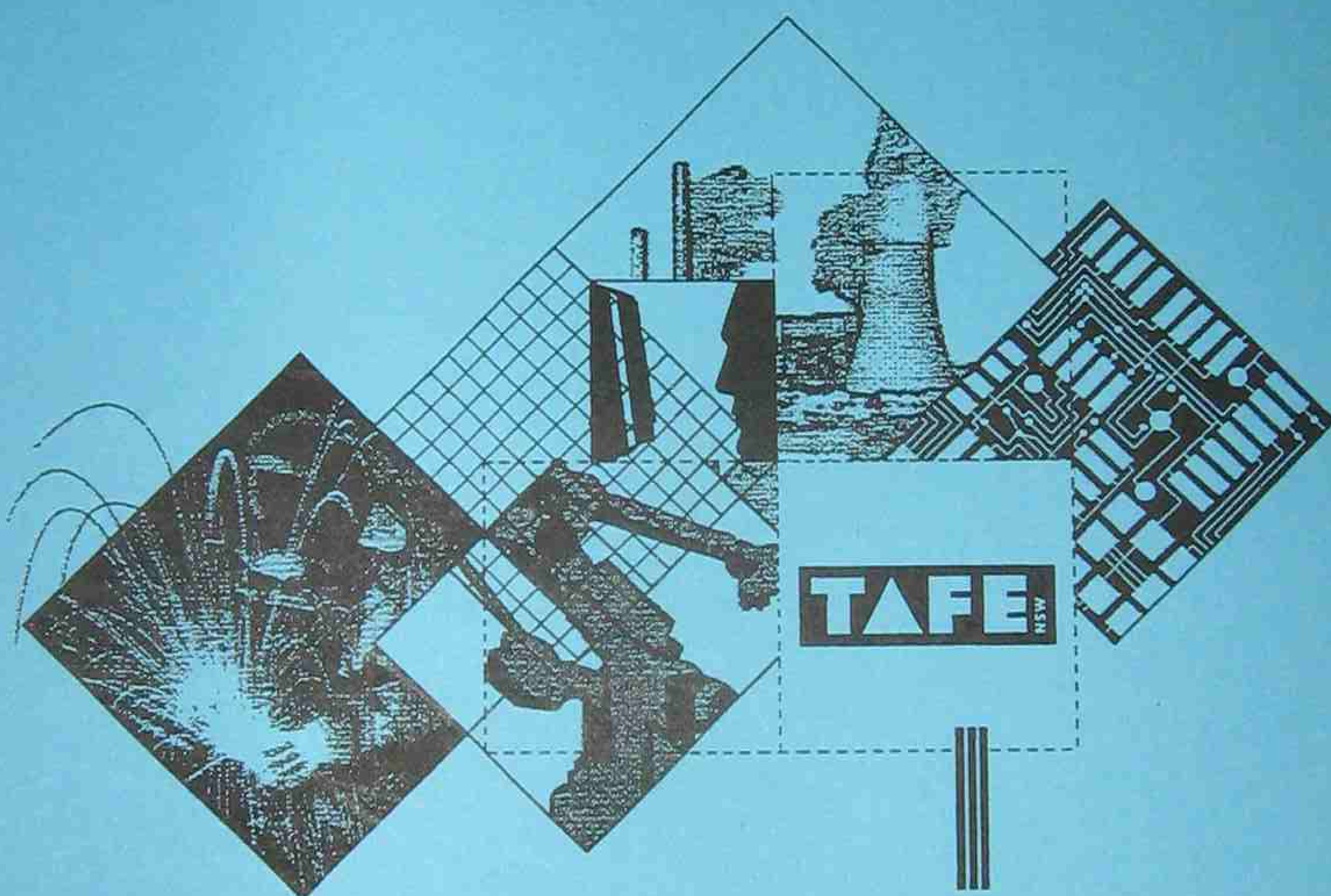
Fig.8: A 5V power supply suitable for powering a TTL digital circuit

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.

****

NSW Module Resource Manual for the TAFE Engineering Technician and Engineering Associate National Curriculum



Electrical/Electronic Stream

Analogue Electronics 1

Student Workbook

National Module Code: EA100

NSW Module Number: 7761A

First published in January 1996 by
TAFE Engineering Services
Industry Training Division
PO Box 218 Bankstown NSW 2200

National Library of Australia pub'n no:
ISBN 0 7310 6191 8

ANALOGUE ELECTRONICS 1
EA100/7761A

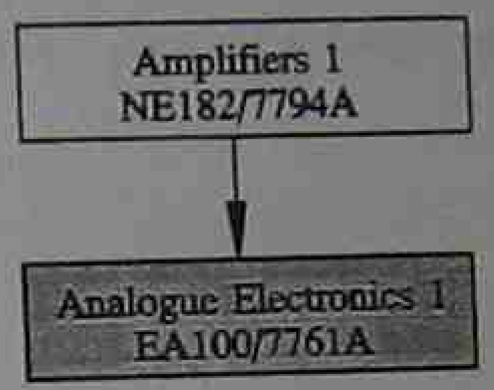
CONTENTS

Notes

About this module

This module is a part of the electrical/electronic stream in the TAFE National Engineering Technician and Engineering Associate Curriculum.

The prerequisite for this module is NE182/7794A Amplifiers 1. The diagram below shows the place of this module Analogue Electronics 1 in the telecommunications engineering stream.



This module extends the knowledge in the prerequisite modules by studying the performance limitations and specifications of operational amplifiers. On completion of this module you will have developed an understanding of the design, analysis and selection of workable substitutions in circuits using modern operational amplifiers and analogue integrated circuits.

This workbook is not a substitute for the recommended text and references, but it will help you by providing technical information in this subject area.

The review questions and skill practice exercises will help you find out how well you have understood the topic and help you prepare for the competency tests.

The module is divided into five sections. They are designed to be worked through progressively. The sections are:

- Basic op amp circuits
- DC non-idealities
- Slew rate
- AC noise
- Frequency compensation

What you will need for this module

Recommended text
Jacob, J. Michael. *Applications and Design with Analog Integrated Circuits* (second edition). Regents/Prentice-Hall, 1993.

Additional references
Coughlin, Robert F. and Frederick F. Driscoll. *Operational Amplifiers and Linear Integrated Circuits (forth edition).* Prentice-Hall, 1991.

Faulkenberry, Luces M. *Introduction to Operational Amplifiers with Linear Integrated Circuit Applications.* Wiley, 1982.

Gayakwad, Ramkant A. *Op-Amps and Linear Integrated Circuits (third edition).* Prentice-Hall, 1993.

Rutkowski, George B. *Integrated Circuit Operational Amplifiers (second edition).* Prentice-Hall, 1984.

Other reference material
Linear data books and application notes published by various IC manufacturers, e.g. National, Fairchild, Motorola, Analog Devices.

Student purchases
The following components and materials are required for the skill practice exercises and practical tests for this module.

- Type 741 op amp - 1 required; at least one spare recommended (741, 741C, 741A, 741E etc. are all acceptable)
- Type 301 op amp - 1 required; at least one spare recommended
- Protoboard - 1
- Banana sockets - about 6
- Resistors (1/4 watt) - must include several different values in every decade up to about 2M2
- Capacitors - must include some electrolytics values up to 100μF; small capacitors about 3pF, 10pF, 30pF and 100pF; a variable capacitor in the range 3pF - 30 pF; and several non-electrolytics in the range 100pF -100nF
- Cutting pliers and wire stripper
- Connecting wires (cables for connection to lab instructions will be available in the lab)
- Scientific calculator eg. Casio FX82 or similar
- Graph paper - several sheets of linear x linear and log x linear
- Digital multimeter will be supplied to you in the lab, but you may use your own.

Module organiser

Section	*Activity*	*Suggested time*
1	Basic op amp circuits Skill practice 1 Skill practice 2	6 hrs 30 mins
2	DC offsets Skill practice 3	4 hrs 30 mins
3	Slew rate Skill practice 4	4 hrs 30 mins
4	Noise Skill practice 5	9 hrs 30 mins
5	Frequency compensation Skill practice 6	7 hrs 30 mins

Notes

MODULE SECTIONS

Section 1: Basic Op amp circuits

SUGGESTED DURATION	PREAMBLE
6 hrs 30 minutes	To acquaint you with the design, operation and testing of simple linear op amp circuits as detailed in the objectives. In the study of this topic: • theoretical work assumes ideal op amps • only dual supply operation is considered.

Objectives

At the end of this section you should be able to:

- ☐ state the meaning of the term 'operational amplifier' (or op amp) and state the main properties of an ideal op amp
- ☐ given suitable specifications, design the following kinds of operational amplifier circuits using dual power supplies:
  - inverting amplifiers
  - non-inverting amplifiers
  - voltage followers
- ☐ for each of the circuits in the second objective, state or sketch the correct phase relationship between input and output and predict maximum output voltage swing and minimum load resistance based on the maximum output current capability of the op amp
- ☐ given typical transconductance and transresistance amplifiers, use the relations between the output and suitable inputs
- ☐ understand, and apply in practice, proper breadboarding techniques with attention to selection of components, layout, insertion and removal of components, good wiring, earthing, supply decoupling and safe power-on sequence
- ☐ construct the circuits listed in the second objective and measure the voltage gain, input resistance, maximum output voltage swing and maximum output current, with due attention to the device specifications of maximum power supply voltages, input voltages and power supply connections.

References
The following references deal with topics in this section.
1. Jacob(1993) pp 1-22, 52-65
2. Rutkowski (1984) pp 38-51
3. Gayakwad (1993) pp 109-115, 139, 272-274, 264-269
4. Coughlin & Driscoll (1991) pp 10-11, 13-19, 39-46, 53-60, 103-110, 117-122

The operational amplifier

The operational amplifier (op amp) is a high-gain DC amplifier with differential input and single-ended output as illustrated in Figure 1.

'Differential input' means that there are two inputs - the 'non-inverting' or (+) input and the 'inverting' or (-) input. The amplifier amplifies the difference voltage between the two inputs.

'Single-ended' output means that there is one output terminal, and the output voltage is taken from that terminal to ground. Level shifting circuits inside the op amps ensure that they can work with a variety of power supply voltages and still give the same output.

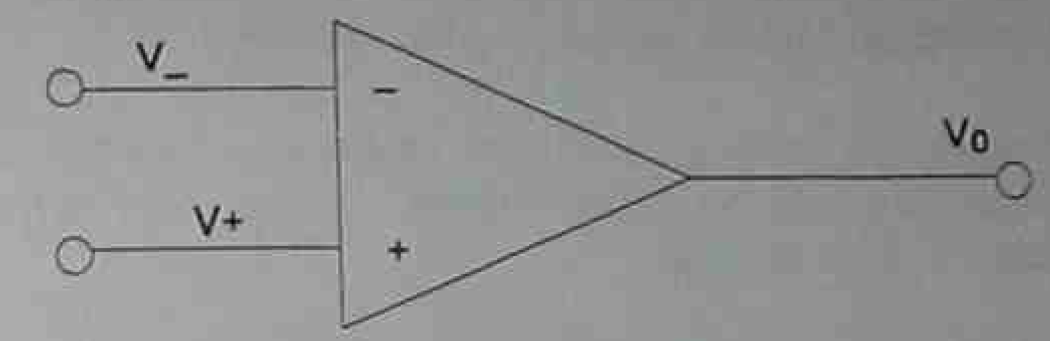


Fig. 1 Operational amplifier

The op amp is capable of working right down to DC, which is zero-frequency. This is possible because there are no blocking capacitors in the op amp.

Op amps come in many types, some having better specifications than the others. Some are general purpose, some optimised for good DC performance, some optimised for use as switches, some for high frequency etc. They are available in many packages such as 8-pin mini dual-inline-package (DIP), 10-pin flatpack, 8-pin round metal can or dual packages (two op amps in one package). General purpose types 741 and 301 are used for work in this module.

Absolute maximum ratings
Op amps are very rugged and easy to use, but you must check and stay within their absolute maximum ratings. If these are exceeded, the op amp may be destroyed. The most important ones are:

- maximum supply voltages. For the type 741, these are ±18V (for dual supply operation) or 36V (for single supply). Other types may have slightly different specifications.
- maximum input voltage at each input terminal. Typically these can range within 1 volt of the DC supply voltages. (Most op amps have built-in protection against too much input voltage.)
- maximum output current. The load resistance must be large enough to limit the current drawn from the output terminal to a safe value. (Many op amps have output current limiting protection, but this will still distort the output.)

With all the built-in protection, about the only ways to destroy an op amp are to interchange the + and - supply voltages or to connect the output terminal to a supply rail or the input signal source.

Ideal operational amplifier

For a quick understanding of op amp circuits, it is useful to suppose that the op amps are ideal. The properties of an ideal operational amplifier are that:

- the op amp has infinite gain ($A_{VOL} = \infty$)
- the op amp has infinite input resistance, between the input terminals and ground, as well as between the two input terminals themselves
- the output resistance of the op amp is zero.

Real op amps come close to the ideal. For practical op amps, the voltage gain may be ≈ 200 000 at DC and drop to a few hundred at upper audio frequencies (e.g.20 kHz). The input resistance may range from a few MΩ to a few GΩ depending on the type. The output resistance in typical circuits may be under 1Ω.

The ideal op amp is also supposed to be noiseless, its parameters independent of frequency, and not introduce unwanted phase shifts between the input and output. These issues will be dealt with in later sections.

In the study of this topic, theoretical work assumes that the operational amplifiers are ideal. Only dual supply operation is considered.

Linear amplifiers and negative feedback

Op amps have very large voltage gain. If we put the input signal directly between the + and - inputs, the output will tend to become so large that it will be distorted. For example, if the input voltage is 1 mV and the voltage gain is 50 000, the output will try to be 1 mV * 50 k = 50V, which is way beyond the voltage handling capacity of the op amp. So the output will not reach 50V, and will be distorted.

A *linear amplifier* has an output which has the same shape as the input without distortion (though phase shift is allowed). So an op amp cannot be directly connected as a linear amplifier.

Negative Feedback (NFB) is a method of reducing the effective voltage gain of a circuit in a controlled manner and thereby, linearize the amplifier circuit. With NFB, the output voltage (or part of it using a voltage divider) is connected back to the *inverting input.* If the output voltage gets too big, then the - input voltage also increases due to NFB. So the effective input voltage for the op amp is decreased, and the output also falls.

The voltage gain of the op amp, after using NFB, is called the *closed loop gain* (A_{VCL}) of the amplifier. This is usually much less than the open loop gain of the op amp itself.
Note carefully that when we talk about the 'gain of an amplifier', we usually mean the closed loop gain (unless stated otherwise).

Analysis of ideal linear (negative feedback) op amp circuits

The analysis of all negative feedback op amp circuits follows a few simple rules. These rules apply if the op amp is working as a linear amplifier, i.e. when there is NFB and there is no distortion. The rules are as follows.

- Because of the large open loop gain, a negligible voltage difference between the + and - inputs is sufficient to produce a sizeable output.
  ∴ $V_+ = V_-$ for normal operation.
- Because of the large input resistance, no current flows into the input terminals of an op amp. (Note that current may flow into the other circuit components around the op amp.)
- Because of the low output resistance, regardless of the load resistance connected to the output of the op amp, the output voltage does not vary. (Note that the load current must be less than the maximum output current rating of the op amp.)

Non-inverting amplifier

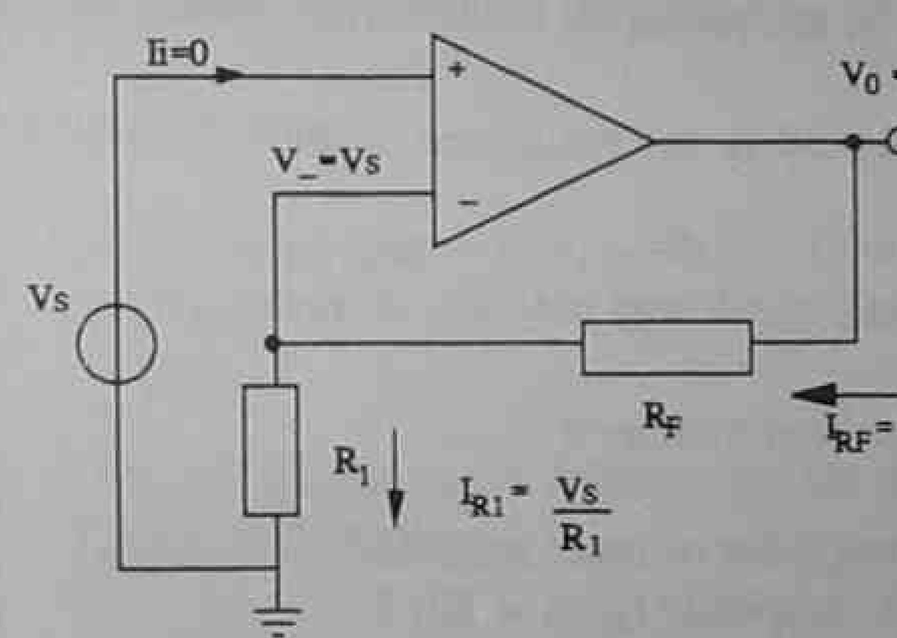


Fig. 2

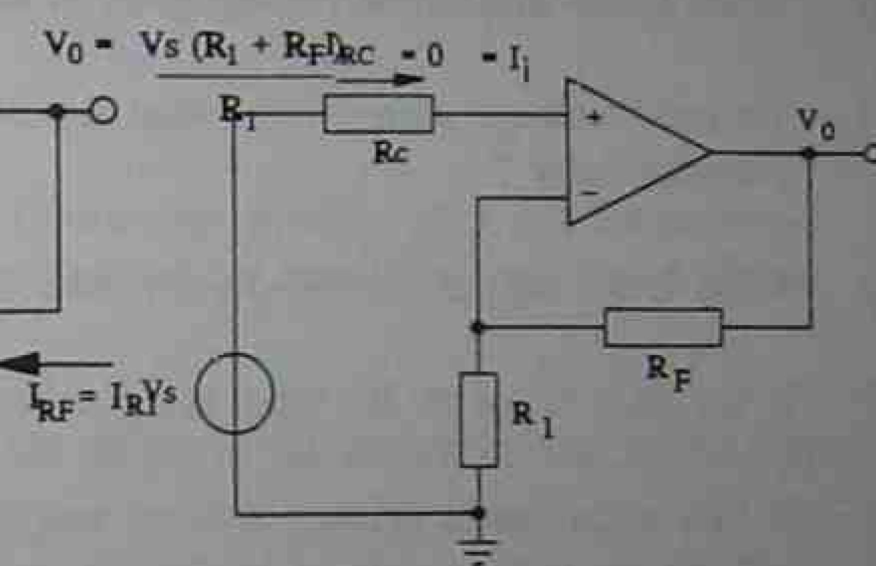


Fig. 3

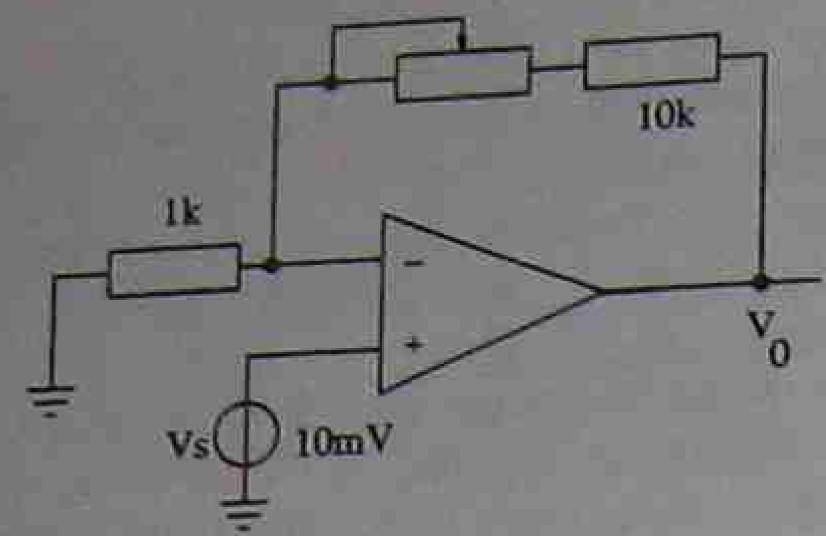
Non-inverting amplifier configurations

The circuits of Figures 2 and 3 are both non-inverting amplifiers. The input source is wired to the +input, which makes the amplifier non-inverting. The DC supply connections are not shown, even though they must be present.

If the op amp is ideal :

- the (closed-loop) voltage gain $A_V = v_o/v_s = (1 + R_F/R_1)$
- the input and output voltages are in phase
- the input resistance seen by the source $\frac{V_s}{I_i} = \infty$ (Note : it is not R_c)
- the output resistance = 0
- R_c makes no difference to the properties mentioned above. (The reason for including it is given in the next section.)

Example 1 : Non-inverting amplifier



Questions

(a) What is the minimum voltage gain of the circuit ?

(b) To get a voltage gain of 61, what should be the resistance of the potentiometer ?

(c) What is the signal current drawn from the source ?

(d) If the gain is set to 30, what is the output voltage ?

(e) If this amplifier is driving another amplifier of input resistance 5kΩ, what will be the output voltage of the first amplifier (gain = 30) ?

(f) If the signal source has internal resistance of 1 kΩ, what will be the output voltage (gain = 30) ?

Solutions

(a) $A_{v\,min} = (1 + 10k/1k) = 11$ (when the pot is set to 0Ω).

(b) For $A_v = 61$, $R_F = (61-1) * 1k = 60k\Omega$
Since 10 kΩ is already present and fixed, required pot resistance = 50 kΩ

(c) 0

(d) $v_o = 30 * 10\ mV = 300\ mV$

(e) unchanged

(f) unchanged

Inverting amplifier

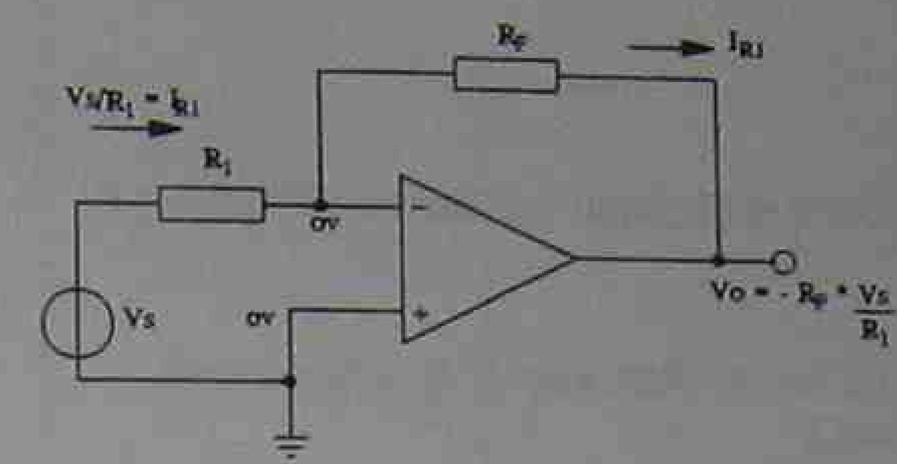


Fig. 4

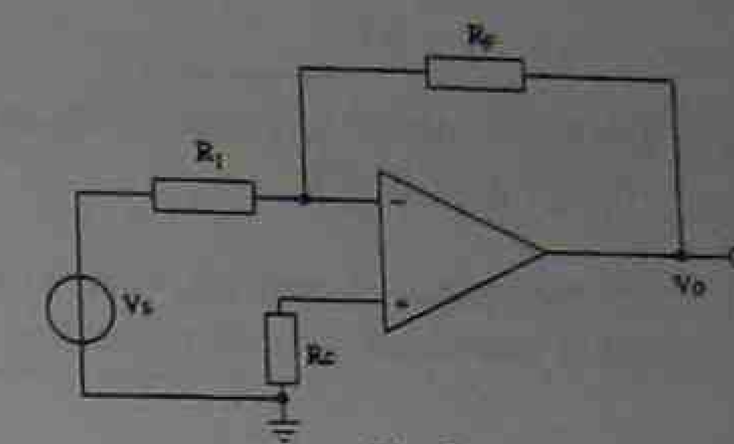


Fig. 5

Inverting amplifier configurations

The circuits of Figures 4 and 5 are both inverting amplifiers. The input source is wired to the -input, which makes the amplifier inverting. The DC supply connections are not shown, even though they must be present.

For ideal op amps:

- the voltage gain = $v_o/v_s = -R_F/R_1$ (Note : the voltage gain, for the same components, is one less than for non-inverting amplifiers, and has a minus sign.)
- the output and input voltages are out of phase (as shown by the minus sign in the gain formula)
- input resistance of the amplifier = R_1 (Note : it is not ∞.)
- output resistance = 0
- R_c has no effect on the properties of the amplifier.

Example 2 : Inverting amplifier

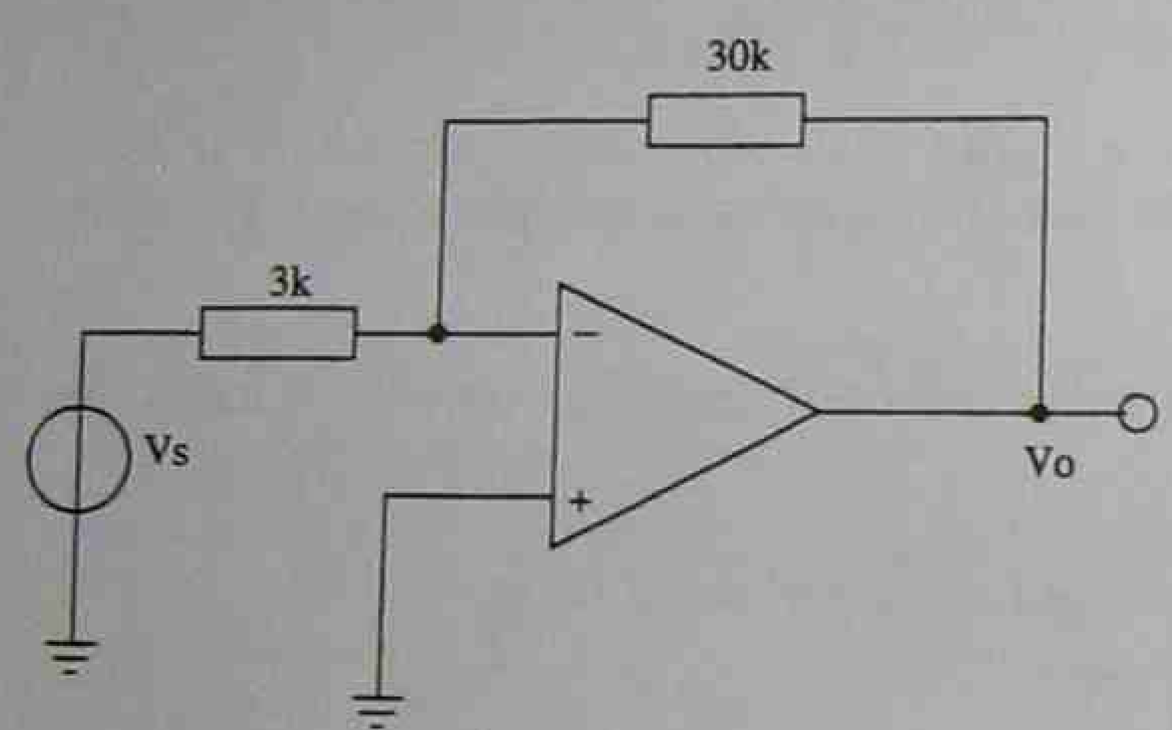
Questions

(a) Draw the circuit of an inverting amplifier with input resistance of 3 kΩ and voltage gain of 10.

(b) For the circuit in part (a), what will be the voltage gain if it is used with a signal source of internal resistance 600Ω.

Solutions

(a) R_1 = input resistance = 3 kΩ
$R_F = 10 * R_1 = 30$ kΩ
The circuit for part (a) is shown below without the power supply connections.



(b) $R_1 = 3$ kΩ as before, but
A_{VCL} = 30 kΩ / (3kΩ + 600Ω) = 8.333

Voltage follower

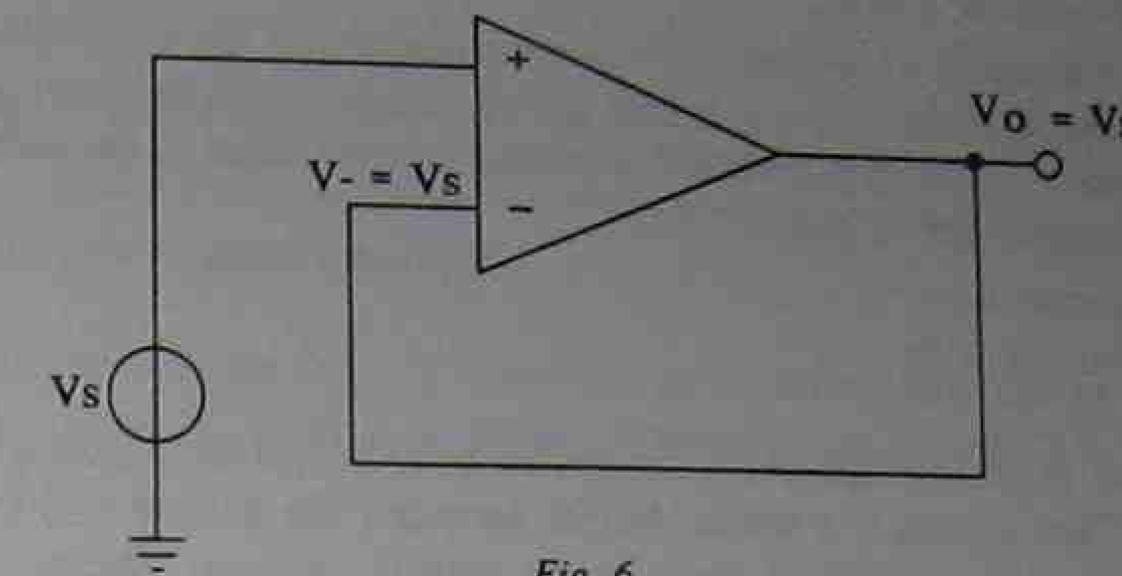


Fig. 6

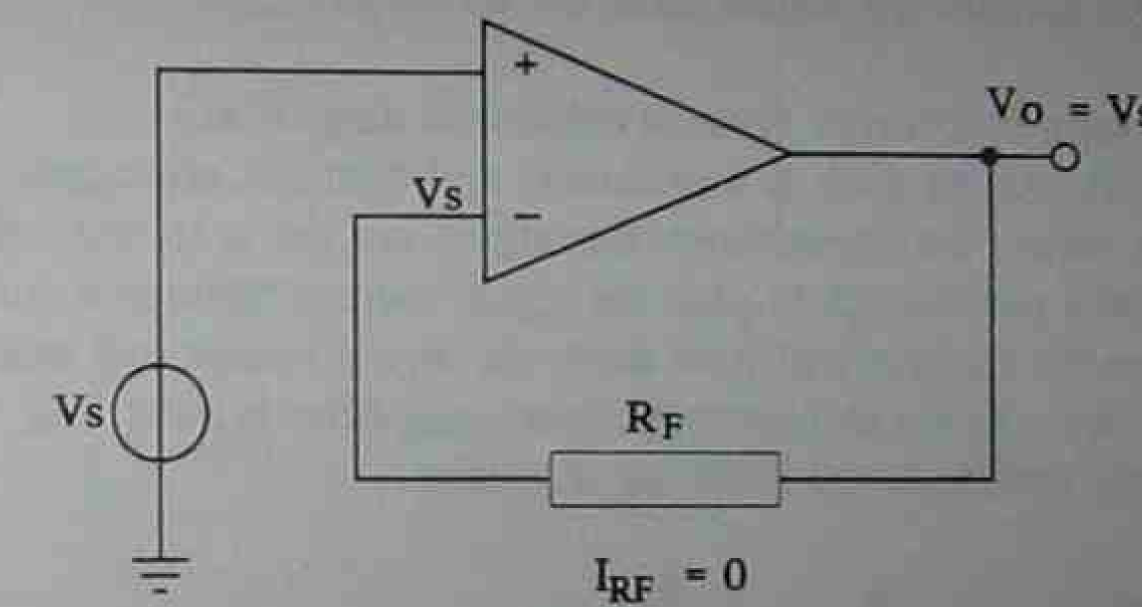


Fig. 7

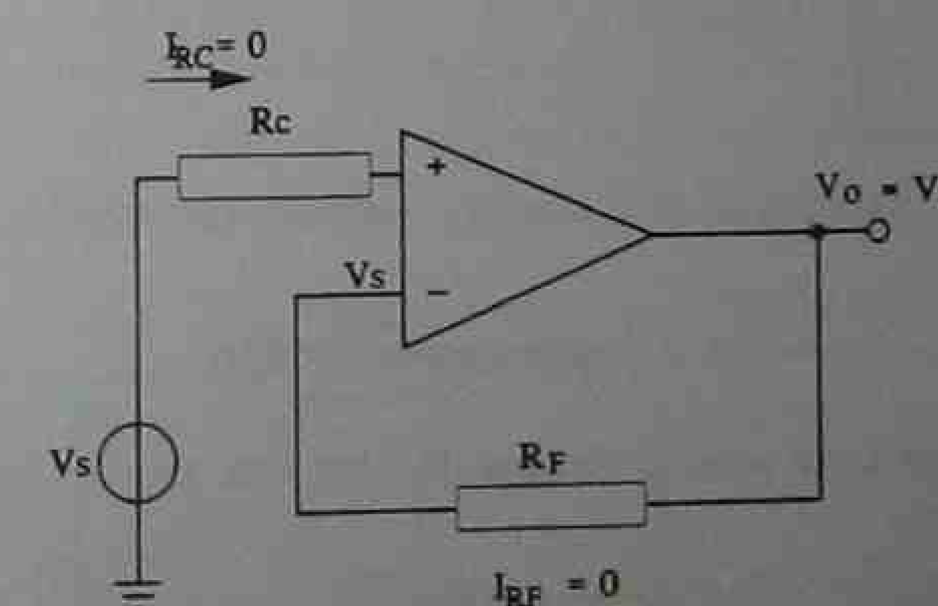


Fig. 8

Voltage follower configurations

The circuits in Figures 6, 7 and 8 are all voltage followers, though circuit 6 is the simplest and most common. In all the voltage follower circuits, the resistance R_1 from -input to ground is absent.

With ideal op amps :

- voltage gain = 1 (i.e. input signal = output signal; there is no voltage gain.)
- output and input are in phase
- input resistance = ∞
- output resistance = 0.

Even though there is no voltage gain, the useful property of this circuit is its high input resistance. This makes it a useful buffer between the source and the amplifier. The gain of the inverting amplifier can change with the source internal resistance. (There are other circuits such as the differential amplifier where the same problem arises.) The buffer isolates the source from the amplifier.

Current to voltage converter (transresistance amplifier)

Many useful signal sources such as transducers and light detectors give a current as their output. The easiest way to measure this signal current is to convert it to a voltage. It is usually not enough to pass the signal current through a resistor to make a voltage, because the resistor will load down the signal source and reduce the signal current. In such cases, an op amp transresistance amplifier is used (Fig. 9). This circuit is very similar to the inverting amplifier.

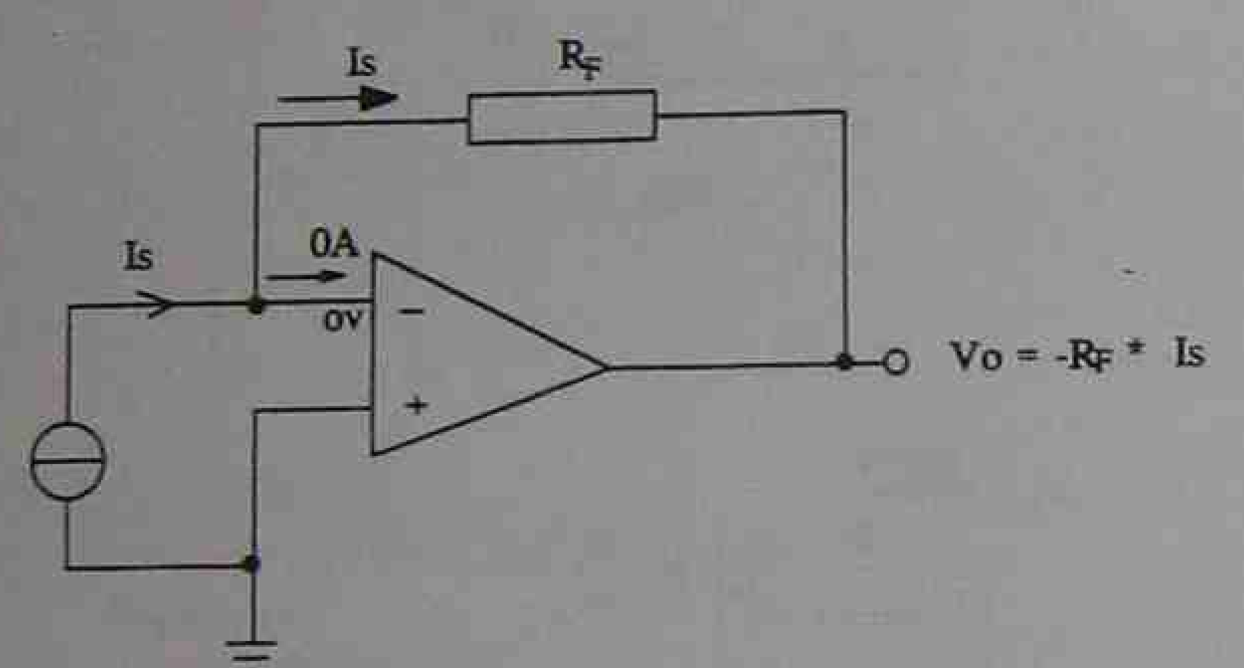


Fig. 9 Transresistance amplifier

The input current must flow around through R_F, because the op amp input does not draw any current. Also,

$V_- = V_+ = 0V$

$\therefore V_o = 0 - R_F * I_s = - R_F * I_s$

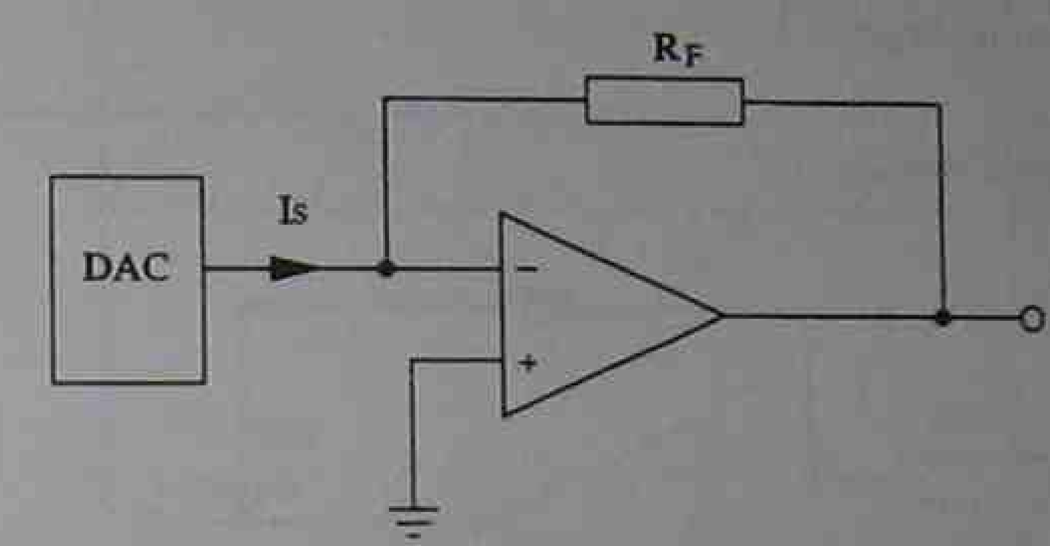
This means that $V_o \propto I_s$. The ratio V_o/I_s is called the transresistance R_T of the amplifier.

In this circuit, the -input is called 'virtual ground' because it is held at 0V, even though it is not physically connected to ground.

The signal source drives all its current into R_F. The output is measured at the op amp output, which has 0 output resistance, so the meter is not loaded, even if R_F is large.

Example 3 : Transresistance amplifier

In the following circuit, the D to A converter gives an output current in the range 0 to 1.992 mA. Select R_F to give an output voltage range of 0 to 5V.



Solution

$R_F = 5V / 1.992mA = 2.51 k\Omega$

Voltage to current converter (transconductance amplifier)

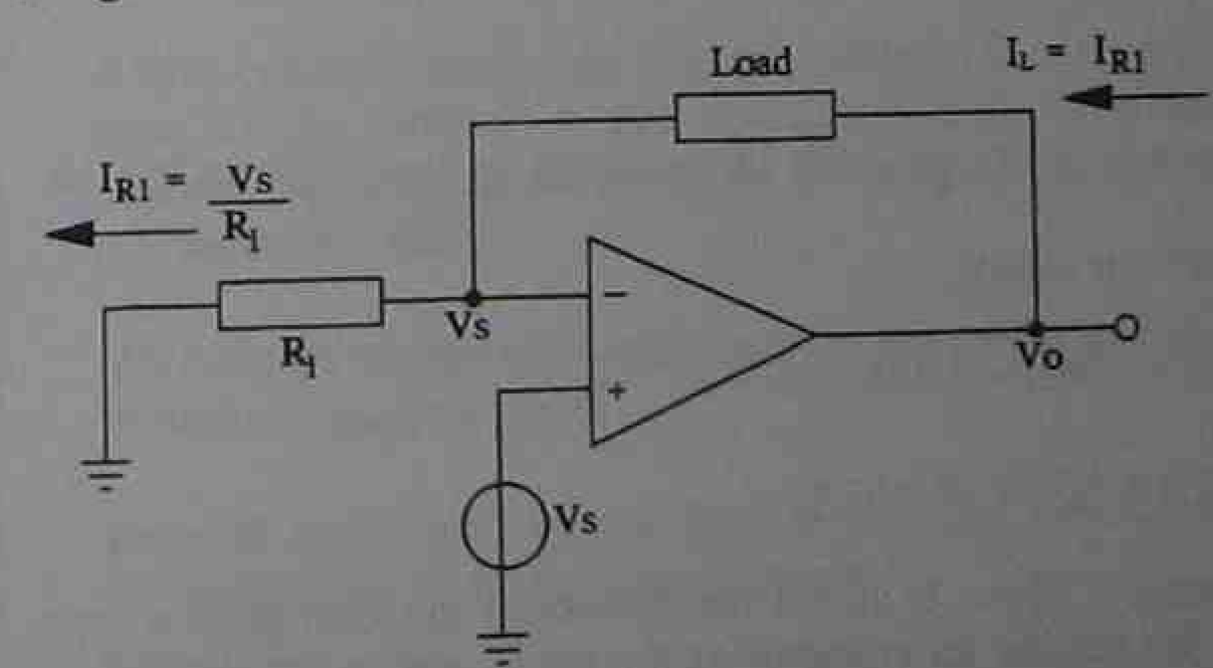


Fig. 10 Transconductance amplifier

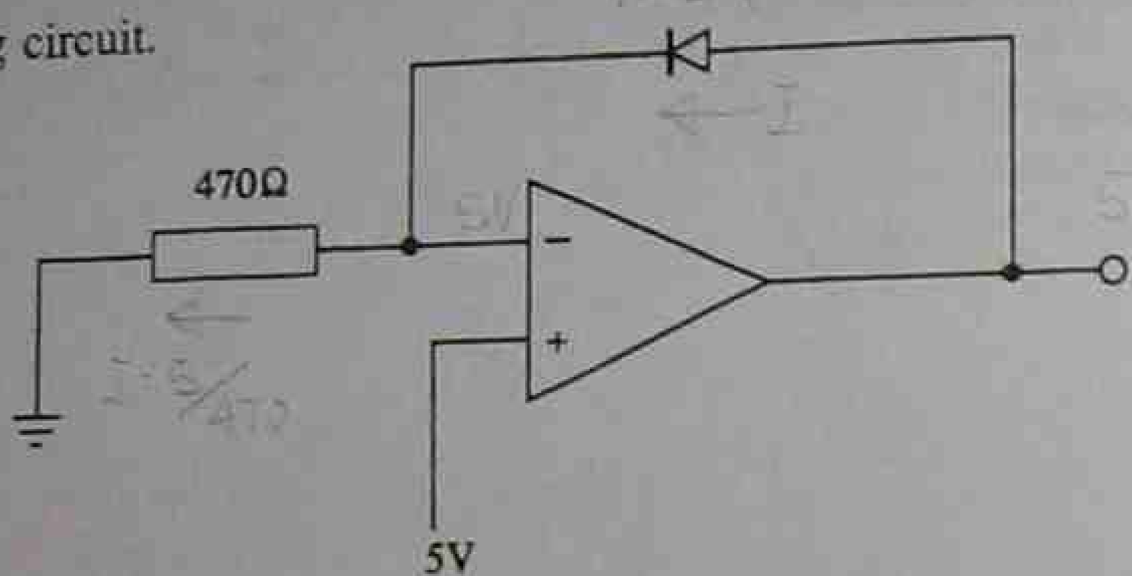
The circuit in Figure 10 is very similar to the non-inverting amplifier. Here, our aim is to pass a known current through the load, not amplifying the input voltage. The load can be any circuit components, possibly even diodes, rectifiers or meters. The purpose of the op amp is to isolate the source from the load. The only conditions are that the load must allow voltage to feed back from the output to the -input, and the output is not saturated (clipped).

Using the circuit rules for negative feedback op amp circuits:

- $V_- = V_+ = V_s$
  $\therefore I_{R1} = V_s / R_1 = I_{Load}$
- $\therefore I_{Load} = V_s / R_1$, independent of the load characteristics, *the load current being set by the value of R_1*
- I_{Load}/V_s is called the transconductance g_m of the circuit
- $V_o = V_s$ + (voltage drop in the load.)

Example 4 : Transconductance amplifier

Study the following circuit.



Questions

(a) Is the diode forward biased or reverse biased ?

(b) Calculate the diode current.

(c) Calculate the output voltage, if the diode voltage drop is 650 mV.

Solutions

(a) The direction of current in R_1, and also in the diode, is from right to left i.e. anode to cathode.
$\therefore$ The diode is forward biased.

(b) Diode current = 5V / 470Ω = 10.63 mA

(c) Output voltage = 5V + 0.65V = 5.65V

Note: The circuit in example 4 above is useful for measuring the voltage of a high impedance source (by putting an ammeter as the load), and to get the V-I characteristics of non-linear devices (by varying R_1 to change the current, without loading the source).

Maximum output swing in op amp circuits

We have mentioned that the output voltage capability of op amps is limited by the power supply. In practice, the maximum output swing (or peak-to-peak voltage) from an op amp circuit is a few volts less than the DC supply voltage differential. If we try to make the output bigger than the maximum swing, it just clips (becomes flat-topped).

The maximum output swing decreases with the load current. For example, a 741 with ±15V supply has a maximum swing of ±14V (or 28V p-p) for a 10 kΩ load, but only ±13V swing with a 2 kΩ load.

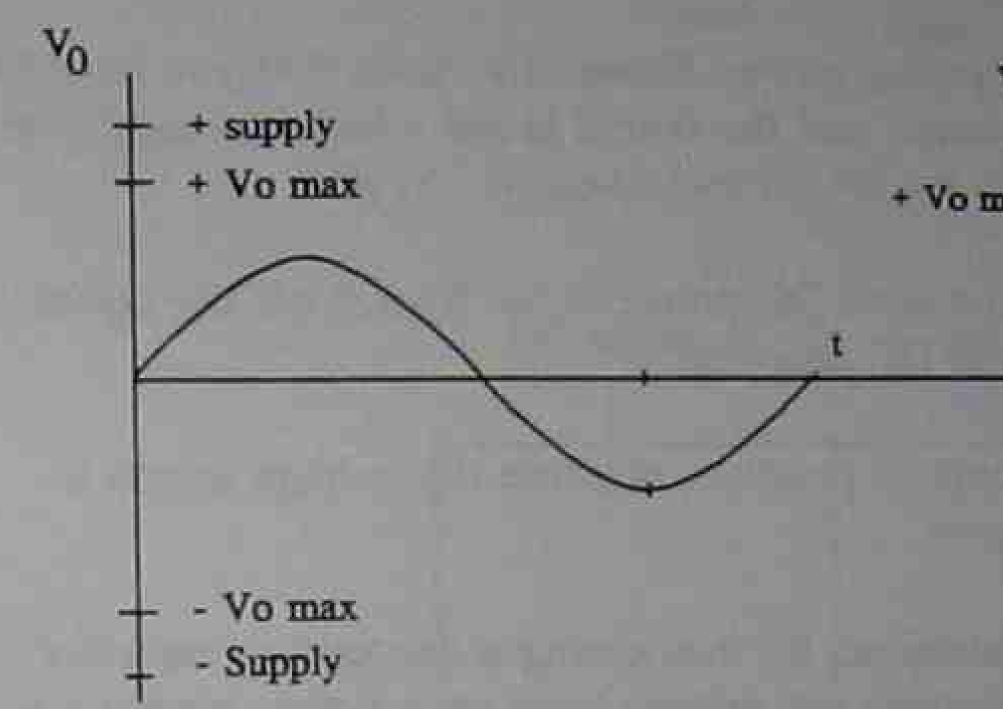


Fig. 11 Normal output

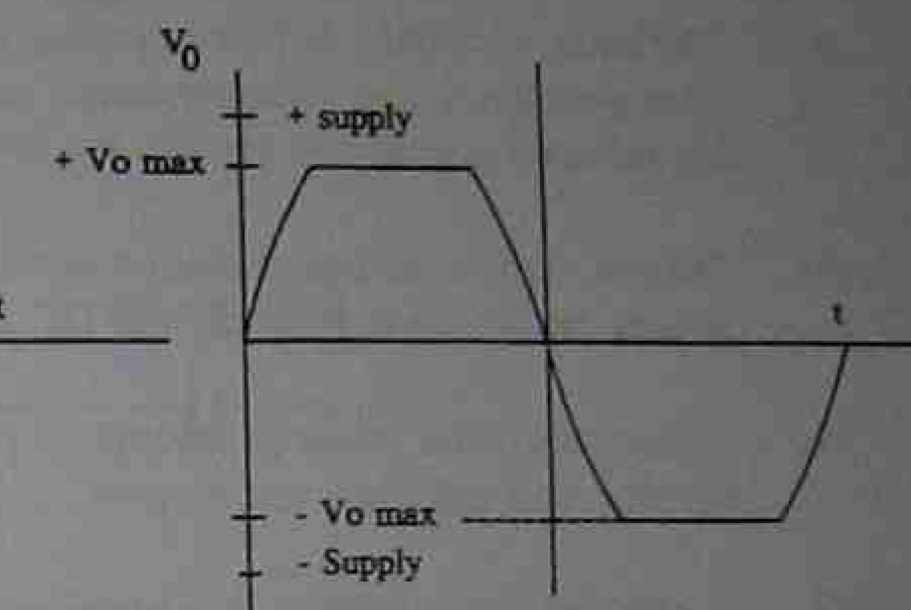


Fig.12 Clipped output

Breadboarding practice

The following tips will help you to do your practical work in this subject with the minimum of hassle and frustration, and to reduce the chances of blowing your ICs.

1. Do any wiring, or changes in wiring, with the DC power supplies switched off.

2. Set up the power supply voltages before connecting them to the circuit. Especially check that the supply voltages are within the op amp ratings.

3. Loose connections are the biggest problems to isolate in test circuits. Make sure that the sockets are mounted tightly to the breadboard and wires are inserted into sockets securely and not wrapped around leads. Alligator clips are not recommended.

4. Wire the power supply leads first. Do not forget to connect the power supply ground to circuit ground.

5. Keep all leads and wires as short as possible. Use shielded cables if possible for connection to external instruments.

6. Do not try to force thick component leads into the holes in the protoboard. This will damage the board, and the next time you use the board, you will be mystified why the circuit does not work.

7. Take care in inserting and removing ICs into boards. It is easy to bend an IC pin and very hard to see it. (It is possible to get a special tool to remove ICs from boards.)

8. Try to connect all ground connections to a single point rather than use a long ground rail, which could pick up noise.

9. Recheck all wiring before applying power. Especially check that you have not swapped the + and - supply leads, and the output is not wired to a supply or signal source lead.

10. Do not switch on signal source until DC power is on. Switch off the signal source before switching off the DC supplies.

11. Ammeters often cause problems. If possible, measure the voltage across a resistor and calculate the current.

12. If external noise pickup is a problem, try connecting a decoupling capacitor (10 nF to 100 nF - the exact value does not matter) from each power supply pin to ground. If you have to shield the whole circuit, a simple idea is to place it in a closed biscuit tin with holes cut out to bring out the leads.

13. Do not measure a voltage directly at an IC pin. You may accidentally short two adjacent pins together.

Power supply connection for op amps

For proper operation, the op amp needs DC power supply connections. Usually, we use two DC voltages — a positive supply and a negative supply of equal value. The maximum supply voltages are given in the op amp data sheets, e.g. ± 22V for general purpose op amps such as type 741. In circuit diagrams, usually the power supply connections are not drawn, though they are supposed to be there. It is important that you connect the DC supplies to your op amp properly, especially taking care to connect the power supply ground to circuit ground.

The pin connections for the type 741 op amp and the method of connecting two separate DC supplies to make a single +, - and ground supply are given below. Many other types of op amp such as 301 and LF351 also have the same pin connections as the 741 (i.e. they are pin compatible) but have improved specs.

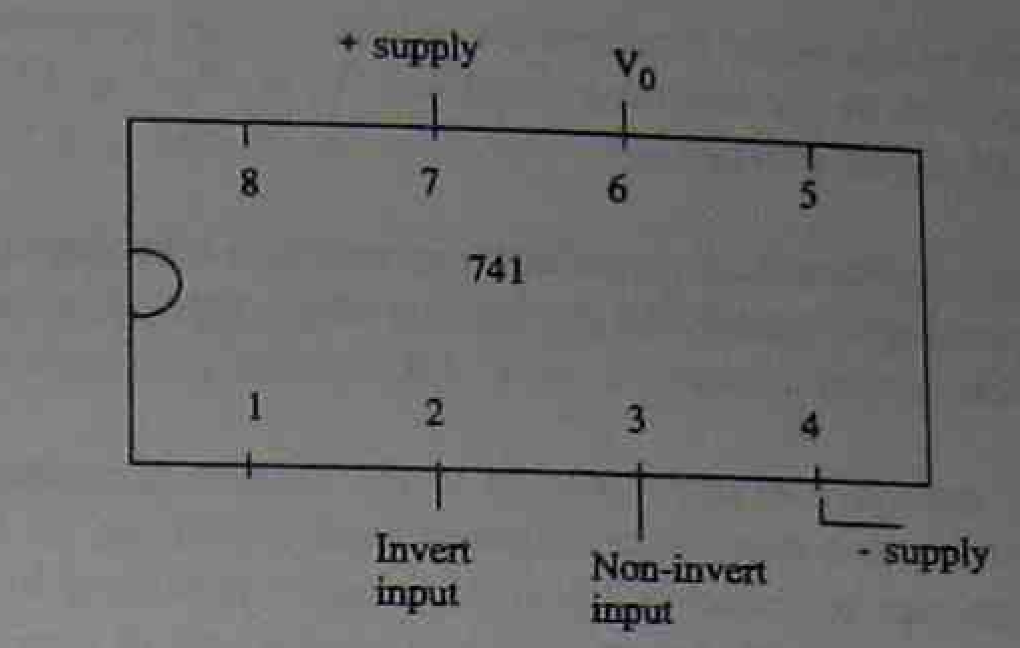


Fig.13 Pin connections for 741 and 351 type op amps

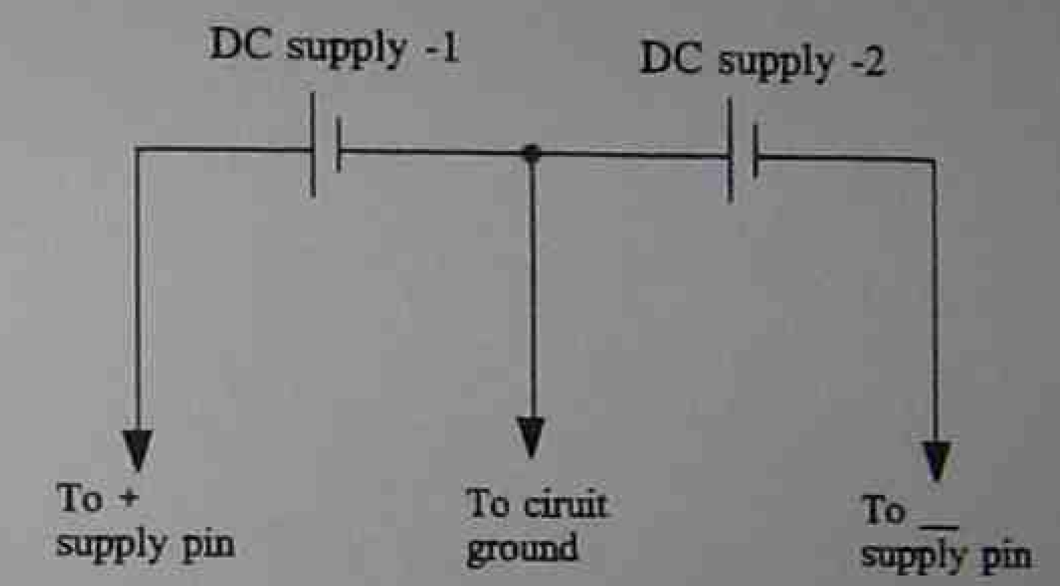


Fig.14 Dual power supply connections for op amps.

Summary

The ideal op amp has infinite gain, infinite input resistance and zero output resistance. Real op amps are close to ideal.

In non-inverting amplifiers, the input signal has a direct connection to the +input. For these amplifiers, $A_v = 1 + R_F/R_1$; $R_i = \infty$, $R_o = 0$; output and input in phase.

In inverting amplifiers, the input signal has a direct connection to the -input. For these amplifiers, $A_v = - R_F/R_1$; $R_i = R_1$, $R_o = 0$; output and input out of phase. For the same components, the inverting amplifier has a smaller voltage gain than the non-inverting amplifier.

In voltage voltage followers, there is no R_1 and the signal is connected to the +input. The $A_v = 1$ i.e. input voltage = output voltage; $R_i = \infty$, $R_o = 0$; output and input in phase.
This circuit does not amplify the signal, but is used to isolate the source from the load to prevent loading errors.

The current to voltage converter produces an output voltage proportional to the input current. It is a variation of the inverting amplifier. $V_o = -R_F * I_s$. This circuit is good for measuring small signal currents.

The voltage to current converter is a variation of the non-inverting amplifier. It is used to drive a current proportional to the input voltage through a load, which can be a linear or non-linear circuit element. $I_L = V_s / R_1$

For safe operation, you have to pay attention to the maximum supply voltage, maximum input voltage and maximum output current which can be handled by the op amp. Their values can be found in linear data books published by manufacturers. The maximum output voltage of the op amp is typically one or two volts below the power supply voltage.

Review questions

These questions will help you revise what you have learnt in Section 1.

1. An inverting amplifier works with ±12V supplies. It has a gain of -10 and an input resistance of 10kΩ. The input signal is 500 mV sine wave peak to peak, and has no internal resistance.

   (a) Sketch the circuit.

   (b) Sketch output voltage. Show important amplitude values and phase relations.

(c) Sketch the output voltage if the input voltage is 3V p-p. Show important amplitude values and phase relations.

2. (a) in Q.1 how will the output voltage change if the signal source has a significant internal resistance?

____

____

(b) Sketch an addition to the circuit in Q.1 to overcome the problem mentioned in part 2(a) above.

3. In the following circuit,

(a) What will be the average meter current if the input voltage is 5V rms sine wave? (Note: For sinewaves, full wave rectified average = 0.9 * rms)

____

____

____

____

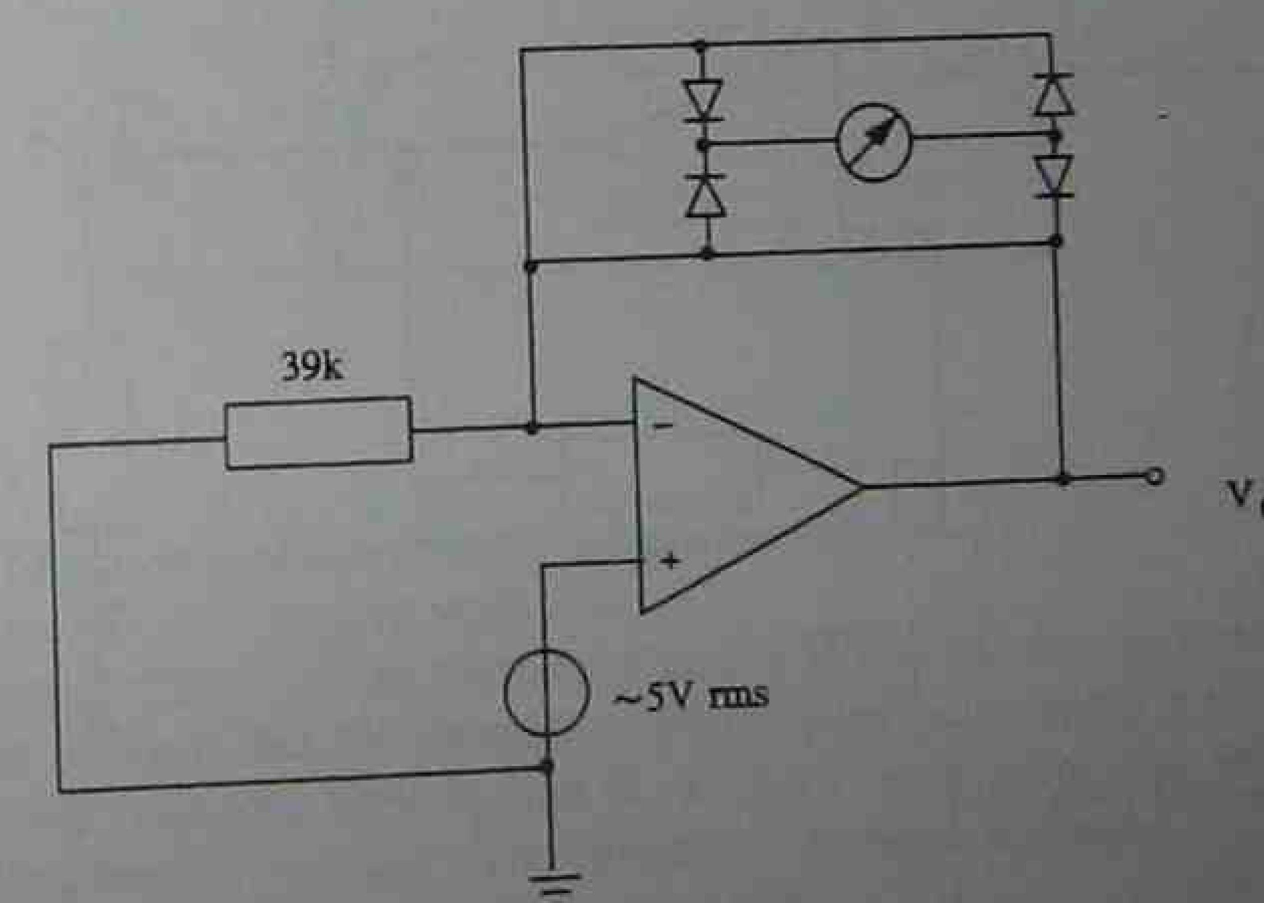
(b) If the diodes have 0.6V forward voltage drop each and the meter has zero resistance, what is the maximum output voltage (assume no clipping)?

____

____

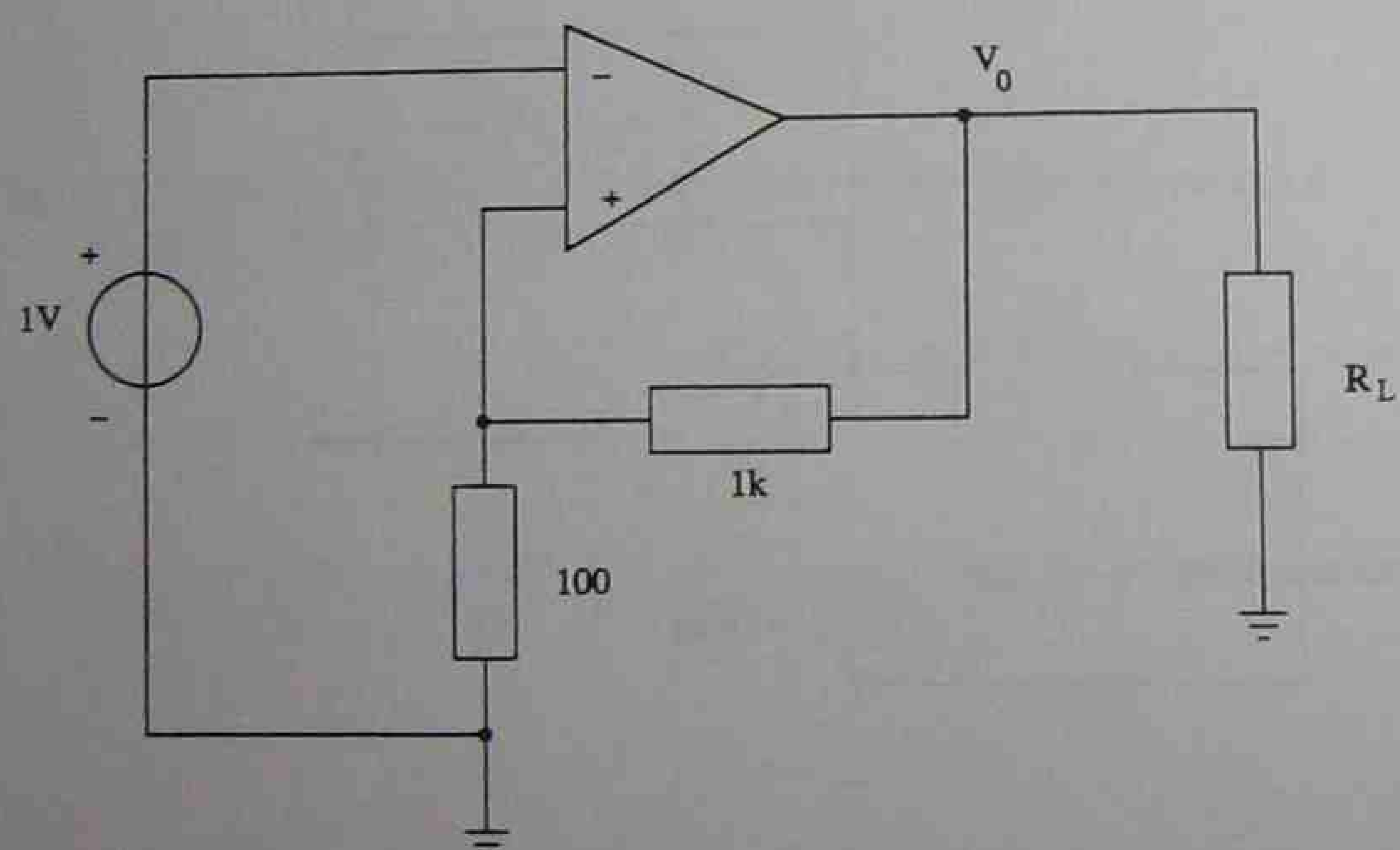
____

____



4. A transducer output has a temperature sensitivity of 150 nA/°C. Draw a circuit to change this to a sensitivity of 180mV/°C.

5. In the following circuit, the op amp has a maximum output current of 25mA. What is the minimum value of load resistance?



Skill practice 1

Suggested duration

1 hour 15 minutes

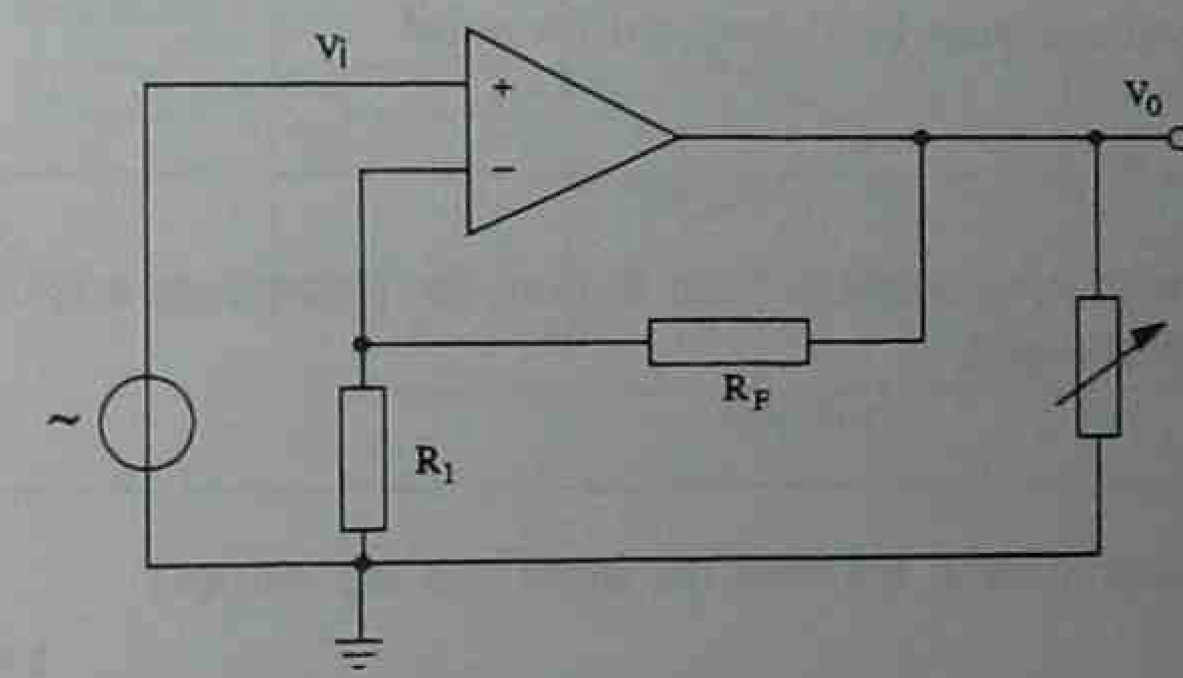
Tasks

- To measure the voltage gain of an amplifier
- To measure maximum output voltage swing of an amplifier
- To measure maximum output current from an amplifier

Equipment

- One type 741 op amp or similar (e.g. LF351)
- Sine wave generator (general purpose audio frequency function generator is adequate)
- 15 MHz dual trace oscilloscope
- ±15V DC supplies
- Decade box (range at least as wide as 100Ω to 100kΩ)
- Selection of resistors

Circuit diagram



Procedure

Step 1 Setup and observation of output waveforms

- Use ±15V DC supply voltages.
  Connect the circuit as shown, referring to the pin connections diagram of your op amp and the method of power supply connection in the previous section. You can vary R_F and R_1 within a wide range, for example R_F = 10 kΩ and R_1 = 1 kΩ. The procedure remains the same.
  Make the decade box resistance 10 kΩ.
- Set the signal generator to 1 kHz.
  Observe the input and the output on two channels of a CRO and adjust input voltage till the output is reasonably big (i.e. well above the noise level and not clipped).
- Observe the phase relation between the input and the output.

Step 2 Measurement of voltage gain

- Measure V_o and V_i using the same instrument (i.e. both can be measured p-p using a CRO, or both rms using a voltmeter).
  V_o =
  V_i =
- Calculate the experimental voltage gain = V_o/V_i =

Step 3 Maximum undistorted output voltage swing

- Connect a CRO channel to the output and increase the input signal until the output just begins to clip.
- The maximum undistorted output peak-to-peak voltage swing =

Step 4 Maximum output current

- Keep the output signal just below clipping. Decrease R_L until the output just begins to get distorted. Measure the **peak** output voltage and divide it by R_L, which gives the maximum output current.
  v_o (peak) = R_L =
- Maximum output current = $V_o/R_L - V_o/(R_F+R_1)$ =

Discussion questions

1. Calculate the voltage gain by theoretical formula:

   $A_v = 1 + R_F/R_1$ = ______________________

   Compare this with your result in Step 2. Find the percentage error in your experimental result.

   ____

2. What is the phase relation between the input and the output?

   ____

3. How many volts is your result for the maximum output voltage swing below the power supply differential?

   ____

4. Refer to the databook for your op amp and find out the specification for the maximum output current of your op amp.

   ____

   What is the percentage difference from your result in Step 4?

   ____

Skill practice 2

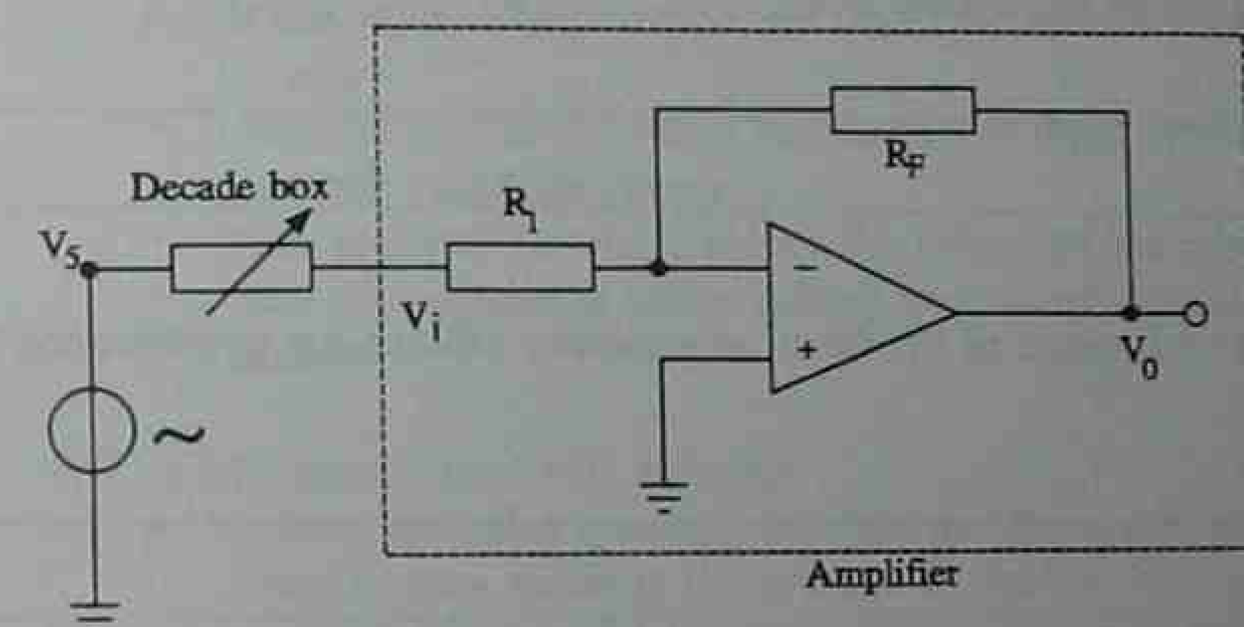
Suggested duration
1 hour 15 minutes

Task
To measure the input resistance of an amplifier.

Equipment

- One type 741 op amp or similar (e.g. LF351)
- Sine wave generator (general purpose audio frequency function generator is adequate)
- 15 MHz dual trace oscilloscope
- ±15V DC supplies
- Decade box (range at least as wide as 100Ω to 100kΩ)
- Selection of resistors

Circuit diagram



Procedure

Step 1 Setup

- Use ±15V supplies.
- Connect the circuit with suitable values of R_F and R_1 (e.g. 10 kΩ and 1 kΩ respectively).
- Make the decade box resistance = 0.
- Set generator to 1 kHz.
- Observe the output on a CRO and adjust input signal amplitude to get a good output (as in skill practice 1).

Step 2 Measurements

- Observe V_i and V_s on two channels of a CRO.
- Increase the decade box resistance until V_i is as close as possible to 50% of V_s. (This may not be always possible.)
- At this point, measure V_i and V_s and note the decade box resistance, R_D.

Step 3 Calculation of input resistance

- Calculate the input resistance, $R_i = R_D * V_i / (V_s - V_i)$

Discussion questions

1. For your amplifier circuit, what is theoretically the input resistance?

____

2. What can you say about the input resistance if V_i remains very nearly V_s, no matter how you adjust the decade box?

____

____

3. What can you say about the input resistance if V_i remains close to zero, no matter how you adjust the decade box?

____

____

4. Does the internal resistance of the function generator affect your experimental result? Why?

____

____

Section 2: DC Non-idealities

SUGGESTED DURATION	PREAMBLE
4 hrs 30 mins	To define the DC non-idealities of amplifiers: input bias current, input offset current, input offset voltage and their drift with temperature; predict their effect on the output of common amplifier circuits; and explain methods to nullify the effects.

Objectives

At the end of this section you should be able to:

☐ in relation to input offset voltage:
- define input offset voltage and read typical values for common types of op amps
- calculate output DC offset caused by the input offset voltage for common amplifier circuits
- state practical means to reduce the effects of the input offset voltage
- state the purpose of offset nulling and recognise common offset nulling circuits

☐ in relation to input bias currents:
- define input bias currents and state that the input bias current values given in data sheets is the average of the two bias currents
- calculate the output DC offset voltage caused by the input bias currents without compensation
- state practical means to reduce the effects of the input bias currents
- state the purpose of bias compensation; calculate and place the correct bias compensation resistor for common operational amplifier circuits

☐ define input offset current; calculate the output DC offset voltage caused by input bias currents for compensated circuits

☐ in relation to the effect of DC offsets on output:
- calculate the total DC output offset due to input offset voltage and bias currents for both compensated and uncompensated circuits
- state and sketch the general effects of input offset voltage and bias currents on AC signal outputs

- ☐ in relation to drift in DC offsets:
  - calculate the effects of drift in input offset voltages and current due to temperature change and due to power supply variations on the output DC offset voltage
  - state practical means to reduce the effects of drift
- ☐ measure the input offset voltage, input bias currents, and input offset current for an op amp
- ☐ demonstrate nulling the output DC voltage of any common type of op amp (eg. 741).

References

The following references deal with topics in this section.

1. Jacob (1993), pp 171-186
2. Rutkowski (1994), pp 58-67
3. Gayakwad (1993), pp 157-194
4. Coughlin & Driscoll (1993), pp 231-248.

Introduction

Until now we have studied ideal operational amplifiers. Real, practical operational amplifiers come close to the ideal, but not quite. For quick and approximate work, we can indeed consider the op amps to be ideal and get useful results. But if we want to get the best out of our op amps, or we work with very small or very fast signals, we have to understand the kinds of errors introduced by real amplifers, and hopefully correct for them. This is the theme of this and the next three sections.

In this section, we study the DC errors in operational amplifiers. These errors are due to input offset voltage, input bias currents and input offset current. In the next three sections, we look at problems in working with AC signals.

The DC errors studied here are usually quite small, and they do not affect the AC signals greatly. However, in many measurement and testing applications e.g. light levels, temperature, pressure, the signals are very small and very slowly varying, so the DC errors in the amplifiers may be significant.

Input offset voltage (V_{io})

If we connect an op amp as a voltage follower and ground the input, we would expect zero output voltage. But practically there would be a small DC output voltage. This is called the input offset voltage.

The op amp has a differential amplifier at the input stage, consisting of one transistor for the inverting input and another for the non-inverting input. These two transistors have slightly different DC voltage and current characteristics, causing a small DC offset voltage at the output even if there is no input. That is, the input offset voltage is caused by small DC imbalances at the input stage of the amplifier. Typical values may range from a few mV to a few tens of μV.

Notes

- The input offset voltage does not directly affect AC signals.
- The input offset voltage does not actually exist at the input terminals and cannot be measured there. You can only see its effect at the output.
- Values of V_{io} vary from type to type and from device to device for the same type. Typical values of V_{io} for any type can be read from data sheets.
- The input offset voltage can be positive or negative depending on the internal details of the op amp. The data sheets always give the magnitude of V_{io} as a positive number.

Effect of input offset voltage on output DC voltage

The output DC voltage (due to the input offset voltage)

$$= V_{io} * (1 + R_F/R_1) \qquad \text{Equation 1}$$

- V_{io} is the input offset voltage of the device
- R_F is the negative feedback resistor
- R_1 is the total (effective) resistance from the inverting input to ground. If more

than one resistor is in the path from the inverting input to ground, you must take the series or parallel effective resistance appropriately.

You can recognise the second factor in equation 1 above as the gain of the non-inverting amplifier.

You should note that the same formula applies whether the amplifier is wired as non-inverting or inverting.

In the circuit below, the op amp data specify maximum V_{io} to be 3 mV. The signal source has internal resistance of 600 Ω. The maximum output DC voltage caused by the input offset voltage is 3 mV * (1 + 100kΩ / (1kΩ + 600Ω)) = 187.5 mV DC

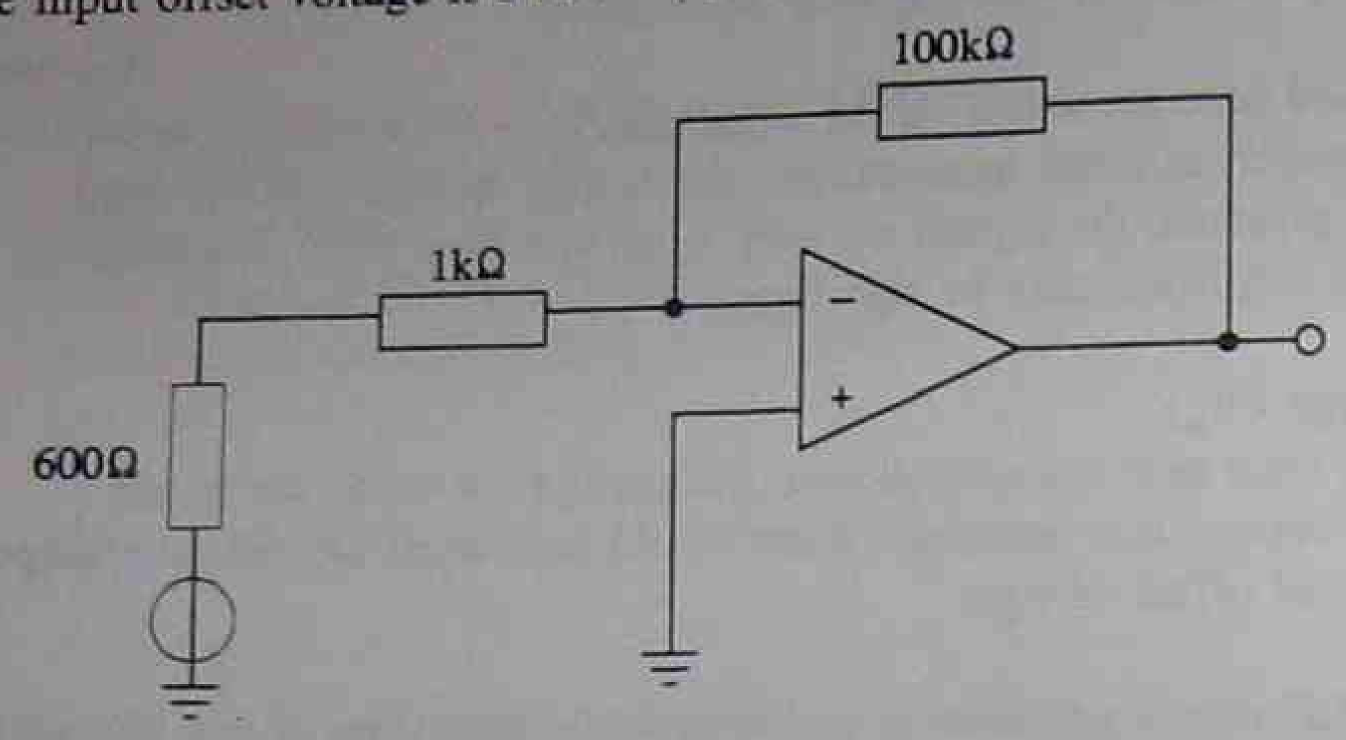


Fig. 1 Calculation of the effect of input offset voltage

Reducing the effects of input offset voltage

The effect of input offset voltage could be a nuisance in high gain amplifiers. If the problem needs attention, you can use some of the following techniques.

- Choose high quality op amps with low V_{io} specifications (Look through linear data books.)
- If possible, work with AC signals rather than DC. For example, a pulsed (square wave) light source may be used, rather than a steady (DC) one.
- Use an offset nulling circuit, as explained below.

Offset nulling methods

If the offset voltage is a significant problem, you will have to cancel out its effect by using an offset nulling circuit. The general idea is to inject a small voltage into the input stage, just enough to cancel out the DC imbalances. The data sheets for each type of device usually specify the best circuit to null the offset, for that device. For example, the recommended circuit for the 741 type op amp (taken from the National linear data book) is shown in Figure 2. (Connect the ends of a 10 kΩ pot between pins 1 and 5 and connect the centre lead to
- supply).

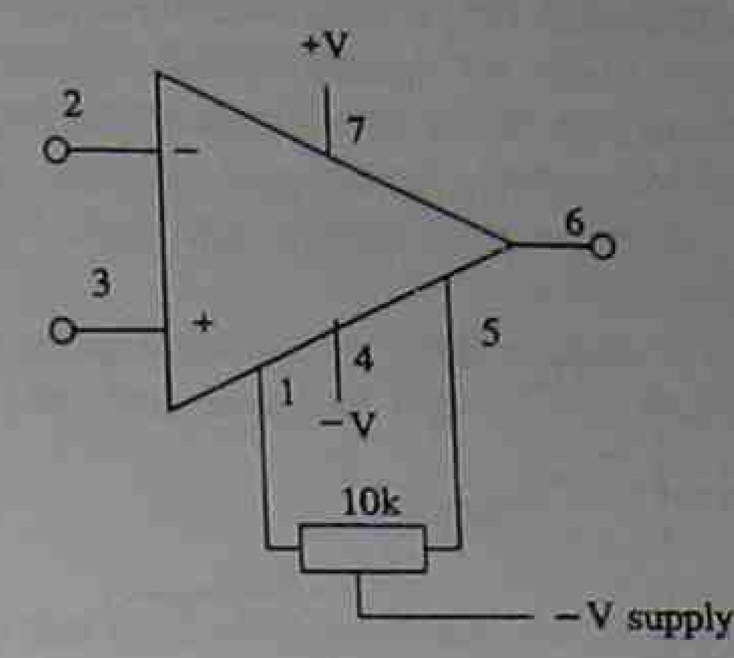


Fig. 2 Offset nulling for type 741 op amp.

After connecting the amplifier circuit and the offset nulling component, ground the input, observe the DC voltage at the output and adjust the 10 kΩ pot until output DC is zero.

Note that this circuit is not universal. It works for type 741 op amp and a few others. If you use other types of op amp, you have to check their data sheets to find the correct nulling circuit. Some common nulling circuits are given in Coughlin & Driscoll, p.244.

If the data sheets do not give any nulling circuit, you can use the following universal nulling circuit shown in Figure 3. This circuit effectively adds a small DC voltage at the input to oppose the internal DC offset voltage.

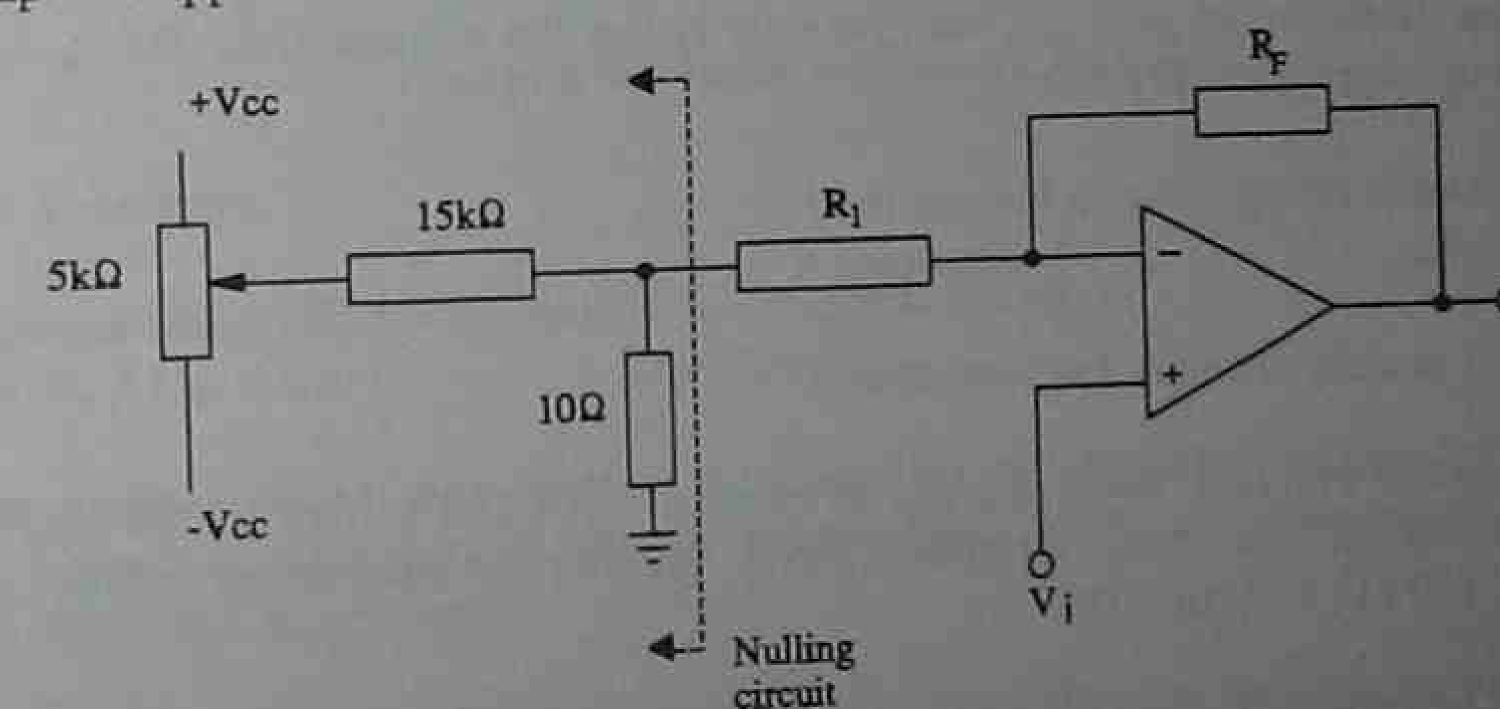


Fig. 3 Universal offset nulling circuit

Input bias current (I_B) and its effect on output DC voltage

Input Bias currents are another reason why the op amp may have a DC voltage at the output even if there is no input. Even though the ideal op amp is supposed not to draw any input current, the transistors at the input stage need small base biasing DC currents to work properly. These currents are called the input bias currents.

There are two input bias currents (one each for the inverting
inputs) called I_{B-} and I_{B+} (as in Figure 4). I_{B-} and I_{B+} are nearly, but not exactly, equal. Data sheets usually give the average of the two bias currents and call it the 'Input bias current', I_B. It is defined as:

$I_B = (I_{B+} + I_{B-}) / 2.$ Equation 2

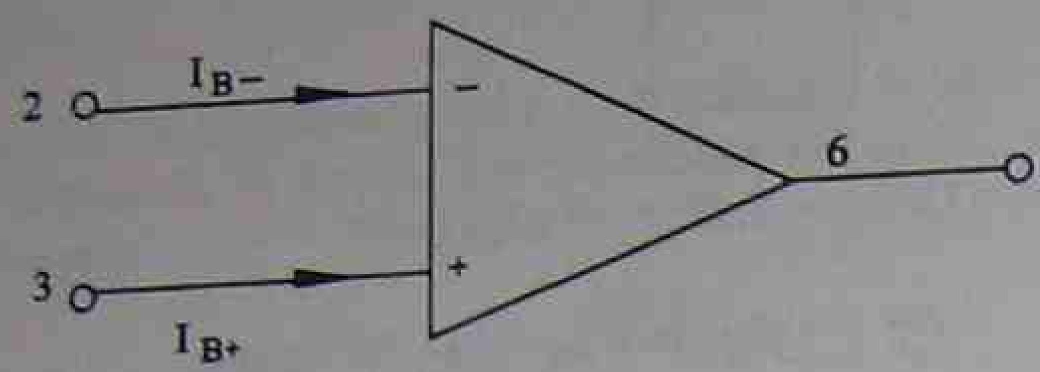


Fig. 4 Bias currents of an op amp

The value of I_B depends on the type of op amp. For general purpose op amps such as type 741, I_B may be about 100 nA. For FET input and 'superbeta' (i.e. Darlington) input op amps, I_B may be as low as 100 pA.

The input offset currents cause a DC voltage at the output because they flow through the external resistors in the circuit causing a DC voltage drop in them.

The DC output voltage (due to bias current)
$= I_{B-} * R_F$ Equation 3

If the data sheets do not give I_{B-} , but give only I_B (i.e. the average of I_{B-} and I_{B+}), then you can approximately take

The DC output voltage (due to bias current)
$= I_B * R_F$ Equation 4

Equation 3 is more accurate than equation 4.

In the circuit of Figure 1, supposing the op amp has input bias current of 100 nA, the output DC voltage due to the bias current alone, ignoring the input offset voltage, is 100 nA * 100 kΩ = 10 mV DC.

Methods to reduce the effect of input bias currents

- Choose an op amp with low bias currents specifications e.g. FET or superbeta input stages.
- Choose the smallest possible value of R_F. For example, you get the same gain by choosing R_F = 100 k and R_1 = 10 k, or by choosing R_F = 10 k and R_1 = 1 k. The second set of values give much less unwanted output DC offset voltage.
- Use bias current compensation as discussed below.

Bias current compensation and input offset current

We have seen that I_{B-} flowing through R_F causes an output DC offset voltage. This voltage can be partially cancelled by allowing the other bias current I_{B+} to flow through a resistor, thereby developing a voltage of the opposite polarity. This method of cancellation is called bias current compensation.

For bias current compensation, we need to place a bias current compensation resistor R_c in series with the non-inverting input. The value of R_c is calculated as

$R_c = R_1 \parallel R_F.$ Equation 5

As before, R_1 is the total effective resistance between the
-input and ground (including all series and parallel resistances in the path). Similarly, R_c includes the effect of any other resistance present between the +input and ground. After connecting R_c, no other circuit adjustment is necessary.
The value and location of R_c is the same for both inverting and non-inverting amplifiers (Figures 5 and 6).

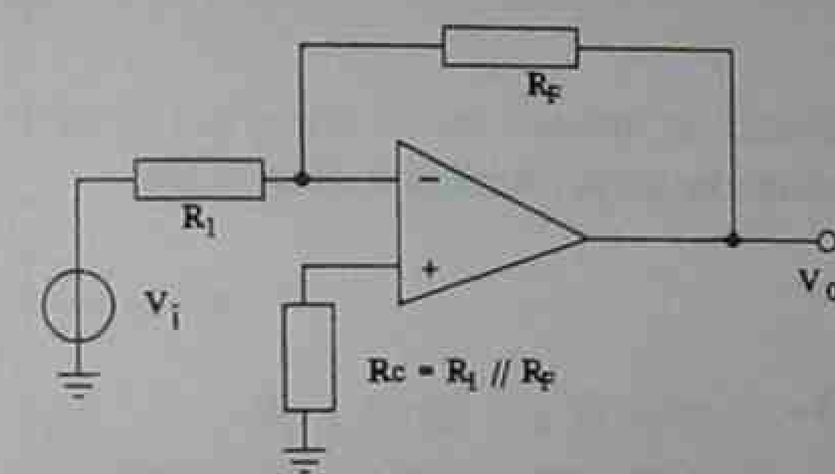


Fig. 5

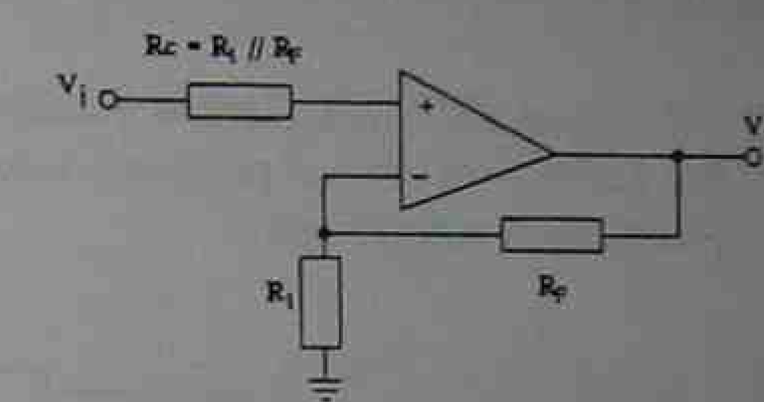


Fig. 6

Location of bias current compensation resistors

With R_c present,

the output offset DC voltage (due to bias currents alone)
$= R_F * (I_{B-} - I_{B+})$ Equation 6

Comparing equation 6 with equation 3, we see that the output DC voltage due to the bias currents is much reduced, because the two bias currents are nearly equal and their difference is very small.

The *input offset current* (I_{os}) is the magnitude of the difference between the two bias currents and its value is given in data sheets.
$I_{os} = |(I_{B-} - I_{B+})|$ Equation 7

Using the definition of the input offset current,
the output offset DC voltage (due to bias currents alone, when bias compensation resistor is present)
$= R_F * I_{os}$ Equation 8

The input offset current is **not** a real current which flows anywhere. It is just a mathematical concept representing the imbalance of I_{B+} and I_{B-}, which is useful because bias compensation is commonly used to minimise the effect of DC input bias currents.

You are reminded once more that equation 3 and equation 6 calculate the same thing. Equation 3 is used when bias compensation is not present and equation 6 used when it is present.

Example 1 : Bias current compensation and its effect

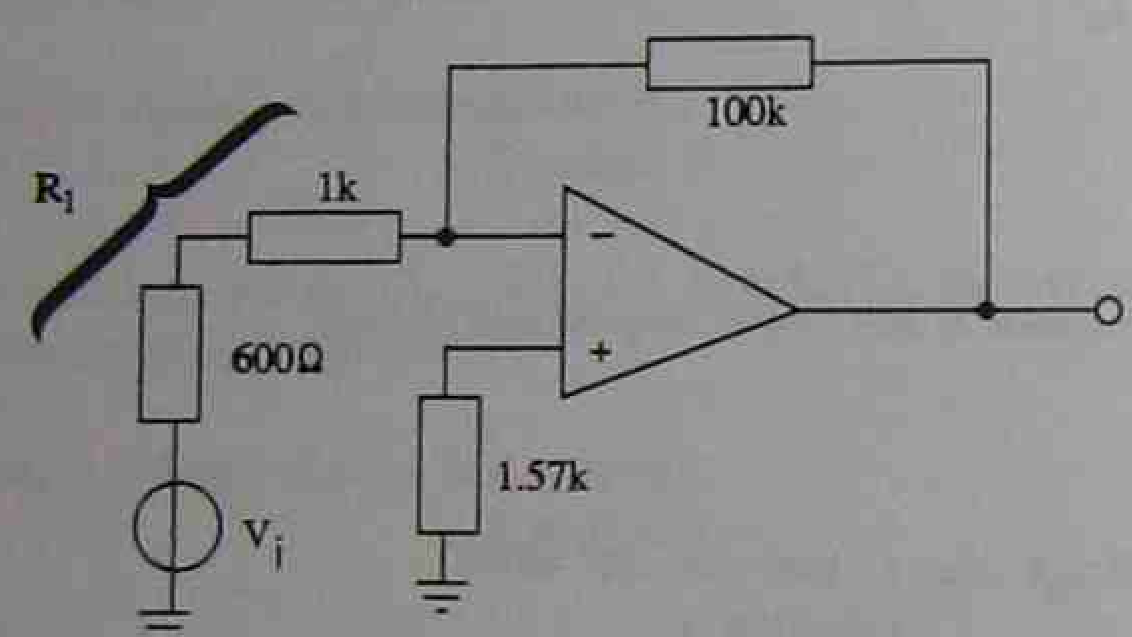
(a) For the circuit of Figure 1 design a bias current compensation.

(b) If the op amp has input offset current of 20 nA, what is the output DC offset voltage due to bias currents for the compensated circuit ?

Solutions

(a) $R_c = 100\ k\Omega \parallel (1k\Omega + 600\Omega) = 1.57\ k\Omega$

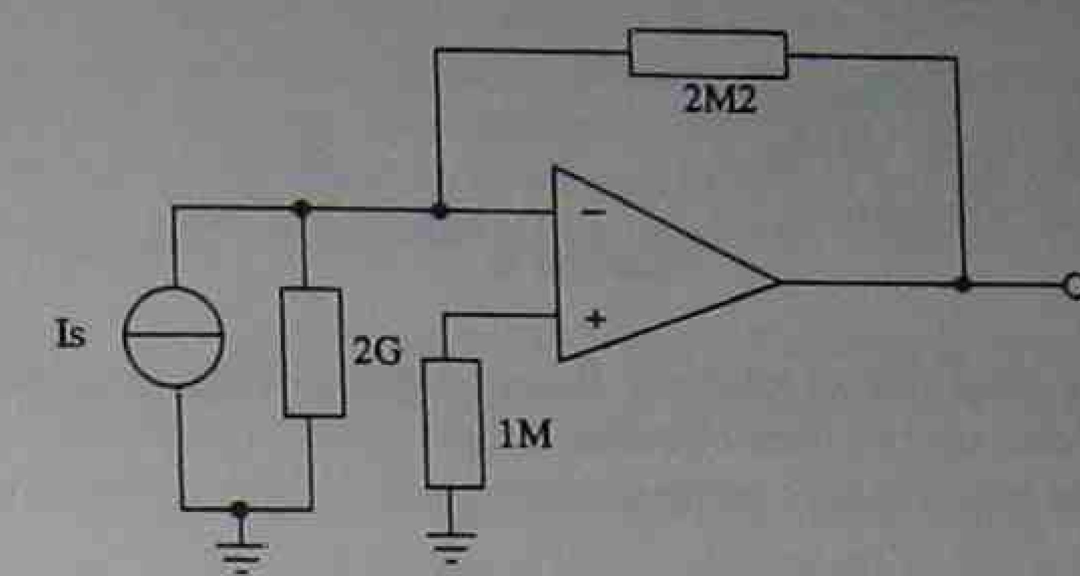
Put 1.57kΩ (or nearest preferred value) resistor in series with the +input for bias current compensation.



(b) $V_{o(DC)}$ due to bias currents with compensation = $100\ k\Omega * 20\ nA = 2\ mV$

Example 2 : Bias current compensation and its effect

(a) Design a bias compensation circuit for the following circuit.



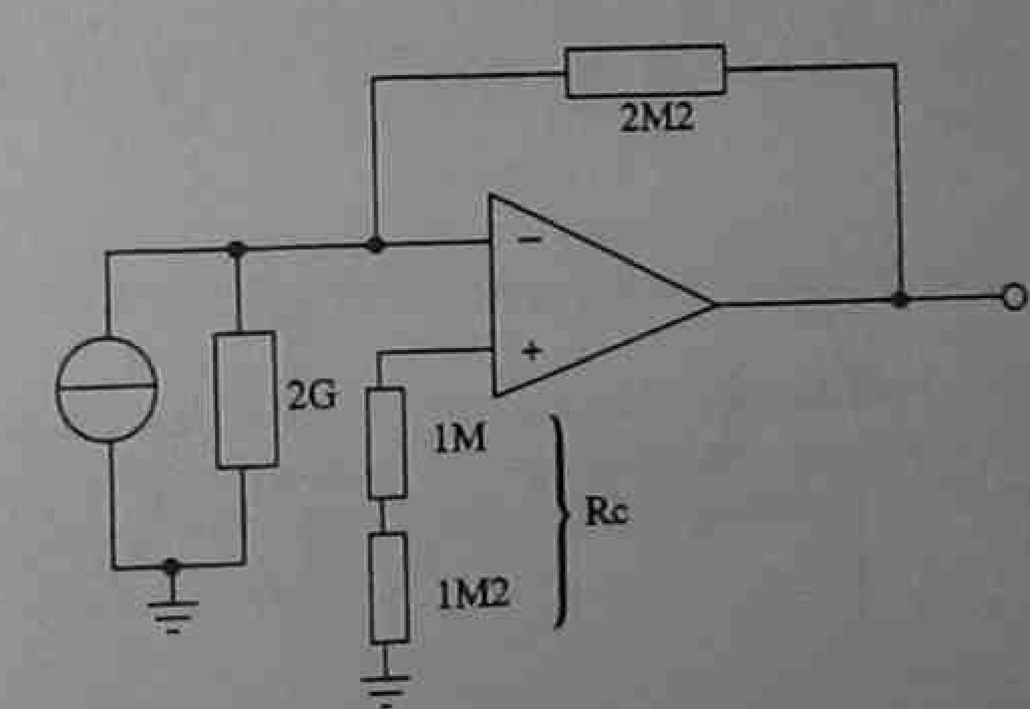
(b) If the input offset current is 10 nA, what is the output DC offset voltage due to bias currents after compensation ?

Solutions

(a) $R_c = 2M2 \parallel 2\ G = 2M2$

Resistance already present from +input to ground = 1M

∴ Place a resistor 2M2 - 1M = 1M2 in series with the +input as shown.



(b) $V_{o(DC)}$ due to bias currents = 10 nA * 2M2 = 22 mV

Note: R_c has no effect at all on the voltage gain or input resistance of the amplifier.

Total output DC offset voltage due to input offset voltage and input bias currents

The input offset voltage and input bias currents both add unwanted DC voltages at the output. These two effects are independent of each other, and both may be + or - depending on the individual device. With luck, these two effects may partly cancel each other, but in the worst case, the two effects will be additive.

Worst case total output DC offset voltage =
output DC due to input offset voltage +
output DC due to input bias currents

Equation 9

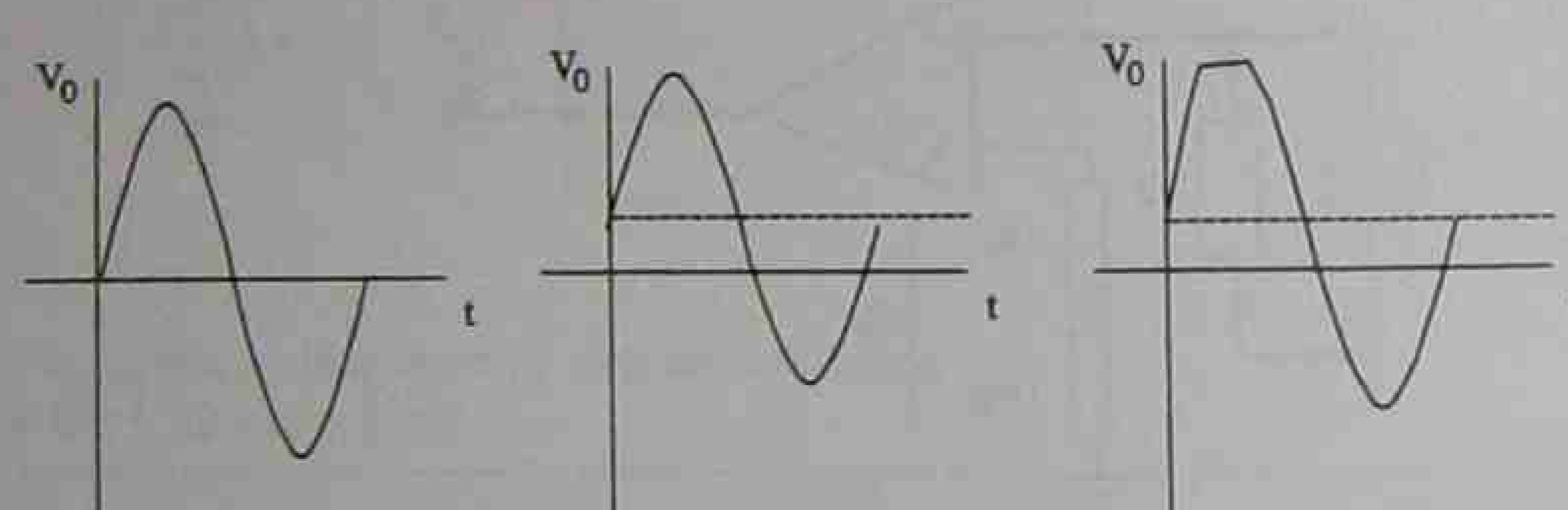
The first term on the right hand side of equation 9 is calculated using equation 1. The second term is calculated using either equation 6/equation 8 or equation 3/equation 4, depending on whether bias compensation resistor is used or not used.

For the circuit in Figure 1, supposing that V_{io} = 3 mV and I_B = 100 nA, and there is no bias current compensation, the worst case DC output offset voltage is 187.5 mV + 10 mV = 197.5 mV.

For the circuit in Example 2, supposing that V_{io} = 0.5 mV, I_{os} = 10 nA and I_B = 40 nA, and the circuit is bias compensated, the worst case DC output offset voltage is 0.5 mV * (1 + 2M7/2G) + 10nA * 2M2 = 22.5 mV.

Effect of input offset voltage on AC signals

Input offset voltage and current are DC effects and have no direct effect on AC signals. The output AC signal will be level shifted by an offset, as given by equation 9. The output DC offset could cause problems in further processing unless blocked out suitably (remember that the op amp can amplify DC), and also, reduce the maximum output AC voltage swing because one half of the AC will clip before the other half, as shown in Figure 7.



(a) ideal output *(b) level shifting due to offset* *(c) reduced swing due to offset*

Fig. 7 Effect of DC offsets on AC signals

Drift in offset voltage and offset current

The input offset voltage and bias currents vary with temperature and with supply voltage. Such variation is called drift. Because of drift, the output DC offset voltage also changes with temperature or supply voltage changes.

Data sheets specify temperature drifts for various device types (µV/°C for input offset voltage; pA/°C for input offset current or input bias current). Usually V_{io} increases with temperature for all types of op amps. I_B and I_{os} decrease significantly with temperature for BJT amps and increase rapidly with temperature for FET amps. The significance of this is that even if we carefully null out the output DC offset at room temperature, it won't stay nulled when the device temperature changes.

Change in input offset voltage = Drift in V_{io} * change in temperatureEquation 10
Change in input offset current = Drift in I_{os} * change in temperatureEquation 11

The power supply rejection ratio (PSRR), also called supply voltage rejection ratio, is sometimes specified in data sheets as the change in V_{io} for a 1V change in supply voltage e.g. 50µV/V. In other data books, PSRR is given as so many dB's e.g. 96 dB. These two definitions are unfortunately inverses of each other. (PSRR of 96 dB = antilog(-96/20) = 15.8µV/V). The significance of PSRR is that if the power supply is not well stabilized, both V_{io} and the output DC offset will drift.

Change in input offset voltage = change in supply voltage * PSRR (following the first definition)

Equation 12

Note that if there is AC ripple in the power supply, or if the supply leads pick up noise, it will show up as extra noise in the output. This is why you are advised to put bypass capacitors from supply pins of IC to ground.

Change in output offset voltage can be calculated using equations 1, 6 and 9 except that we must use changes in V_{io} or I_{os} instead of V_{io} or I_{os}.

Example 3 : Drift in output DC offset due to change in temperature

A bias compensated amplifier has R_F = 100 kΩ and R_1 = 1600 Ω. The maximum drift in input offset voltage is 30 µV/°C and the maximum drift in input offset current is 300 pA/°C. If the circuit is offset nulled at 20°C, what is the worst case output DC offset voltage at 80°C ?

Solution

Change in V_{io} = 30 µV/°C * (80° - 20°) C = 1.8 mV
Change in I_{os} = 300 pA/°C * (80° - 20°) C = 18 nA

∴ worst case change in output DC offset
= 1.8 mV * (1 + 100k/1.6k) + 18 nA * 100kΩ
= 116.1 mV

Example 4 : Drift in output DC due to change in power supply voltage

In example 3, the op amp has PSRR of 95dB. If the supply voltage changes by 2V, what will be the change in the output DC voltage ?

Solution

change in V_{io} = 95dB down from 2V = 2V/antilog(95/20) = 2V/56234
= 35.6μV

∴ change in output DC offset
= 35.6μV * (1+100k/1.6k)
= 2.3 mV

The best protection against drift problems is to keep the op amp as cool as possible (by heat sinking, fan cooling, keeping power transistors and regulators away from the op amp etc.), to use stable power supplies and bypass supply pins to ground with capacitors. For precision work, very low drift op amps are available. If you can manage to work with purely AC signals, the whole issue is not very significant.

Summary

1. The input offset voltage and input bias currents cause an unwanted DC offset voltage (DC level shifting) at the output.

2. The output DC voltage due to input offset voltage increases with the voltage gain of the circuit. The effect is the same for both inverting and non-inverting amplifiers (equation 1).

3. The effect of input offset voltage can be nullified by using the recommended offset nulling circuit for the device.

4. The output DC voltage due to input bias currents increases with the value of the feedback resistor (equation 3 or 4).

5. The effect of input bias currents can be minimized by using small value resistors and by using a bias compensation resistor in series with the +input (equation 5).

6. The input offset current is the difference between the two bias currents. Its value is much less than those of the bias currents. With bias current compensation, the output DC voltage due to bias currents depends on the input offset current and so is much reduced (equations 6 and 7).

7. The total output DC offset voltage is the sum of the effects due to the input offset voltage and bias currents.

8. Drift refers to the variation in the input offset voltage and input bias currents with temperature or supply change. Its effect on the change in output DC voltage is calculated in a similar way to equation 9.

Review questions

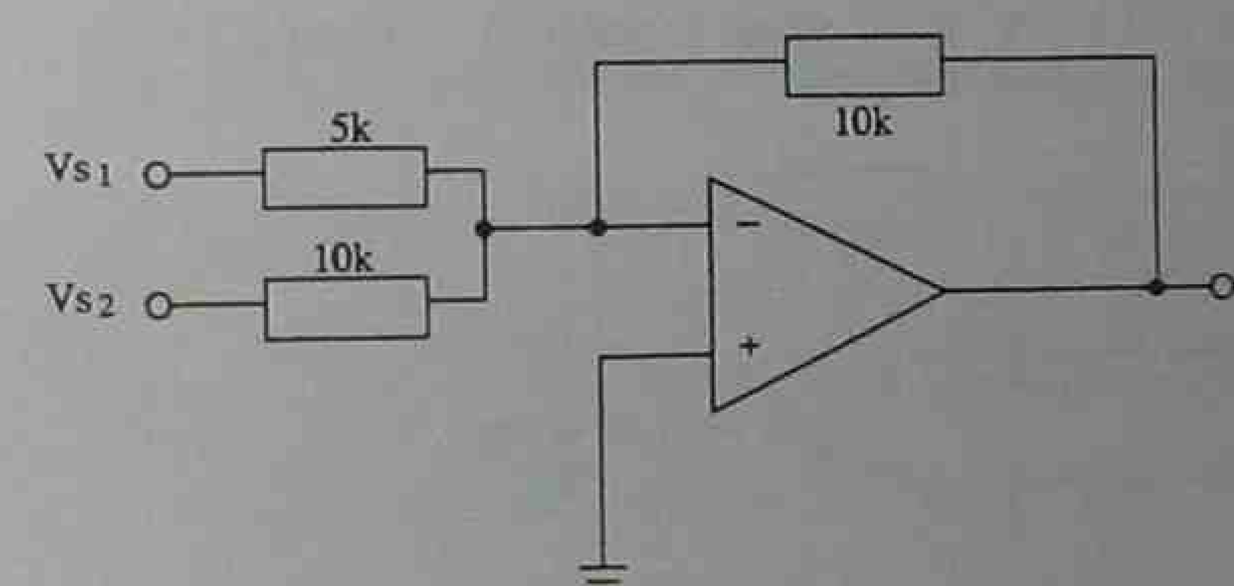
These questions will help you revise what you have learnt in Section 2.

1. For the summing circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

(a) Calculate the worst case output DC offset voltage.

(b) Sketch a suitable bias compensation for this circuit and calculate the added component value.

(c) Calculate the worst case output offset DC voltage after bias compensation.

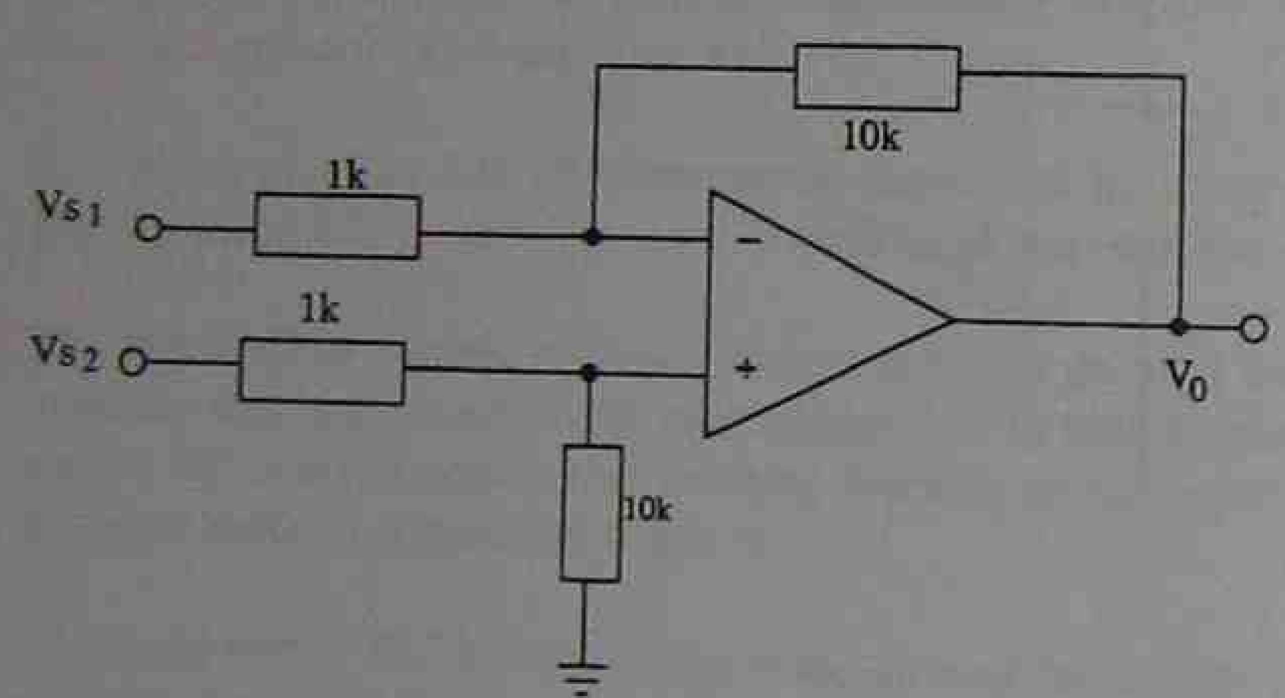


2. For the differential amplifier circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

(a) Show that the circuit is properly bias compensated.

(b) Calculate the worst case output DC offset voltage.

(c) The drift in the input offset voltage is 6 μV/°C and the drift in the input offset current is 100 pA/°C. If the circuit is nulled at 20°C, calculate the output DC offset voltage at 70°C.



3. Briefly explain the effect of DC offsets on (i) DC input signals and (ii) AC input signals.

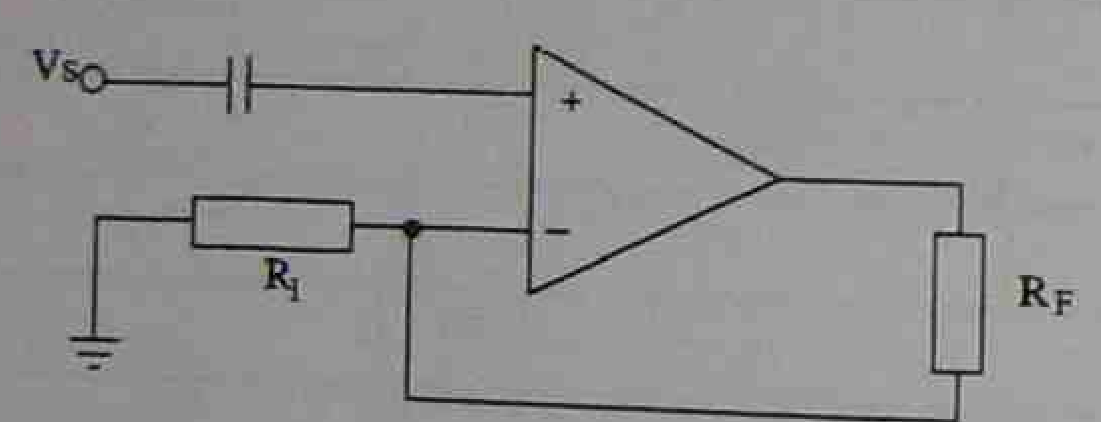
4. What are common ways to avoid errors due to drift in DC offsets in amplifiers?

5. What are the common methods to reduce the errors due to input offset voltage ?

6. What are the common methods to reduce the errors due to input bias currents?

7. From a linear data book, find (i) an op amp type with $V_{io} \leq 1$ mV and (ii) an op amp type with $I_{os} \leq 1$ nA.

8. The following circuit will not work. Why?



9. Refer to the circuit of Q.2 above. If the PSRR of the op amp is 30μV/V, and the power supply has unfiltered ripple of 4V p-p, what will be the output ripple?

Skill practice 3
DC offset effects

Suggested duration

1 hour 30 minutes

Tasks

- To measure the input offset voltage of an op amp
- To measure the input bias currents of an op amp
- To measure the input offset current of an op amp
- To null the output DC offset voltage of an op amp

Equipment

- 1 of type 741 op amp (Other types can be used. Check their data sheets for pin connections, recommended offset null circuit, and whether an external compensating capacitor is needed.)
- ±15 V DC power supplies
- resistors : 2 of 47Ω, 1 of 4k7, 2 of 1M, 1 of 10k potentiometer (multiturn preferred)
- DC voltmeter

Additional Information

The input offset voltage and offset current cannot be measured directly. The bias currents, being very small, cannot be measured using common laboratory instruments. These parameters must be calculated by measuring the DC output voltage and working backwards to the input.

For the pin connections of the 741 op amp and the method of connecting the dual power supply, check the skill practice exercises of Section 1. Though circuit diagrams usually do not show power supply connections, they are supposed to be there.

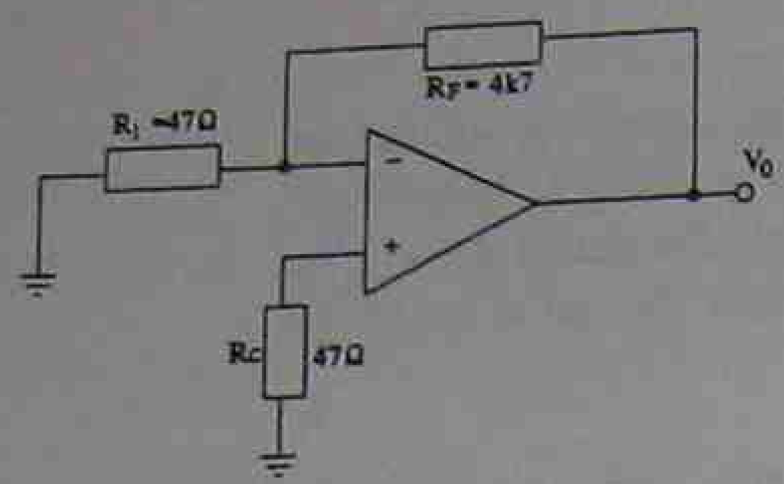
No external signal source is necessary. In the following circuits, the input is connected to ground.

(For Steps 1, 2, 3 and 4, if you do not have the recommended resistors, use others of nearby value. For Step 5, you must use a 10 kΩ pot.)

Procedure

Step 1 *Measurement of input offset voltage, V_{io}*

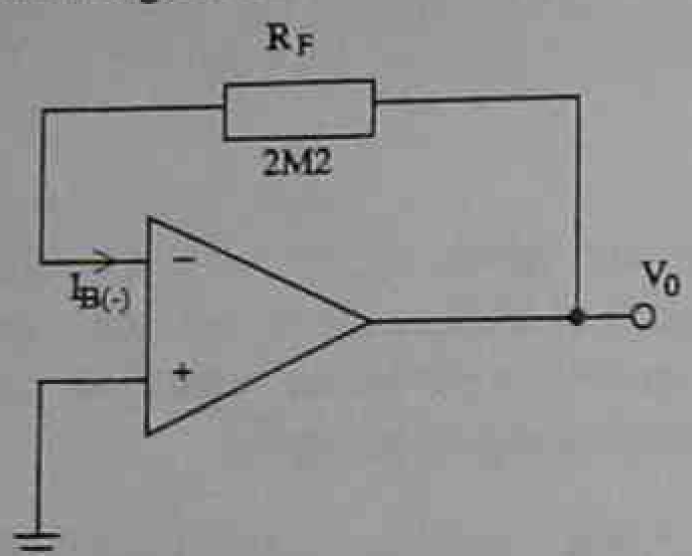
- Connect the following circuit and measure the output DC voltage V_o.



- Calculate $v_{io} = V_o / (1 + R_F / R_1) =$

Step 2 *Measurement of bias current I_{B-}*

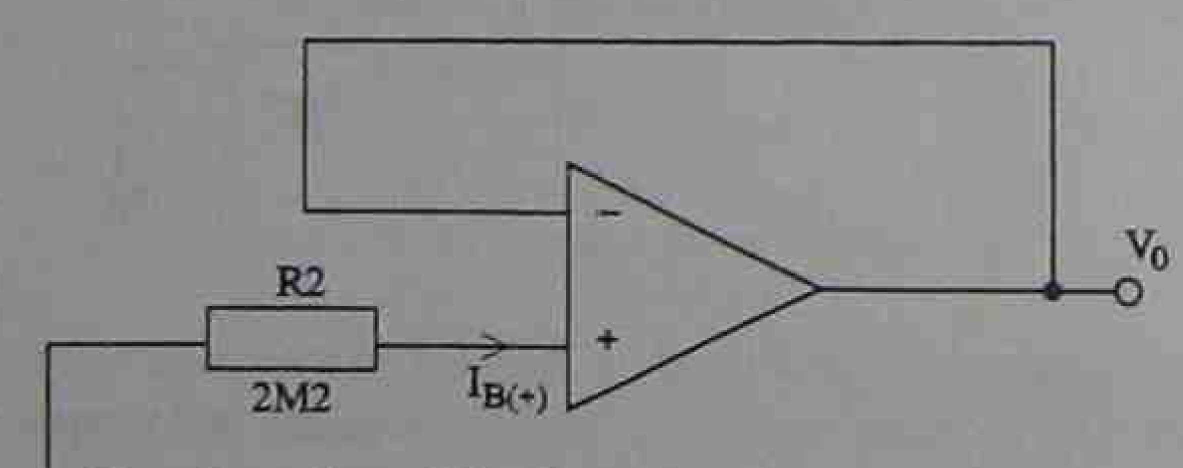
- Connect the following circuit and measure the output DC voltage V_o.



- Use the value of V_{io} from Step 1 and calculate
  $I_{B-} = (V_o - V_{io}) / R_F =$

Step 3 *Measurement of bias current I_{B+}*

- Connect the following circuit and measure the output DC voltage V_o.

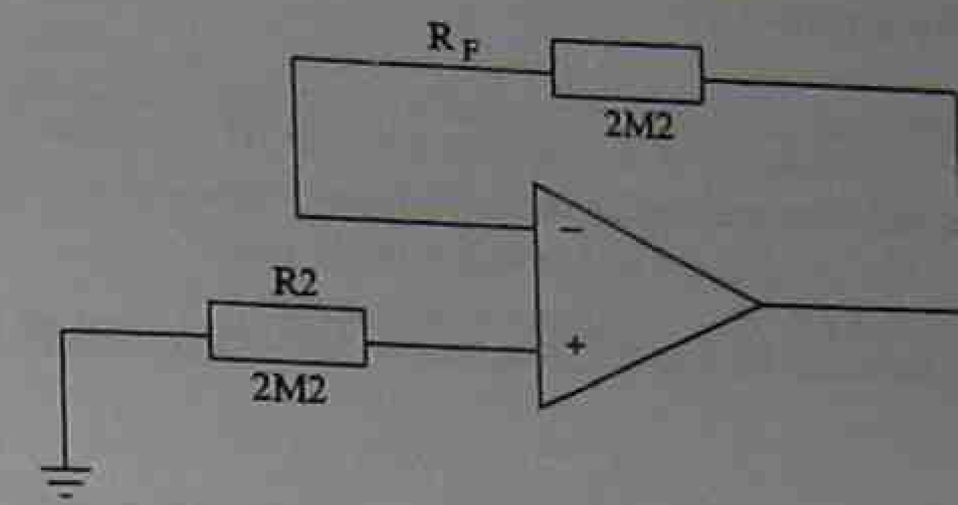


Use the value of V_{io} from Step 1 and calculate

$I_{B+} = - (V_o - V_{io}) / R_2 =$

Step 4 *Measurement of input offset current I_{os}*

- Connect the following circuit and measure the output DC voltage V_o.



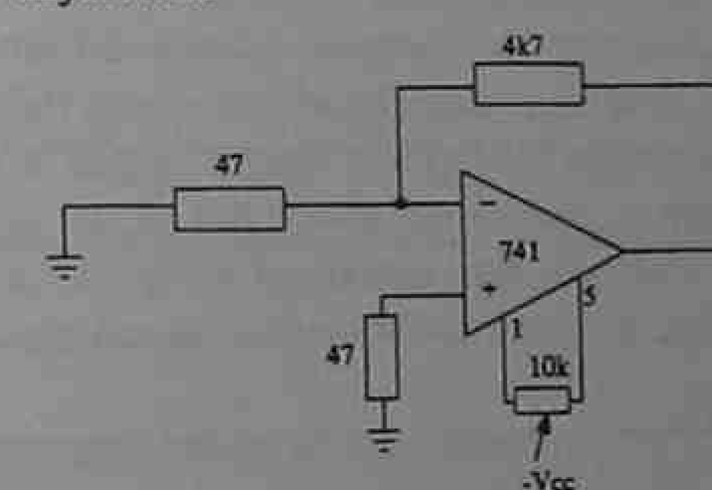
- Use the value of V_{io} from Step 1 and calculate

  $I_{os} = |(V_o - V_{io})| / R_F =$

- You can also use the values of I_{B-} and I_{B+} from Steps 2 and 3 and calculate

  $I_{os} = |I_{B+} - I_{B-}| =$

Step 5 *Offset null adjustment*



- Connect the circuit. (This circuit only works for 741 op amps. If you use any other types, check the data sheets for the correct circuit. The procedure is the same for all types of op amp.)
- Connect a DC voltmeter or CRO in DC mode to the output.
- Adjust the 10 k pot until the output DC voltage is zero.

Discussion questions

1. Compare your experimental results for V_{io}, I_B and I_{os} with data sheets specifications for your op amp.

2. Compare the output DC voltages in Steps 2 and 4 and state the effect of bias compensation.

3. Why are the resistors in Step 1 chosen to be very small?

4. Which of the two methods for measuring I_{os} (Step 4) is more accurate? Why?

Section 3: Slew rate

SUGGESTED DURATION	PREAMBLE
4 hrs 30 mins	To develop qualitative and quantitative understanding of the distortion caused by slew rate limitation; to interpret data sheet information relating to slew rate; and select components to minimise this distortion.

Objectives

At the end of this section you should be able to:

- ☐ understand slew rate and its effect on amplifier performance, and in particular:
  - define slew rate and describe its significance in applications
  - state the conditions under which slew rate distortion is prominent
  - sketch the effect of slew rate on square wave outputs of amplifiers
  - state and use the formula relating output voltage step change, rise time and slew rate
  - sketch the effects of slew rate on sine wave outputs of amplifiers
  - state and use the formula relating the output voltage sine amplitude, maximum undistorted frequency of operation and slew rate
  - define full power bandwidth and calculate
  - given the output voltage swing vs frequency graph of an amplifier, read the value of the full power bandwidth, maximum available voltage swing at any given frequency and calculate the slew rate.
  - state some common methods to improve the slew rate performance
- ☐ measure the slew rate of an amplifier
- ☐ observe the improvement in slew rate by varying the compensation capacitor in an externally compensated op amp.

References

The following references deal with topics in this section.

1. Jacob (1993), pp 195-199
2. Rutkowski (1994), pp 97-101
3. Gayakwad (1993), pp 225-231
4. Coughlin & Driscoll (1993), pp 260-264

Introduction

In many applications, amplifiers work with step changes in the input voltage e.g. sharp changes in music levels for audio amplifiers; switching the output from off to on or the other way in amplifiers used for interfacing to digital circuits. If we apply a step input, we would expect a step output (Figure1). In reality, the output of the amplifier does not respond instantly to the input (Figure 2).

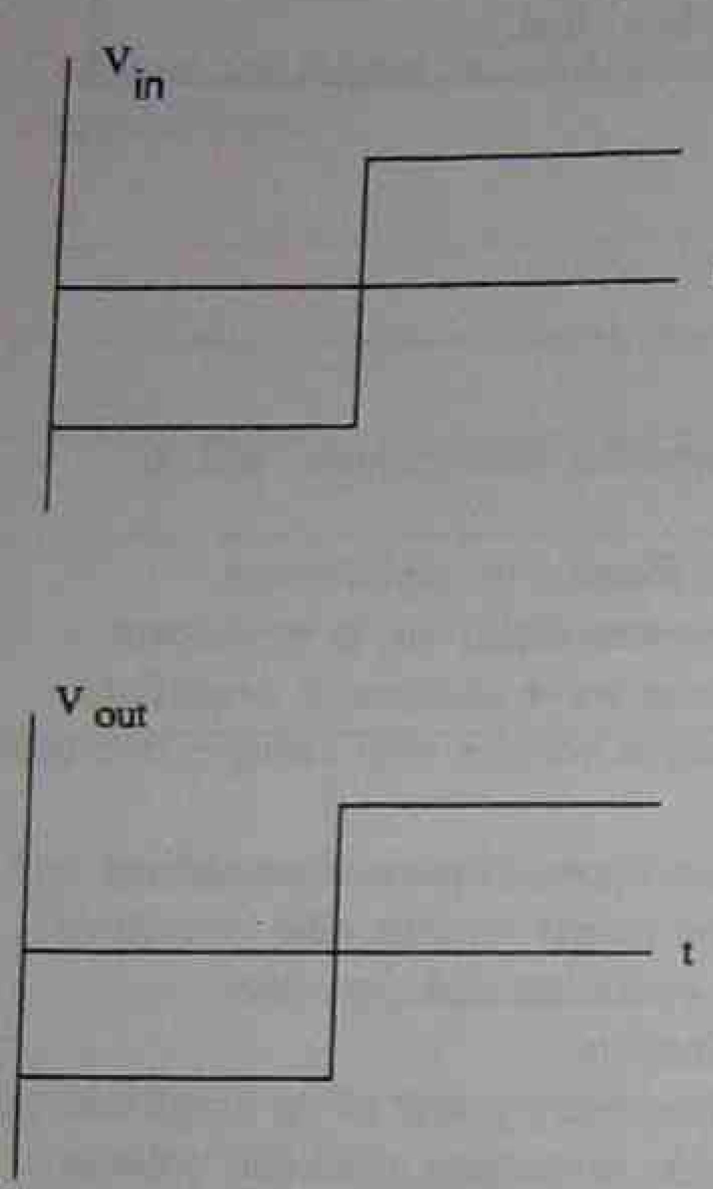


Fig. 1 (Ideal output)

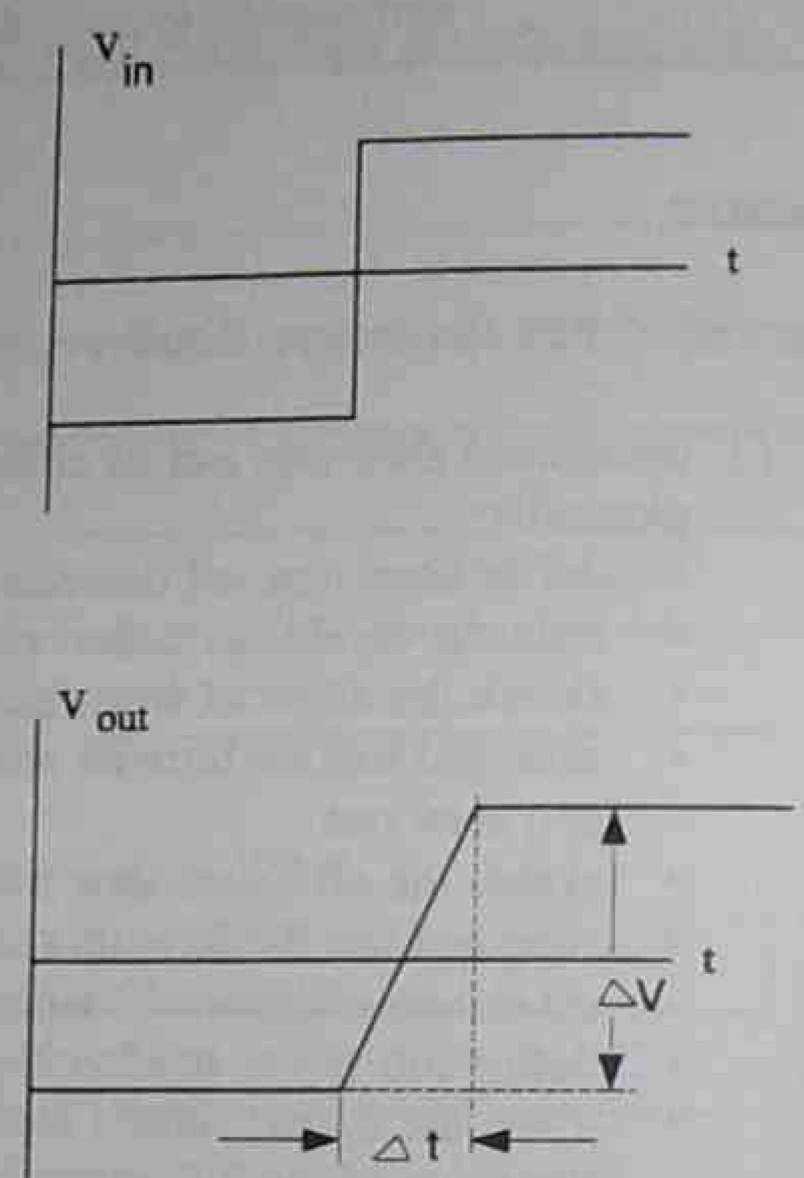


Fig.2 (Slew rate limited)

The non-zero 'catch-up' time required by the output is due to a characteristic called the *slew rate.* The slew rate of the amplifier is the maximum rate of change of the output voltage. In other words, the output cannot rise or fall faster than the slew rate.

In Figure 2, the output is rising as fast as it can, so

the slew rate $= \Delta V / \Delta t$ (Volts / µs)
= change in output voltage / rise time — Equation 1

Note that usually the time unit in the denominator is µs, not seconds. For example, usually we write 0.5 V/µs rather than 500 000 V/s.

Cause of slew rate

There are some small internal capacitances in any amplifier. In some amplifiers, we have to connect small external capacitors to ensure proper operation. These capacitances have to charge up to the correct voltages before the output can settle down. It is the non-zero charging time of the capacitances that causes the slew rate.

Methods to improve slew rate

A high slew rate is usually desirable, because it means that the output can respond quickly to a change in the input. However, you should be aware that a high slew rate is sometimes accompanied by ringing. This means that the output overshoots and undershoots around the final value for a while before settling down. Methods which eliminate ringing usually also reduce the slew rate.

One obvious way to have a high slew rate is to choose an amplifier with a high slew rate specification. The common 741 op amp has a rather poor slew rate of only 0.5 V/µs. Other direct replacements for the 741 such as the LF351 have much higher slew rates (about 10 V/µs). There are high speed op amps which have slew rates ranging from 50 V/µs to more than 1000 V/µs.

In some 'externally compensated' op amps, the 'compensating' capacitor C_c (which affects the slew rate) can be chosen by the user, subject to some criteria which we discuss in a later section. With such op amps, quite large slew rates can be obtained at small expense.

Effect of slew rate on square wave response

Figure 2 shows the effect of slew rate on a square wave output, which is distorted and becomes a trapezoid shape. If the input frequency is high enough, the slew rate limited output never actually reaches the flat part of the square wave before the input steps to the other level; so the result is a triangular wave. However, even if the square wave frequency is low, it will still have two sharp edges and these will be slew rate limited at the output, though the distortion will be less noticeable if you see it in the CRO.

It is important to understand that slew rate limitation has its greatest effects only on large and/or high frequency output signals. Small output signals also have a rise time but this is due to bandwidth limitation, which we study later. We will postpone discussion of what is meant by 'large' or 'small' signals. As a rule of thumb, if the output waveform due to a step looks like Figure 3 or Figure 4, then it is small signal (bandwidth limited). If it looks like Figure 5, then it is a large signal (slew rate limited).

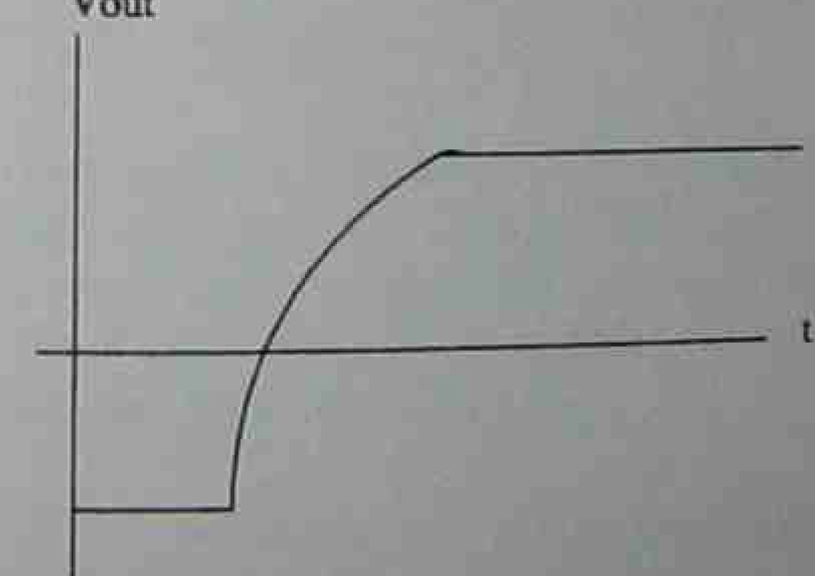


Fig.3

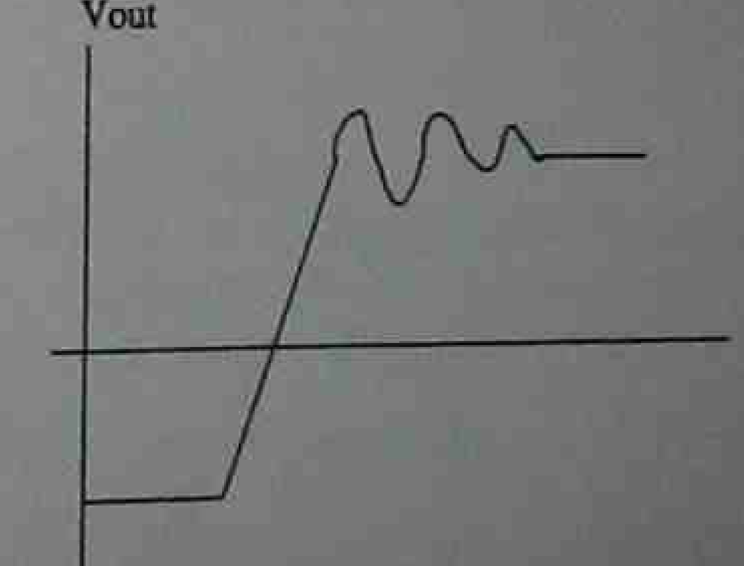


Fig.4

Small signal (bandwidth limited) square wave outputs

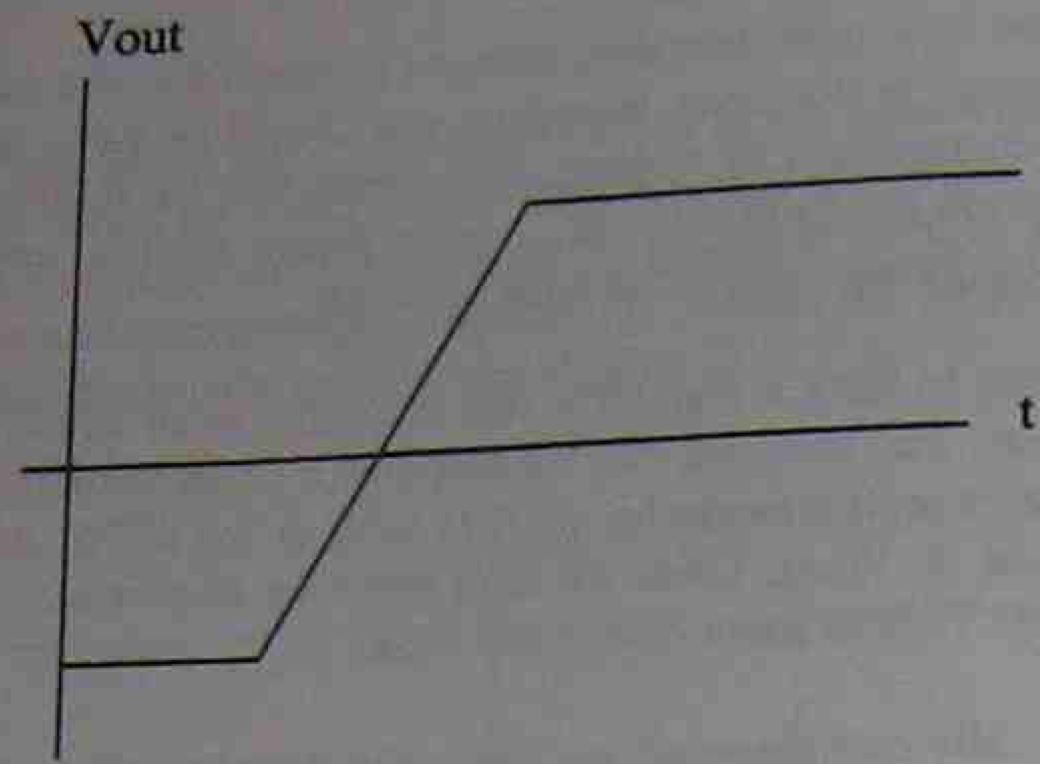


Fig.5. Large signal (slew Rate limited) square wave output

Effect of slew rate on sine wave response

If a sine wave has large amplitude and/or frequency, it is possible that the rate of change of some parts of the sine wave, especially near the zero crossing, will exceed the slew rate and thus cause distortion. This is shown in Figure 7 and 8. Figure 6 shows the undistorted sine wave output.

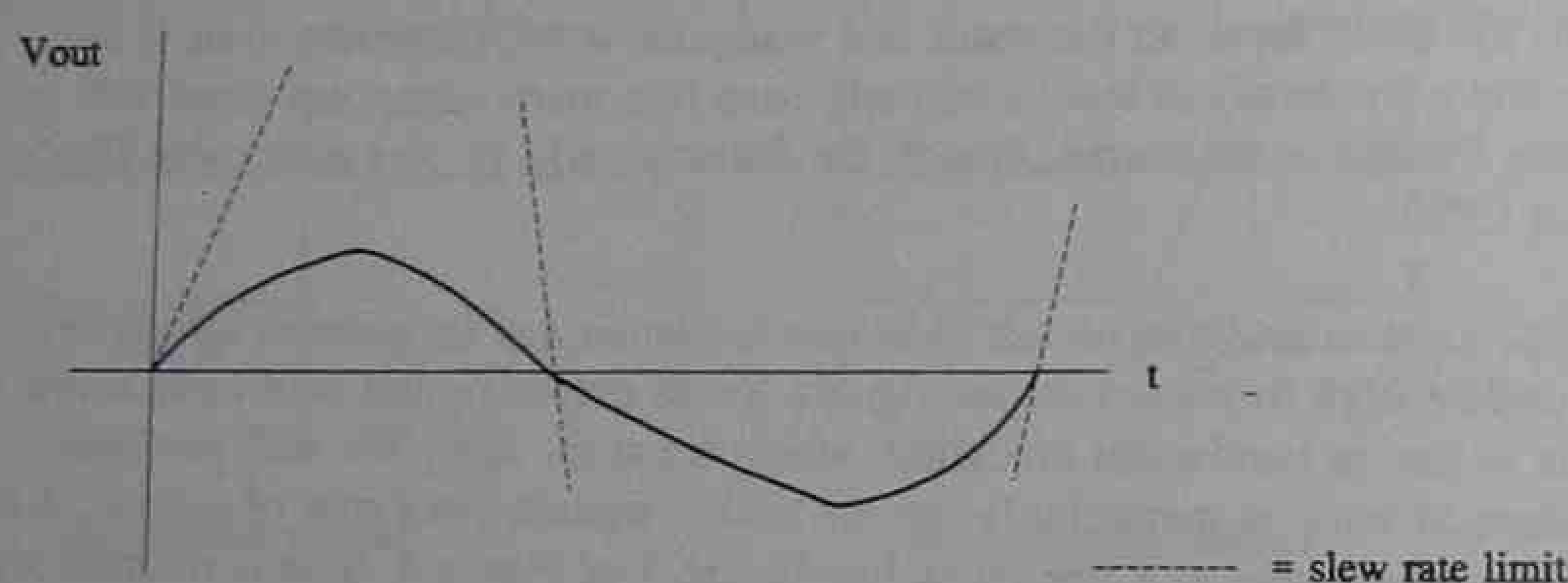


Fig.6 Undistorted sine wave output

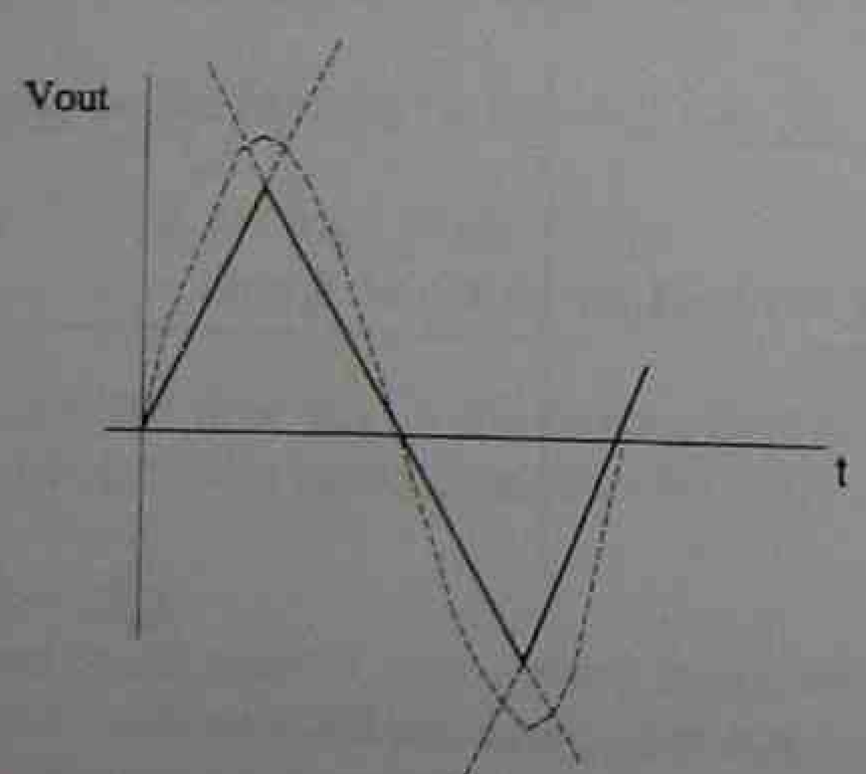


Fig. 7 High frequency sine waves are slew rate distorted

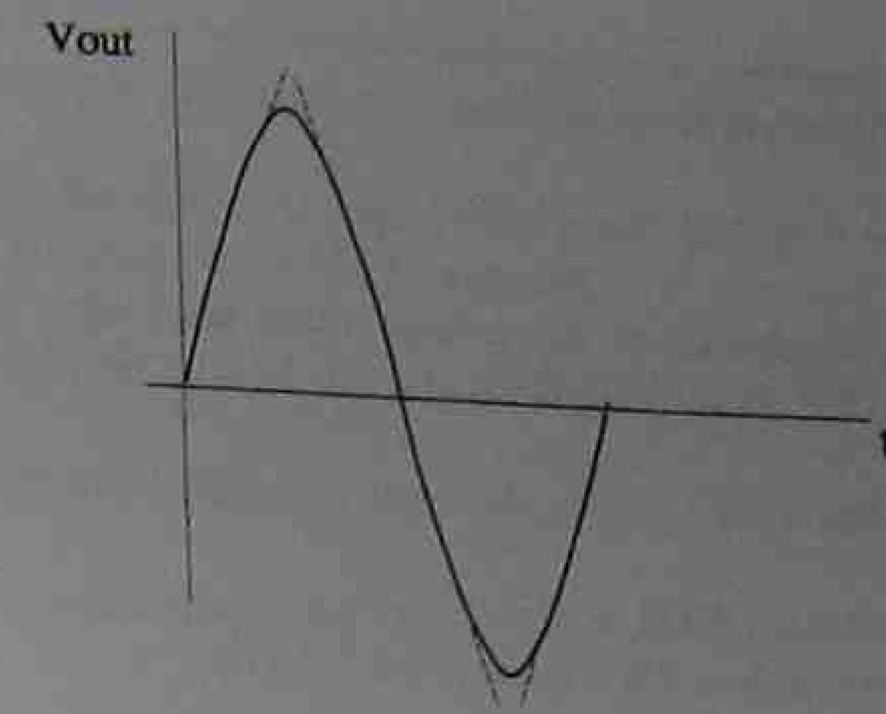


Fig.8. High amplitude sine waves are slew rate distorted

Again, as in the case of square waves, we see that slew rate is a large signal and high frequency problem. The formula relating the slew rate SR, the output *amplitude* (i.e. 0 to peak) $V_{o\ amplitude}$ and the maximum allowable frequency without suffering slew rate distortion f_{max} is:

$SR = 2 * \pi * V_{o\ amplitude} * f_{max}$ Equation 2

This equation simply shows that the maximum achievable amplitude and maximum achievable frequency, without slew rate distortion, are inversely related. If you need a large output voltage swing, you have to keep the frequency low. If you have to work with large frequencies, you have to keep the amplitude low.

If an amplifier is required to output sine waves of 12 V rms at 20 kHz the minimum slew rate it must have is

$2 * \pi * 12\sqrt{2}\ V * 20\ kHz = 2.13\ V/\mu s$

The time taken by a comparator with a slew rate of 1.2 V/µs to switch from +12 V to -12 V is:

$24\ V / (1.2\ V/\mu s) = 20\ \mu s$

Full power bandwidth and output voltage swing graph

The full power bandwidth (FPBW) of an amplifier is the largest sinewave frequency without causing slew rate distortion, when the output amplitude is the maximum unclipped value (about one volt less than the supply voltage). The FPBW can be calculated using equation 2.

$FPBW = SR / (2 * \pi * v_{o\ max\ ampl})$ Equation 3

Example : Full power bandwidth

Question

Calculate the full power bandwidth of an op amp which has a slew rate of 0.8 V/µs and works with ± 15V power supplies.

Solution

Max output amplitude = 15-1 = 14V

Full power bandwidth = 0.8 V/µs /(2 * π * 14V)

= 9.1 kHz

Some op amp data sheets give a graph of 'maximum output voltage swing as a function of frequency'. A typical graph (for the 741 op amp) is shown in Figure 9.

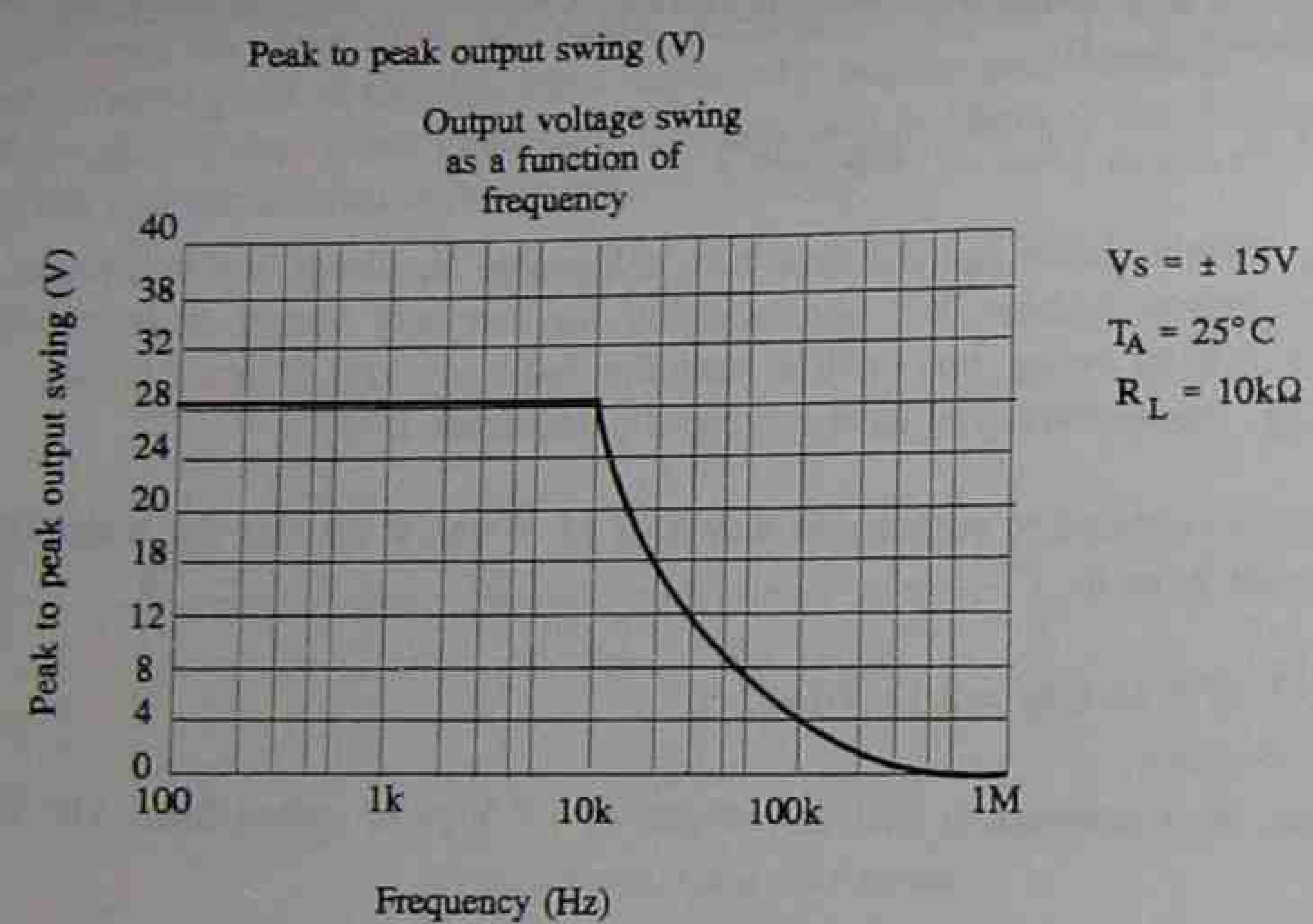


Fig. 9. Output voltage swing graph of the 741 op amp

The flat part of the graph (up to 10 kHz) shows that the maximum output (about 28V peak to peak or 14V amplitude) is limited by the DC supply voltage (clipping limited). At higher frequencies, slew rate limitation takes effect and the maximum undistorted swing is reduced (as in equation 2). The frequency where the two curves meet (10 kHz in Figure 9) is the full power bandwidth.

Take care in reading the graph in Figure 9. In some data sheets, the Y-axis may give the peak-to-peak output voltage; in others, the amplitude. The X-axis is a log frequency scale, whose markings may go 1,2,3,4,5,10,20..... The markings for 6,7,8,9 may not be shown, to keep the graph uncluttered.

Example 2 : Maximum undistorted output voltage calculations

An amplifier with the output voltage swing graph in Figure 9 is wired up with a voltage gain of 10, using ± 15 V DC supplies.

(a) What is the slew rate of the amplifier ?

(b) What is the maximum rms input voltage at 1 kHz without output distortion ?

(c) What is the maximum rms input voltage at 20 kHz without output distortion ?

Solutions

(a) FPBW = 10 kHz

∴ SR = 2 * π * (28V/2) * 10 kHz

= 0.88 V/µs

(b) From the graph, at 1 kHz, max. $v_{o\ p\text{-}p}$ = 28V

∴ max v_i rms = 28V / (2 √ 2) / gain = 1 V rms

(The output is clipping limited below FPBW)

(c) From the graph, at 20 kHz, max. $v_{o\ p\text{-}p}$ = 14V

max v_i rms = 14V / (2 √ 2) / gain = 0.5 V rms

The output is slew rate limited above FPBW

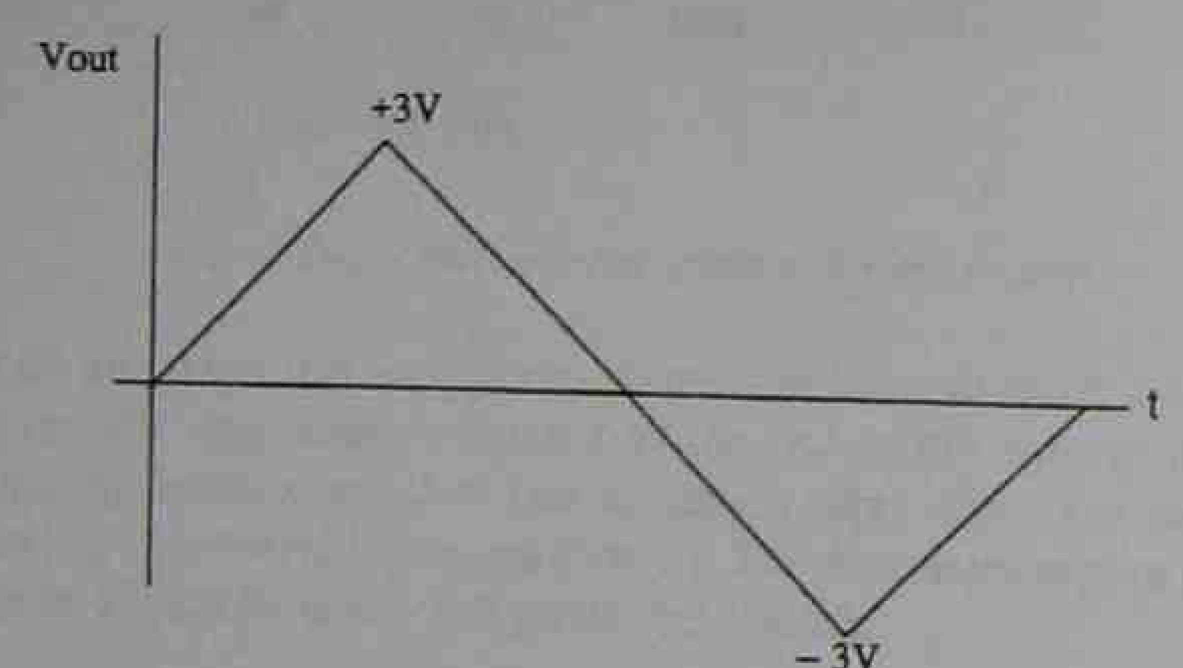
(Note : you could have calculated the answer to part (c) using the slew rate and equation 2. The answer would be about the same, except for the human error in reading graphs.)

Review questions

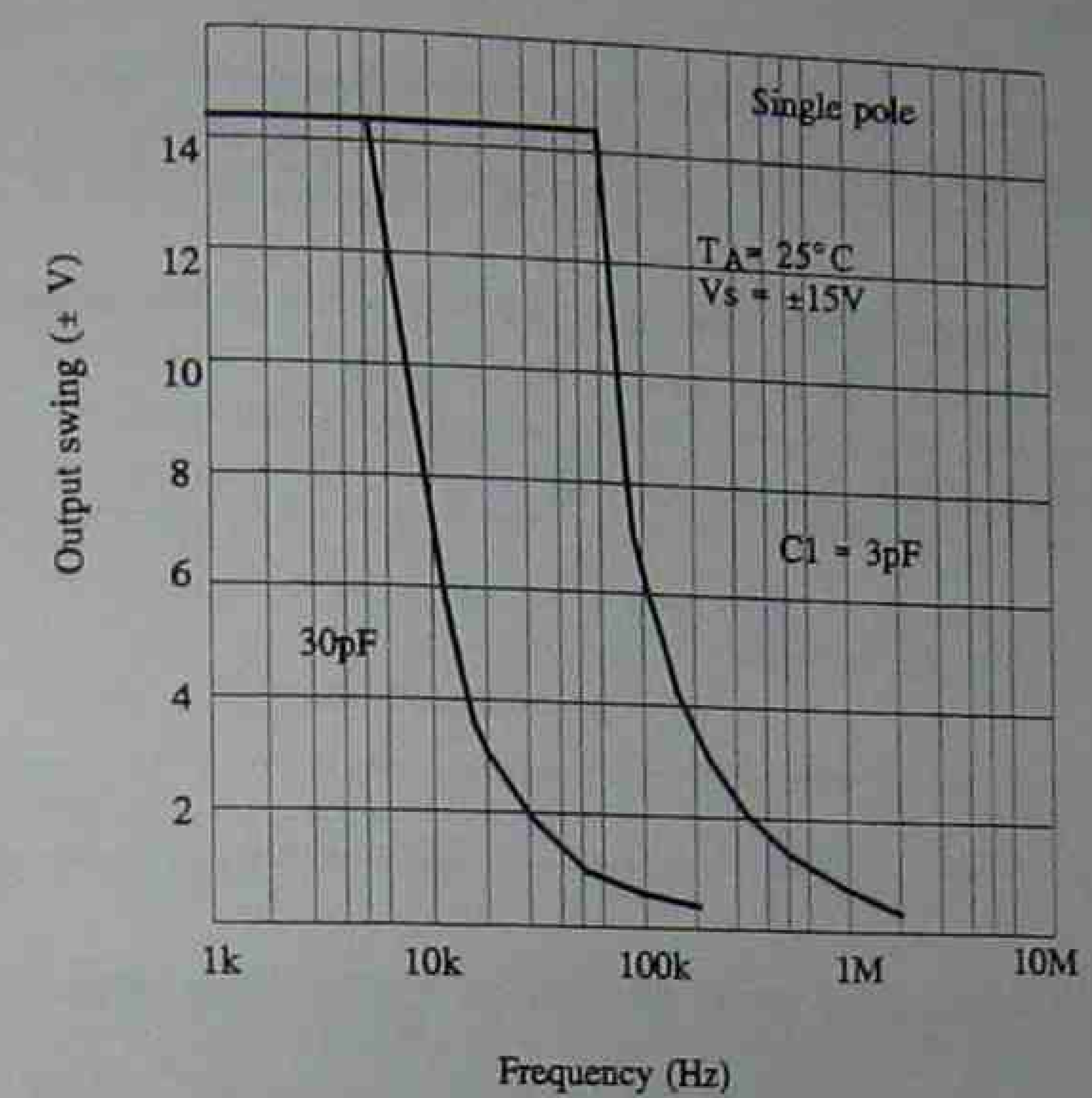
These questions will help you revise what you have learnt in Section 3.

1. An operational amplifier with a slew rate of 0.5 V/μs is connected as a voltage follower and a square wave input at 62.5 kHz, 6V peak to peak is applied. Sketch the output waveform, carefully marking important time and voltage values.

2. An amplifier has a sinewave input at 100 kHz and a triangular wave output as shown below. What is the slew rate of the amplifier ?



3. The output voltage swing graph of an externally compensated operational amplifier is shown below.



(a) What is the FPBW and slew rate for C_1 = 30pF?

(b) What is the FPBW and slew rate for C_1 = 3 pF?

(c) If C_1 = 3 pF is used, determine from the graph the maximum undistorted output amplitude for a sinewave input at 300 kHz.

(d) Using equation 2 and the slew rate calculated in part (b), verify your answer to part (c) by calculation.

____

____

4. Explain the significance of the terms 'slew rate' and 'full power bandwidth'.

____

____

____

____

5. State any **two** methods which can improve the slew rate performance of an amplifier circuit.

- ______________________________

  ____

- ______________________________

  ____

6. Check the data sheets of several op amps and name one type which has a slew rate of at least 50 V/μs.

____

____

Skill practice 4
Slew rate

Suggested duration
1 hour 30 minutes

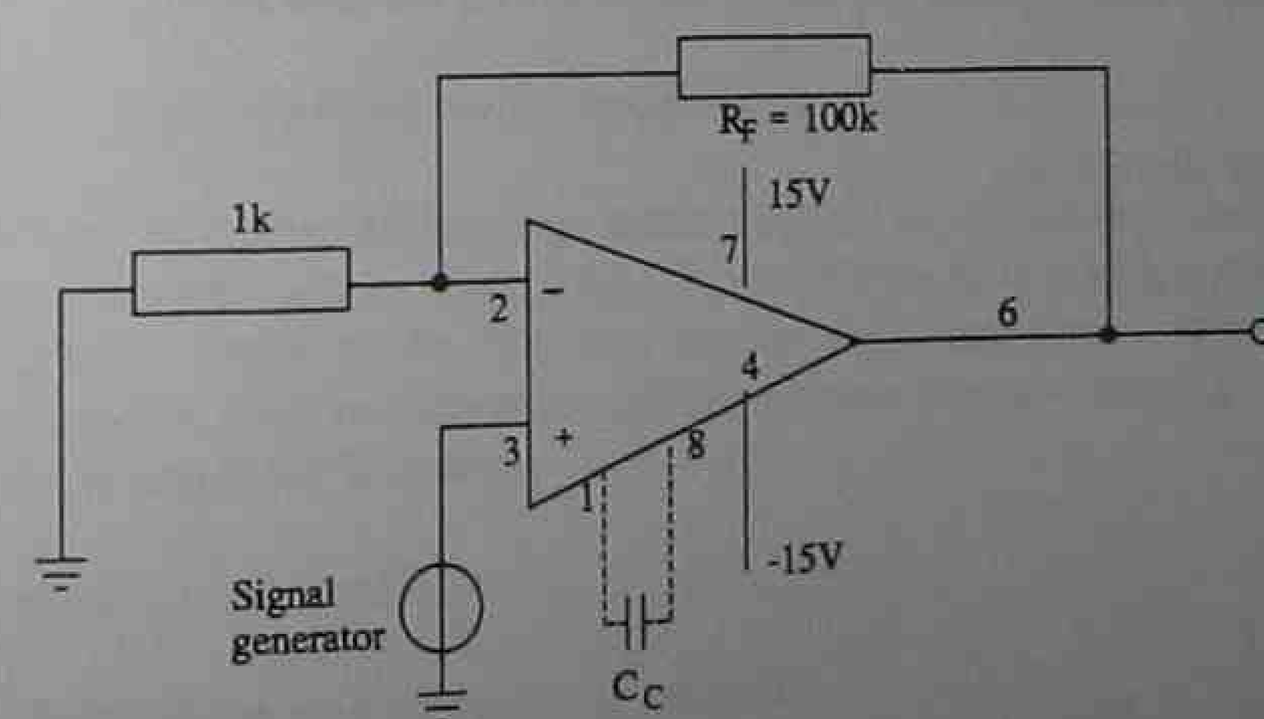
Tasks
- To measure the slew rate of an op amp.
- To observe the effect of changing the compensating capacitance on slew rate in an externally compensated op amp.

Equipment
- Either one op amp type LM301 and a 30 pF capacitor OR one op amp type 741. (741 is not suitable for Step 6.)
- General purpose 15 MHz oscilloscope (one with time-base x5 or x10 switch preferable)
- AC signal source (e.g.function generator)
- Dual DC power supply (fixed ±15V supplies can be used)
- Connectors, breadboard, sockets etc.

Procedure
Step 1 Setup

- Set the power supplies to ± 15V.
- Set the function generator to square wave, about 5 kHz, about 2 V peak to peak.
- Signal frequency and voltage are not critical.

- Connect the circuit shown below. (Do not forget to connect the power supply common to circuit ground.)
- If you are using 301 op amp, connect a 30 pF capacitor across pins 1 to 8. This is the compensating capacitor C_c mentioned in the lesson. If you are using the 741 op amp, this capacitor is not necessary.



Step 2 Output waveform

- Sketch the output waveform.

Step 3 Measurements

- Measure the rise in voltage and rise time. (If your CRO has a time-base x5 or x10 switch, rise time measurement gets more accurate - ask your teacher to show you how).

Rise in voltage =

Rise time =

Step 4 Slew rate calculation

- Calculate the slew rate (equation 1) and compare it with the data sheet specification.

Slew rate (experimental) =

Data sheet specification :

Step 5 Effect of changing the compensation capacitor

- If you are using the 741 op amp, skip this step. If you are using the 301 op amp, remove the 30 pF capacitor and R_F and repeat steps 2, 3 and 4 above.
- Sketch of output waveform:

Rise in voltage =

Rise time =

Slew rate (experimental) =

Step 6 Full power bandwidth measurement

If you are using the 301 op amp, reinsert R_F and the 30 pF capacitor. Change the signal generator to sine wave. Adjust input voltage until output just begins to clip. Observe the output on the CRO and vary the frequency from 1 kHz upwards until it starts to show slew rate distortion (flattening near the zero crossing - see Figure 8 in notes). Measure the following.

Output sine wave amplitude :

Maximum undistorted frequency of operation :

Theory calculation of f_{max} (equation 2) =

Percentage error in measurement :

Discussion questions

1. How does your measured slew rate compare with the data sheet value ? What is the percentage error?

____

____

____

____

2. If you did Step 5 above, what is the effect of reducing the compensation capacitor on the slew rate?

____

____

____

____

Section 4: AC Noise

SUGGESTED DURATION	PREAMBLE
9 hrs 30 mins	To enable you to analyse the noise performance of op amp circuits and recognise methods for improving it.

Objectives

At the end of this section you should be able to:

- ☐ understand the noise model of op amps and in particular:
  - state the meaning of 'noise' as applied to electronic circuits and various common sources of noise
  - state the meaning of the terms 'noise spectrum', 'noise density', 'white noise' and 'pink noise'
  - interpret the input noise model for an amplifier and state the meaning of 'total input equivalent noise voltage'
- ☐ calculate the source noise resistance of common amplifier circuits (including those studied in Section 1, as well as summing amplifier and differential amplifier configurations)
- ☐ use the broadband noise graph of an operational amplifier to estimate the total equivalent input noise voltage
- ☐ use the noise rms addition formula to find the total equivalent noise rms voltage or current given several independent noise voltages or currents
- ☐ calculate the noise rms voltage or current over a given bandwidth, given the noise density specification (either as volts squared per Hz or volts per square root Hz, and similarly for current)
- ☐ write the formula for the total equivalent input noise voltage and calculate it
- ☐ calculate the noise gain of common amplifier circuits and the output noise
- ☐ define 'signal to noise ratio' (SNR); calculate input SNR and output SNR given a circuit, the amplifier's noise specifications and the input signal
- ☐ calculate the effective noise bandwidth when bandlimiting is done by a first order (RC) or by a high order (order ≥ 5) low pass filter
- ☐ recognise common methods to minimise external and internal noise in amplifiers

- ☐ measure/estimate a noise voltage using a
  - true RMS voltmeter
  - digital multimeter
  - CRO
- ☐ measure the input noise voltage of an amplifier for a given bandwidth and source resistance.

References

The following references deal with topics in this section.

1. Jacob (1993), pp 199-203
2. Rutkowski (1994), pp 106-108
3. Gayakwad (1993), pp 197-199
4. Coughlin & Driscoll (1993), pp 264-267

Noise

'Noise' is any unwanted voltage or current at the output of the system. Commonly, noise is thought of as added to the signal and unrelated to it.

Sources of noise in amplifiers and noise model

External noise

The output noise in amplifiers can be due to external or internal sources. External noise can be picked up by incoming signal leads (or communication channels) - for example, the 50 Hz hum from AC power lines, cosmic noise, broadcast signals, DC power supply variations, switches, fluorescent lamps, motors. External noise can sometimes be minimized by careful circuit construction, shielding and earthing. Though external noise is important, we shall not consider it further in this section.

Internal noise

The amplifier also generates noise internally. There are four main sources of internal noise.

1. Noise voltage caused by external resistors (e_R).

   You have learnt Ohm's Law, which says that the current I in a resistor is equal to the voltage difference across the resistor V divided by the the resistance R. However, the electron flow which makes up this current is not quite smooth or uniform. At any instant, a few more or a few less electrons than average reach the positive voltage terminal. This random variation in electron flow shows up as a noise voltage.

   Resistive noise is also called 'thermal' or 'Johnson' noise.

   e_R is the total noise voltage generated by the external resistors which make up the amplifier (R_1, R_F, R_c etc). We shall soon present and use the formula for calculating e_R.

2. Similarly, the resistances inside the amplifier generate noise voltages.

3. The amplifier generates noise current called 'Schottky' or 'shot' or 'recombination' noise. This is caused by the non-uniform, random rate at which the positive and negative charge carriers in the semiconductor devices recombine.

4. The amplifier also generates noise current called 'flicker' noise. This noise is also called '1/f' noise because its power increases with decreasing frequency.

Noise model for operational amplifiers

Since the noise components generated by the operational amplifier are impossible or difficult to calculate, it is common practice to 'lump' or aggregate the internal noise into one single effective noise voltage source and one single effective noise current source, and bring them out to the input. This is very similar to the approach taken with the DC input offset voltage. This gives the noise model for the amplifier as shown below in Figure 1.

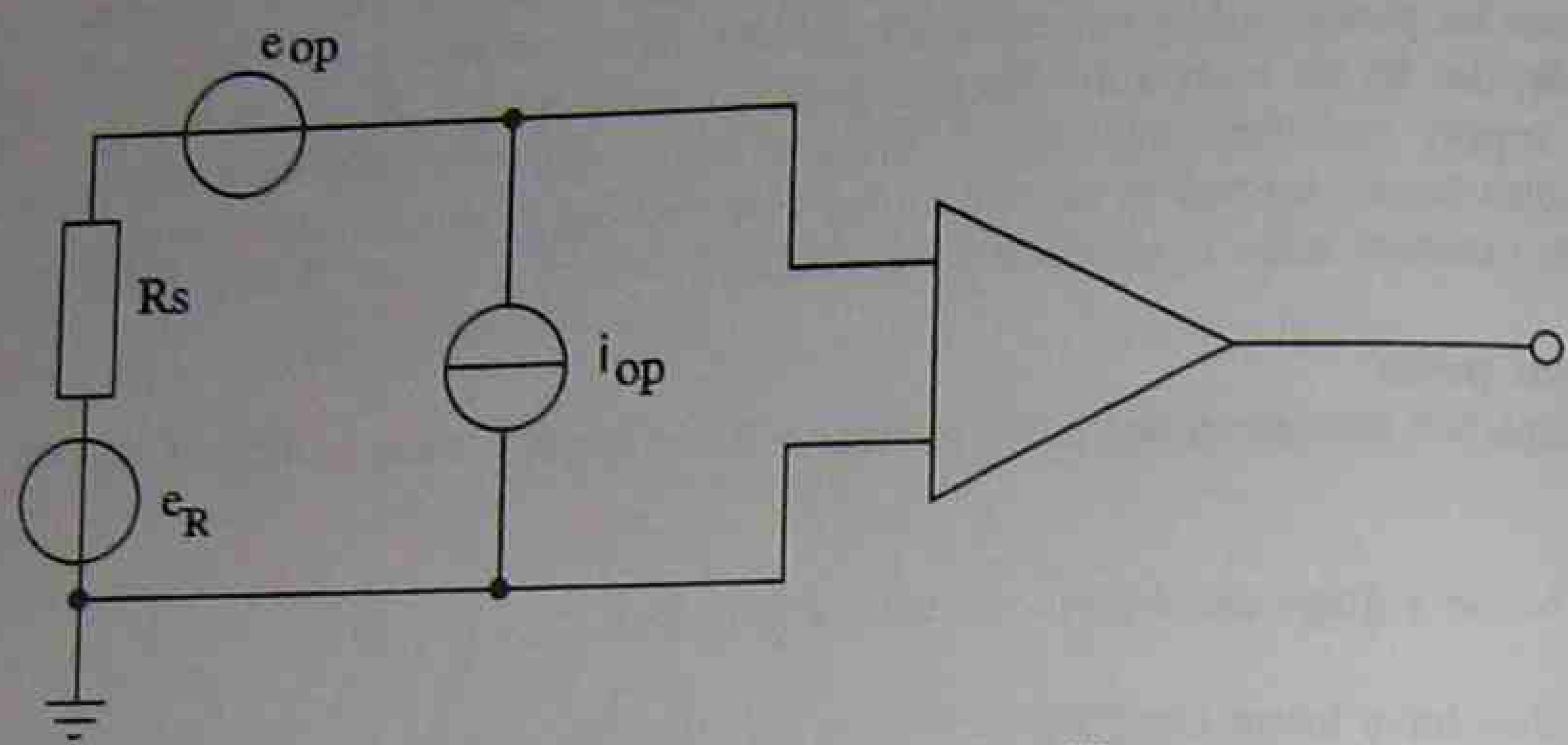


Fig. 1 Noise model for operational amplifiers

R_s = source noise resistance (equivalent to all the externally connected resistors in the circuit)

e_R = thermal noise voltage source due to R_s

e_{op} = equivalent input noise voltage due to the op amp

i_{op} = equivalent input noise current due to the op amp

Note that:

- there are only three sources in the circuit above (not four)
- we keep e_R and e_{op} separate, even though they are in series, because the former is affected by our choice of components, while the latter is a property of the op amp
- e_{op} and i_{op} do not really exist at the input or anywhere else. They only represent the effect of the actual noise inside the amplifier.

Next we study how to combine the effects of the three sources given above into a single equivalent input noise voltage source. Then we study the actual mechanics of calculating the equivalent input noise voltage using formulas and/or the data sheets of the op amp.

Description and calculation of noise

Since noise is usually random and unpredictable, we cannot write any equation for noise voltage or current, nor sketch it with reasonable accuracy. However, it is a fact that the average *power* in a load due to any realistic noise source is fixed. Since power (from any kind of voltage or current source) is proportional to (rms voltage)^2 or (rms current)^2, we can say that the rms voltage or rms current of a noise source (working into a given load) has a fixed and definite value.

Noise voltage or current is always expressed as a rms value.
Note that you cannot use the conversion factor of 1.414 to convert noise rms voltage to noise peak voltage. This factor applies only to sine waves and has no meaning for noise.

Though a single rms value is a convenient description of noise, it does not tell the whole story. Since noise varies randomly, it can be usually expected to have a large spread of frequency components. The *noise power density spectrum* is a graph showing the distribution of the total noise power over frequencies, or the power per Hz of bandwidth as a function of the centre frequency where the power is measured.

The shape of the noise power density spectrum depends on the physical origin of the noise. Some common shapes are:

- constant with frequency (called white noise)
- inversely proportional to frequency (called pink noise) and
- lines at specific frequencies.

Thermal noise and shot noise are white. Flicker noise is pink. The hum picked up from power lines has a line spectrum, since all their power is concentrated at 50 Hz and its harmonics.

An important point to note about noise power (and so, the noise rms voltage) is that it depends strongly on the circuit bandwidth, which is under the control of the circuit designer.

For white noise (which is the most common kind of noise), the noise power is proportional to the bandwidth. Therefore, data sheets often specify the noise density rms volts squared (or noise rms current squared) per Hz and leave it to you to calculate the total noise volts squared over your circuit bandwidth.

Other data sheets give noise density rms volts per square root Hz (or noise rms current per square root Hz). Of course, the square root of Hz is not a physically meaningful unit. It only serves to remind you that the noise rms voltage or current is proportional to the square root of the bandwidth.

Total noise volts over a bandwidth

$$= \sqrt{(\textit{noise density volts}^2/Hz) * \textit{bandwidth}} \qquad \text{Equation 1}$$

or

$$= (\textit{noise density volts}/\sqrt{Hz}) * \sqrt{\textit{bandwidth}} \qquad \text{Equation 2}$$

The same rule works for noise current.

Note : This rule applies only to white noise. However, white noise is the most common kind in wideband amplifiers. The other common kind, pink noise, is important only at low frequencies, in which case noise is probably not a major problem anyway.

Some examples showing the calculation of noise voltage and current follow.

Example 1
If an op amp is stated to have a noise voltage density of 15 nV/$\sqrt{Hz}$, its rms noise voltage over a bandwidth of 30 kHz
= 15 nV/$\sqrt{Hz}$ * $\sqrt{30kHz}$ = 2.6 μV rms.

Example 2
The noise current density of a 741 op amp is specified as 3 * 10^{-25} A^2/Hz. Its rms noise current over a 25 kHz bandwidth = $\sqrt{3E-25A^2/Hz * 25kHz}$ = 87 pA rms.

Example 3
The noise voltage of an amplifier is measured to be 6.5μV over a 40 kHz bandwidth. If the bandwidth is changed to 15 kHz, the noise voltage is estimated as
$6.5\mu V * \sqrt{15kHz/40kHz} = 4\mu V\ rms.$

Addition of noise voltages and currents

Normally the noise in any system results from the addition of noises from many different sources. The noises are random and without any relationship to each other. They are uncorrelated or non-coherent. Uncorrelated signal *voltages or currents* cannot be simply added to get the total. However using advanced mathematics, it can be shown that uncorrelated noise *powers* can be added. This means that noise volts squared (or noise currents squared) can also be added to get the total. After adding the squares, we can take the square root to get the rms value of the noise voltage (or current).

$$e_{total}^2 = e_1^2 + e_2^2 + e_3^2 + \ldots\ldots \qquad \text{Equation 3}$$

or

$$e_{total} = \sqrt{e_1^2 + e_2^2 + e_3^2 + \ldots\ldots}$$

If an amplifier has external noise voltage of 4.5μV and internal equivalent noise of 7μV at its input, the total equivalent noise at the input e_{total}
$= \sqrt{(4.5\mu V)^2 + (7\mu V)^2} = 11.5\mu V$

Note that the answer is *not* (7 + 4.5) μV = 11.5μV.

Ohm's Law is unchanged for noise. The noise voltage v_n caused by a noise current i_n flowing through a resistance R is given by $v_n = i_n * R$.

Total equivalent input noise of an amplifier

Refer to the noise model of the amplifier in Fig. 1. The total equivalent noise at the input is total of the effects due to (i) the thermal noise e_R of the source resistance R_s, (ii) the internal noise voltage of the op amp e_{op}, and (iii) the noise voltage caused by the internal noise current i_{op} flowing through R_s. Using the rules for adding noise voltages and Ohm's Law,

The total equivalent input noise voltage of an amplifier =

$$e_n = \sqrt{e_R^2 + e_{op}^2 + (i_{op}^2 * R_s^2)}\ V(rms) \qquad \text{Equation 5}$$

Example 4: Using the noise addition formula
In an operational amplifier circuit, the source noise resistance = 30 kΩ, the thermal noise due to the source resistance = 2.8μV, the internal noise current of the op amp = 60 pA and the internal noise voltage of the op amp = 4.1μV.

(a) What is the total equivalent input noise voltage e_n ?

(b) What will be the new value of e_n if the bandwidth is tripled ?

Solutions

(a) $e_n = \sqrt{(2.8\mu V)^2 + (4.1\mu V)^2 + (60pA * 30k\Omega)^2}$
$= 5.3\mu V$ (rms)

(b) Since $e_n \propto \sqrt{BW}$, new $e_n = 5.3 * \sqrt{3}\ \mu V$
$= 9.2\ \mu V rms$

Effective noise bandwidth

We have seen that the total noise voltage depends strongly on the circuit bandwidth. However, the bandwidth is just a convenient number, i.e. the frequency where the output has a relative attenuation of 3 dB, or 0.707 times the maximum output. Signals or noise at higher frequencies will also pass through, though at greater attenuation. This means that the total noise actually transmitted by the amplifier is more than what is calculated on the basis of the bandwidth. The situation is shown in Figures 2 and 3 below.

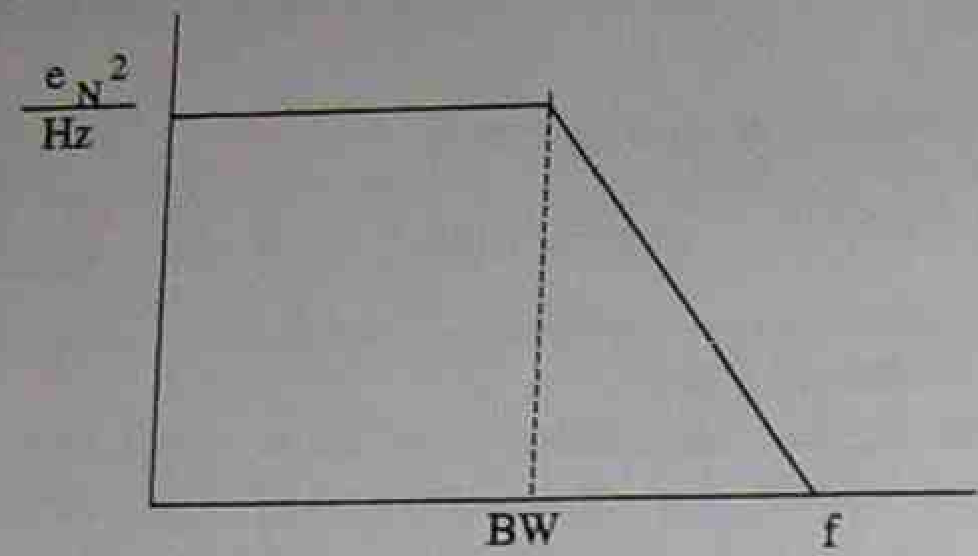


Fig. 2 Practical Bandlimiting Filter response

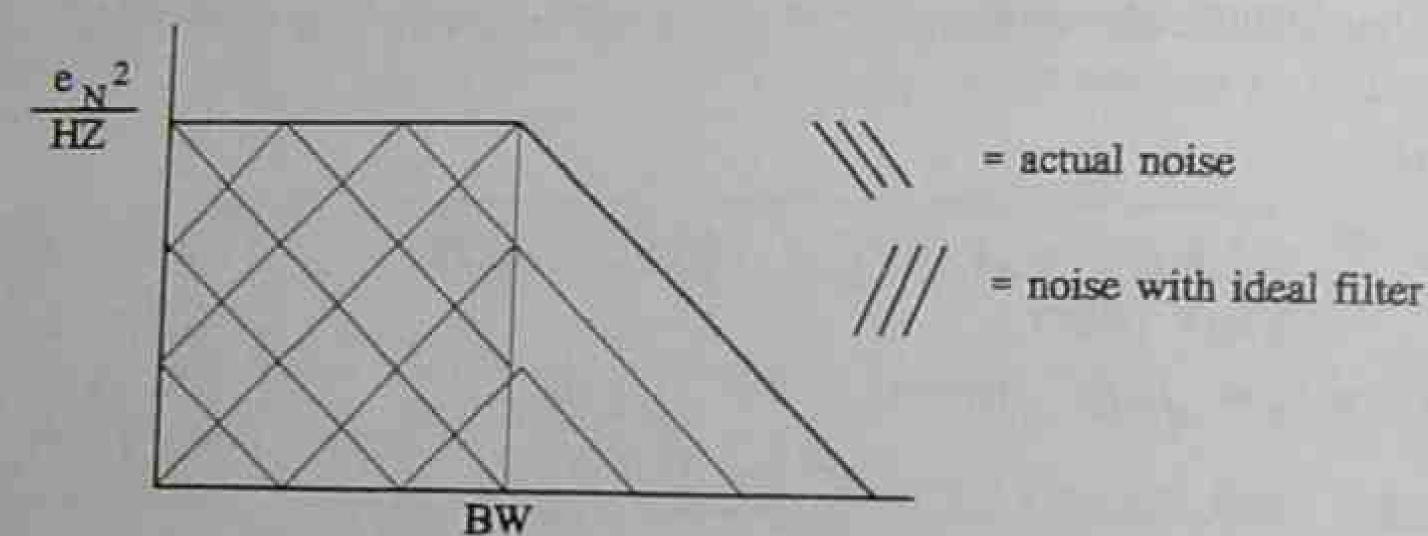


Fig. 3 Comparison of noise allowed in by ideal and practical bandlimiting filters

The extra noise due to the 'tail' of the filter response depends on its rate of roll-off. If the rate of roll-off is 20 dB/decade, then advanced mathematics shows that

Effective noise bandwidth = $\pi/2$ * (3dB BW) — Equation 6

Since noise voltage is proportional to the square root of the bandwidth,

$$\text{noise increase factor for bandwidth correction} = \sqrt{\frac{\pi}{2}}$$

Equation 7

Note that these formulae are valid only for first-order (or single pole, or simple RC) low pass filtered white noise. This is the most common situation. The filtering may be done by external components, or perhaps by the internal capacitances in the amplifier.

For high order filters (order ≥ 5) no correction is necessary because the rate of rolloff is steep and close to ideal.

Example 5 : Effective noise bandwidth

An amplifier has input noise of 15nV/√Hz. It is bandlimited by an RC filter with cutoff frequency of 20 kHz.

(a) What is the effective noise bandwidth?

(b) What is the actual input noise voltage?

Solutions

(a) Effective noise bandwidth = 20 kHz * $\pi/2$ = 31.4 kHz

(b) actual input noise voltage = 15 nV/√Hz * √31.4kHz
= 2.66 μV (rms)

Calculation of input noise resistance

Looking at the total equivalent input noise of an amplifier (equation 5), we see that we need the value of the input noise resistance of the amplifier, R_s.

R_s is the effective resistance (called the Thevenin resistance) between the +input and -input with all sources removed. This means that all voltage sources are short circuited, all current sources are open circuited and the output earthed. All the common negative feedback amplifiers can be simplified to the following form (Figure 4).

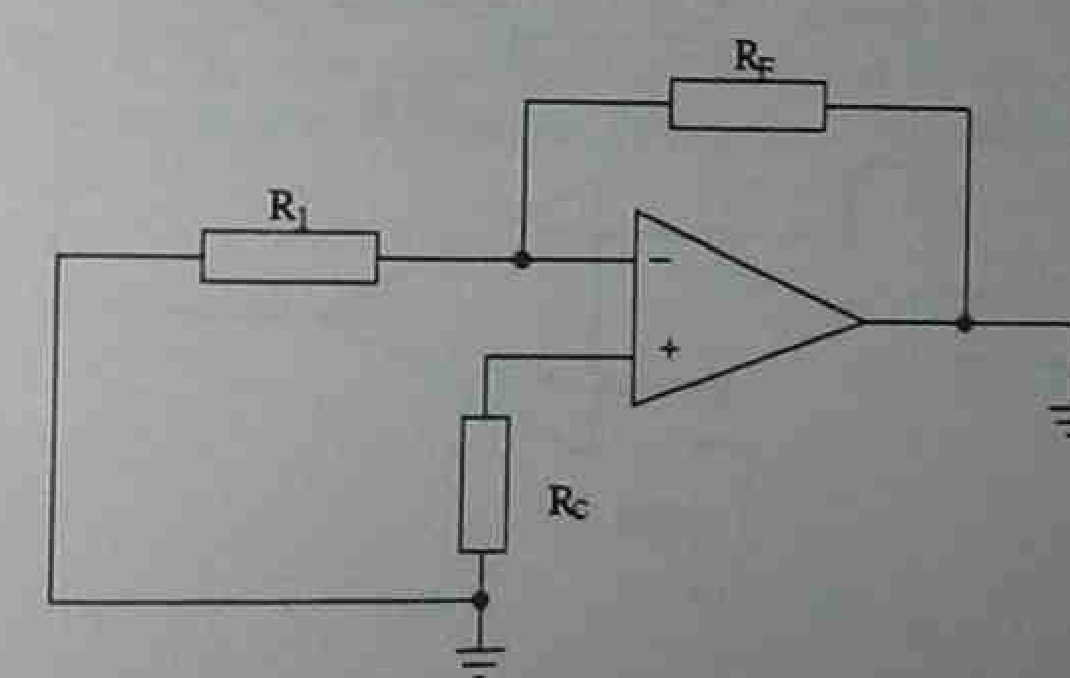


Fig. 4 Negative feedback amplifier

$\therefore R_s = R_c + (R_F \parallel R_1)$ — Equation 8

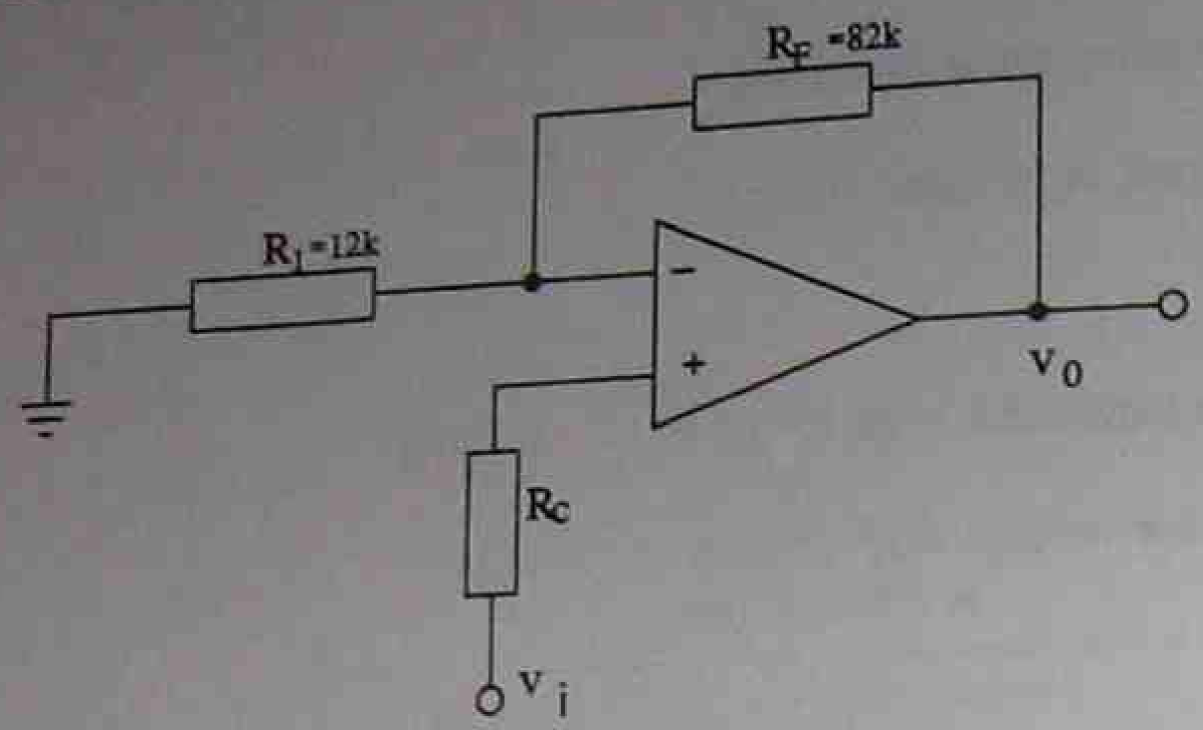
As before, R_c and R_1 are the effective resistances from the +input and -input to ground respectively, after simplifying any series and parallel combinations.(To be exact, the answer in equation 8 must be paralleled with the input resistance of the op amp. Practically, the input resistance is very large and does not affect the result.)

Example 6 : Source noise resistance

For the following circuit, calculate

(a) the value of R_c for bias current compensation

(b) the source noise resistance, R_s, for R_c = 10 kΩ.



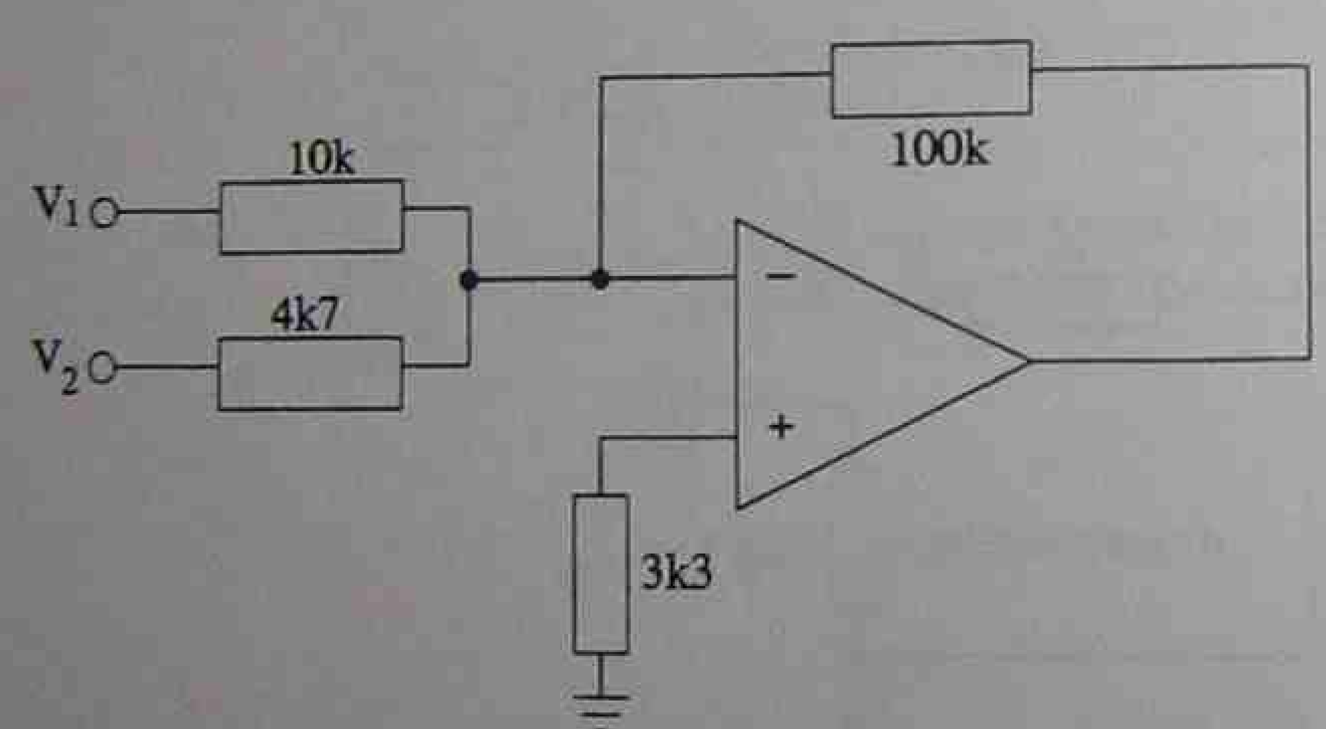
Solutions

(a) R_c = 12k ∥ 82k = 10.4 kΩ

(b) R_s = 10 k + (12k ∥ 82k) = 20.4 kΩ

Example 7 : Source noise resistance

For the following circuit, calculate the source noise resistance.



Solution

To calculate R_s, ground all inputs and the output. This makes 10k and 4k7 in parallel.

R_1 = 10k ∥ 4k7 = 3.2 k

∴ R_s = 3k3 + (3.2k ∥ 100k) = 6.4 kΩ

Determination of total equivalent input noise voltage using the broadband noise graph

The data sheets of many op amps give a 'broadband noise graph' which simplifies the calculation of the total equivalent input noise voltage (e_n.) This graph gives a set of curves giving e_n as a function of R_s for different bandwidths. So if we calculate R_s and know the effective bandwidth, we can read off e_n without further calculation. A typical broadband noise graph as found in the Fairchild linear data book appears below (Figure 5). This graph is for the type 741 op amp.

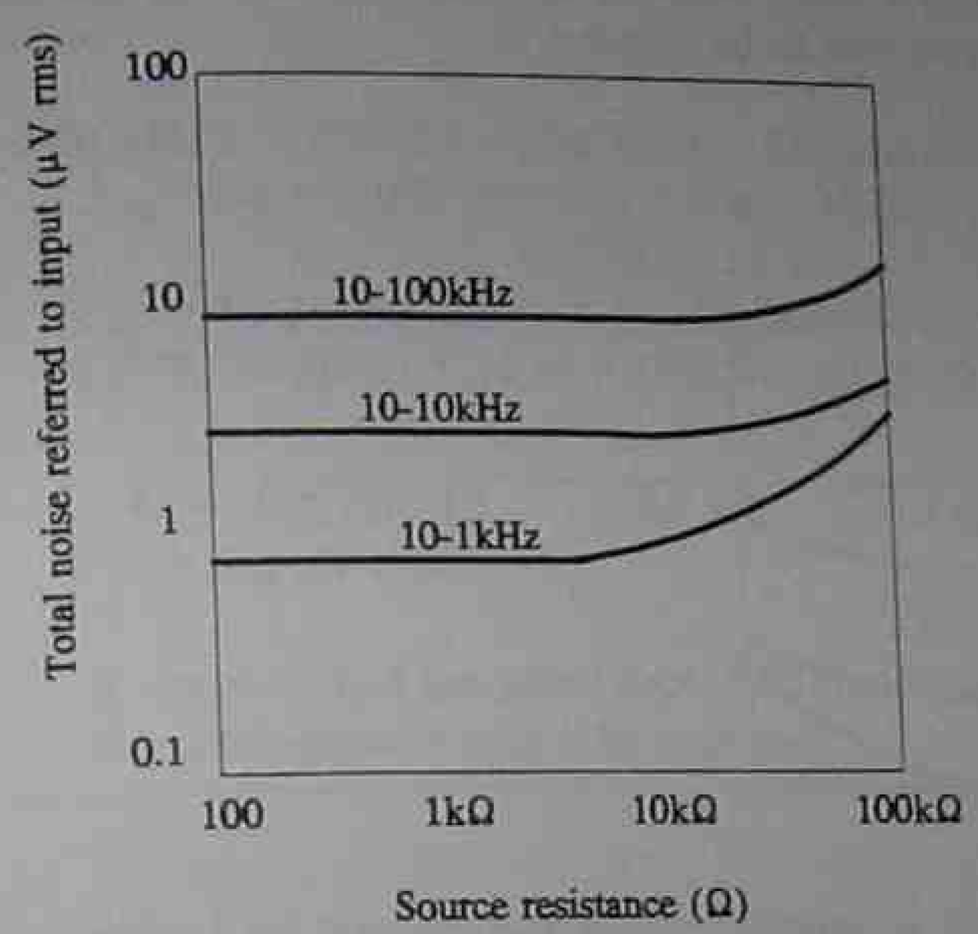


Fig. 5 : Broadband noise graph for 741 op amp

Some care is needed to use this graph. The X and Y axis are log scale and they go *1,2,3,4,5,10* (6,7,8 and 9 are not shown). The bandwidths are actual bandwidths, not ranges. For example, the curve for '10-100 kHz' means 'for cutoff frequencies 10 Hz and 100 kHz'. If the actual circuit has a bandwidth of 30 kHz, for example, you have to estimate the value of e_n between the graphs for 10 kHz and 100 kHz.

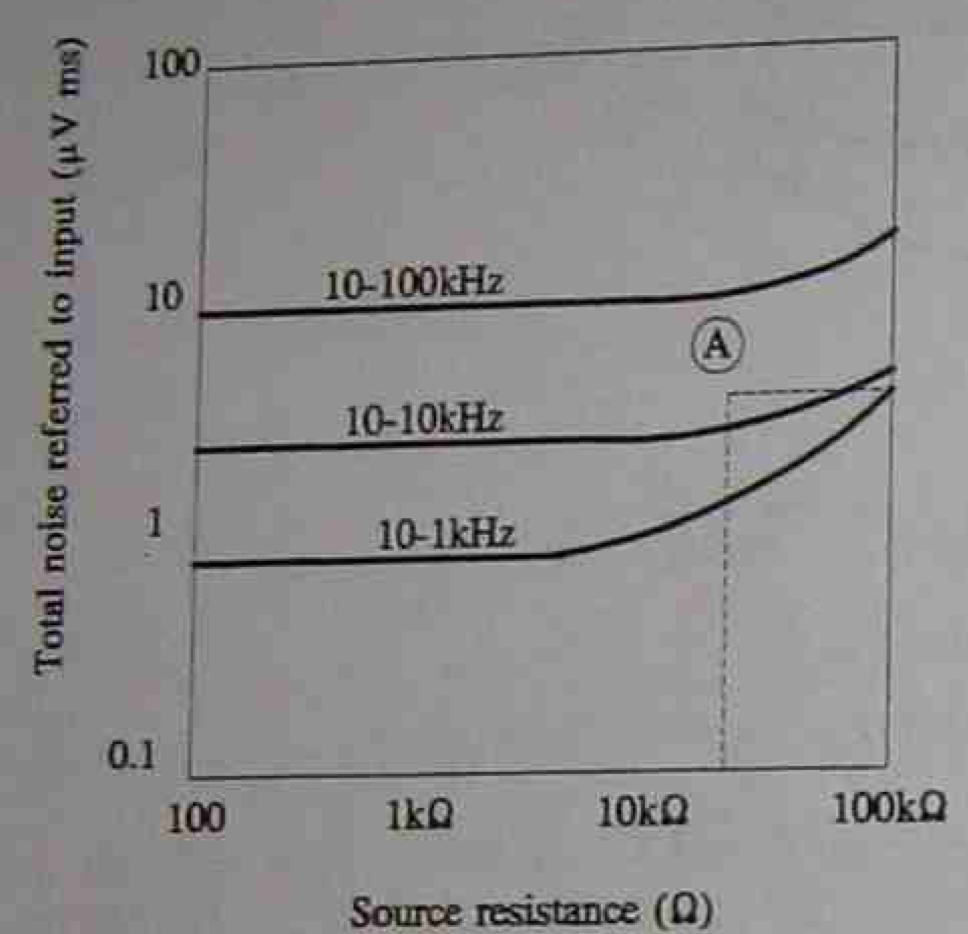
Example 8: Using the broadband noise graph

An amplifier using a 741 op amp has source noise resistance of 20 kΩ. It is bandlimited to 20 kHz using a RC filter. Use the broadband noise graph to obtain the total equivalent input noise voltage.

Solution

Effective BW = 20 kHz * π/2 = 31.4 kHz. Reading the graph for R_s = 20 kΩ and BW = 31.4 kHz, the answer is estimated to be at point A.

$\therefore e_n = 5\ \mu V$ rms.



Calculation of e_n using equation 5

If the broadband noise graph is not available, we have to calculate e_n using equation 5. We have already seen how to calculate R_s, e_{op} and i_{op} if we know the BW and the noise V/√Hz and A/√Hz. The resistive noise e_R is calculated from the formula

$$e_R = \sqrt{4 * k\ T\ R_s\ * B} \qquad \text{Equation 9}$$

k = Boltzmann's constant = 1.38E-23 JK^{-1}
T = absolute temperature K = °C + 273
R_s = resistance
B = effective noise bandwidth

Now equation 5 can be used to calculate e_n.

Example 9 : Calculation of total equivalent input noise

An amplifier using type 741 op amp has source noise resistance of 20 kΩ. The effective circuit bandwidth is 31.4 kHz. From the data sheets, the 741 op amp has squared input noise voltage of 4E-16 V^2/Hz and squared input noise current of 3E-25 A^2/Hz. The temperature is 25°C.

Calculate the total equivalent input noise voltage (e_n).

Solution

R_s = 20 kΩ; T = 273+25 = 298°K; B = 31.4kHz

$\therefore e_R^2$ = 4 * 1.38E-23 * 298 * 20k * 31.4k
= 10.33E-12 V^2

e_{op}^2 = 4E-16 * 31.4k = 12.56E-12 V^2

$i_{op}^2 * R_s^2$ = 3E-25 * 31.4k * $(20k)^2$ = 3.77E-12 V^2

$$\therefore e_n = \sqrt{10.33E-12 + 12.56E-12 + 3.77E-12}\ V$$
= 5.2 μV rms.

Example 4 had the same data. The broadband noise graph gave nearly the same answer with a lot less work.

Noise gain and the output noise voltage

The output noise is calculated by the formula:

output noise = total equivalent input noise * noise gain

We have discussed the calculation of input noise e_n by two methods : using the broadband noise graph and using equation 5. The noise gain is just the *non-inverting* voltage gain of the circuit, whether the circuit is inverting, non-inverting or anything else. This is similar to offset voltage calculation.

Example 10: Output noise voltage calculation

The circuit of example 4 has an effective bandwidth of 100 kHz and uses a 741 op amp at 25°C.

(a) Use the broadband noise graph in Example 5 to calculate the total equivalent input noise voltage.

(b) Calculate the noise gain.

(c) Calculate the output noise voltage.

Solutions

(a) R_s = 6.4k (see example 4). Using the graph in example 5, for 100 kHz, we estimate e_n = 6.5μV rms.

(b) noise voltage gain = 1 + 100k/3.2k = 32.25

(c) output noise voltage = 6.5μV * 32.25 = 210 μV rms.

Signal to noise ratio (SNR): output SNR and input SNR

We see that, even with cheap general, purpose op amps, the output noise is quite small — some hundreds of microvolts. So why worry about it ?

In many circuits, especially those used in communications and measurements with transducers, the signal is also quite small. What really matters is how the signal voltage compares with the noise voltage. The simplest way to describe this is to use the signal to noise ratio (SNR) :

SNR = signal rms voltage / noise rms voltage   Equation 10

$(SNR)_{dB} = 20 \log (SNR_{ratio})$ in dB notation   Equation 11

Note that rms voltages have to be used. Noise voltage is already rms, so signal voltage also has to be made rms.

Equations 10 and 11 work in the normal way for output SNR. For input SNR, the noise should include *only the resistive noise,* not the total equivalent input noise e_n. This is because e_n includes e_{op} and i_{op} which are really added on after the input, so it is not correct to include them in the input SNR.

Input SNR = input signal rms / e_R Equation 12

The output SNR is always less than the input SNR, for two reasons : the former includes op amp noise and the latter does not; and the noise gain ≥ signal gain.

Example 11 : Input and output SNR calculations

Continuing examples 7 and 10, suppose the input signal is sine wave of 20 mV p-p and connected to v_2. (Ignore v_1)

(a) Calculate the output SNR

(b) Calculate the input SNR

Solutions

(a) output signal = -100k/4.7k * 20 mV p-p = 425.5 mV p-p
= 425.5 /(2√2) = 150 mV rms
output noise = 210 μV rms (from example 11)
∴ output SNR = 150 mV/210 μV = 716.4 = 57.1 dB

(b) $e_R = \sqrt{4*1.38E{-}23*298*6.4k\Omega*100kHz} = 3.24\mu V$

∴Input SNR = (20/2√2mV)/3.24μV = 2182.4 = 66.8 dB

Measurement of noise

Since noise voltage is usually expressed as rms, we can measure noise directly with a true-RMS reading meter (such as an AC millivoltmeter) and use the reading directly.

If you use a DVM, you have to multiply the reading by 1.13. This is because the DVM is internally calibrated to read only sinewave rms voltages. If you doubt this, try measuring a square wave rms voltage using a DVM and an ACMVM. You will see different readings and the latter is the correct value.

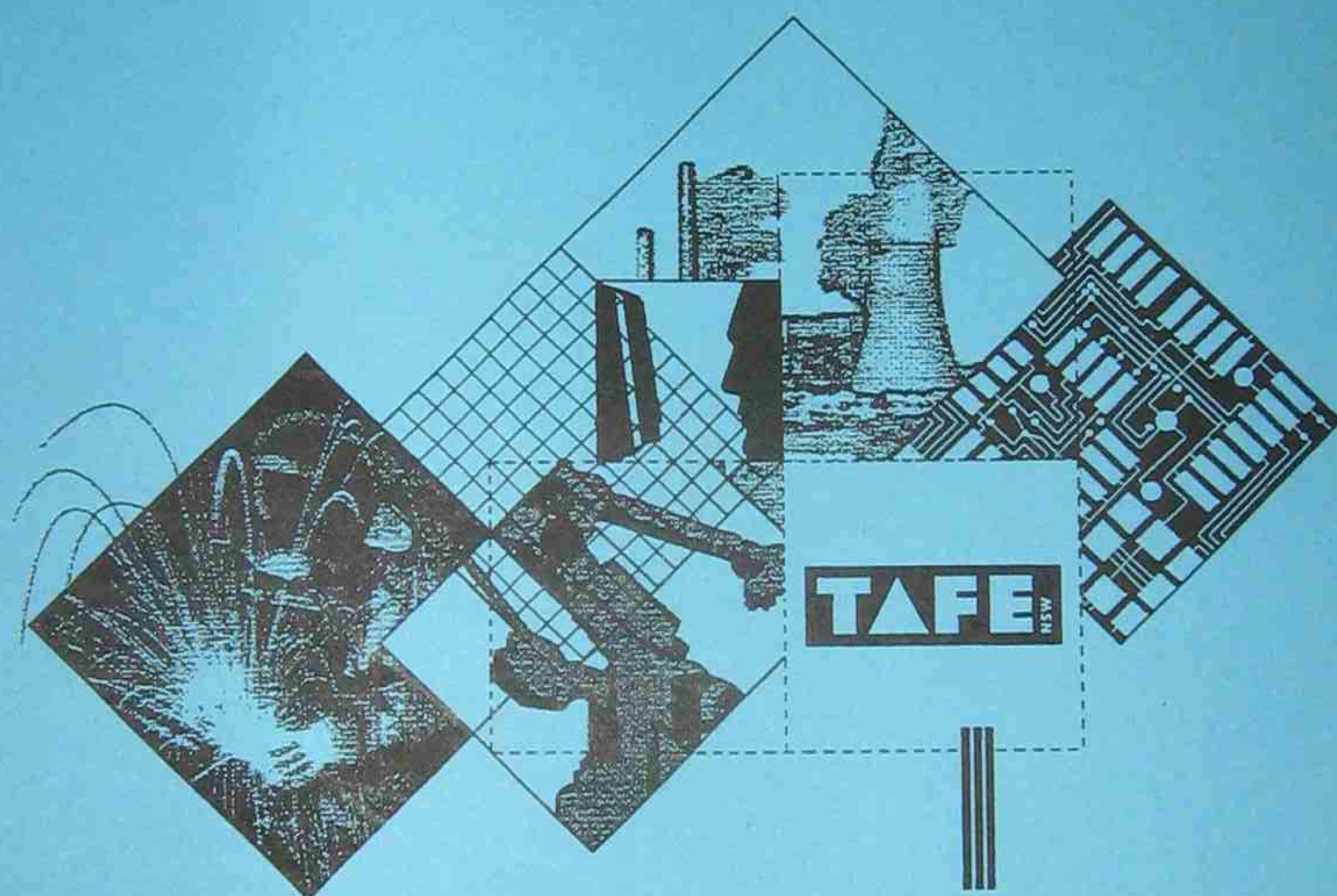
Using a CRO, display the signal with a slow sweep (you can use AC or DC signal coupling), observe the p-p noise voltage and then estimate :

the noise rms voltage = (noise p-p voltage)/6

Note that it is incorrect to divide noise p-p voltage by 2√2. The factor 2√2 works only for sinewaves.

The conversion factors 1.13 and 6 mentioned above come from advanced mathematics beyond the scope of this module.

NSW Module Resource Manual for the TAFE Engineering Technician and Engineering Associate National Curriculum



Electrical/Electronic Stream

Analogue Electronics 1

Student Workbook

National Module Code: EA100

NSW Module Number: 7761A

First published in January 1996 by
TAFE Engineering Services
Industry Training Division
PO Box 218 Bankstown NSW 2200

National Library of Australia pub'n no:
ISBN 0 7310 6191 8

ANALOGUE ELECTRONICS 1
EA100/7761A

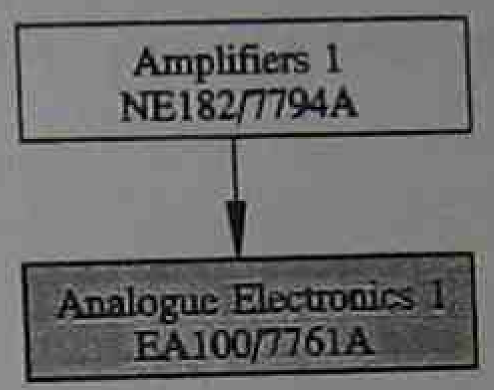
CONTENTS

Notes

About this module

This module is a part of the electrical/electronic stream in the TAFE National Engineering Technician and Engineering Associate Curriculum.

The prerequisite for this module is NE182/7794A Amplifiers 1. The diagram below shows the place of this module Analogue Electronics 1 in the telecommunications engineering stream.



This module extends the knowledge in the prerequisite modules by studying the performance limitations and specifications of operational amplifiers. On completion of this module you will have developed an understanding of the design, analysis and selection of workable substitutions in circuits using modern operational amplifiers and analogue integrated circuits.

This workbook is not a substitute for the recommended text and references, but it will help you by providing technical information in this subject area.

The review questions and skill practice exercises will help you find out how well you have understood the topic and help you prepare for the competency tests.

The module is divided into five sections. They are designed to be worked through progressively. The sections are:

- Basic op amp circuits
- DC non-idealities
- Slew rate
- AC noise
- Frequency compensation

What you will need for this module

Recommended text

Jacob, J. Michael. *Applications and Design with Analog Integrated Circuits* (second edition). Regents/Prentice-Hall, 1993.

Additional references

Coughlin, Robert F. and Frederick F. Driscoll. *Operational Amplifiers and Linear Integrated Circuits (forth edition).* Prentice-Hall, 1991.

Faulkenberry, Luces M. *Introduction to Operational Amplifiers with Linear Integrated Circuit Applications.* Wiley, 1982.

Gayakwad, Ramkant A. *Op-Amps and Linear Integrated Circuits (third edition).* Prentice-Hall, 1993.

Rutkowski, George B. *Integrated Circuit Operational Amplifiers (second edition).* Prentice-Hall, 1984.

Other reference material

Linear data books and application notes published by various IC manufacturers, e.g. National, Fairchild, Motorola, Analog Devices.

Student purchases

The following components and materials are required for the skill practice exercises and practical tests for this module.

- Type 741 op amp - 1 required; at least one spare recommended (741, 741C, 741A, 741E etc. are all acceptable)
- Type 301 op amp - 1 required; at least one spare recommended
- Protoboard - 1
- Banana sockets - about 6
- Resistors (1/4 watt) - must include several different values in every decade up to about 2M2
- Capacitors - must include some electrolytics values up to 100μF; small capacitors about 3pF, 10pF, 30pF and 100pF; a variable capacitor in the range 3pF - 30 pF; and several non-electrolytics in the range 100pF -100nF
- Cutting pliers and wire stripper
- Connecting wires (cables for connection to lab instructions will be available in the lab)
- Scientific calculator eg. Casio FX82 or similar
- Graph paper - several sheets of linear x linear and log x linear
- Digital multimeter will be supplied to you in the lab, but you may use your own.

Module organiser

Section	*Activity*	*Suggested time*
1	Basic op amp circuits Skill practice 1 Skill practice 2	6 hrs 30 mins
2	DC offsets Skill practice 3	4 hrs 30 mins
3	Slew rate Skill practice 4	4 hrs 30 mins
4	Noise Skill practice 5	9 hrs 30 mins
5	Frequency compensation Skill practice 6	7 hrs 30 mins

Notes

MODULE SECTIONS

Section 1: Basic Op amp circuits

SUGGESTED DURATION	PREAMBLE
6 hrs 30 minutes	To acquaint you with the design, operation and testing of simple linear op amp circuits as detailed in the objectives. In the study of this topic: • theoretical work assumes ideal op amps • only dual supply operation is considered.

Objectives

At the end of this section you should be able to:

- ☐ state the meaning of the term 'operational amplifier' (or op amp) and state the main properties of an ideal op amp
- ☐ given suitable specifications, design the following kinds of operational amplifier circuits using dual power supplies:
  - inverting amplifiers
  - non-inverting amplifiers
  - voltage followers
- ☐ for each of the circuits in the second objective, state or sketch the correct phase relationship between input and output and predict maximum output voltage swing and minimum load resistance based on the maximum output current capability of the op amp
- ☐ given typical transconductance and transresistance amplifiers, use the relations between the output and suitable inputs
- ☐ understand, and apply in practice, proper breadboarding techniques with attention to selection of components, layout, insertion and removal of components, good wiring, earthing, supply decoupling and safe power-on sequence
- ☐ construct the circuits listed in the second objective and measure the voltage gain, input resistance, maximum output voltage swing and maximum output current, with due attention to the device specifications of maximum power supply voltages, input voltages and power supply connections.

References

The following references deal with topics in this section.
1. Jacob(1993) pp 1-22, 52-65
2. Rutkowski (1984) pp 38-51
3. Gayakwad (1993) pp 109-115, 139, 272-274, 264-269
4. Coughlin & Driscoll (1991) pp 10-11, 13-19, 39-46, 53-60, 103-110, 117-122

The operational amplifier

The operational amplifier (op amp) is a high-gain DC amplifier with differential input and single-ended output as illustrated in Figure 1.

'Differential input' means that there are two inputs - the 'non-inverting' or (+) input and the 'inverting' or (-) input. The amplifier amplifies the difference voltage between the two inputs.

'Single-ended' output means that there is one output terminal, and the output voltage is taken from that terminal to ground. Level shifting circuits inside the op amps ensure that they can work with a variety of power supply voltages and still give the same output.

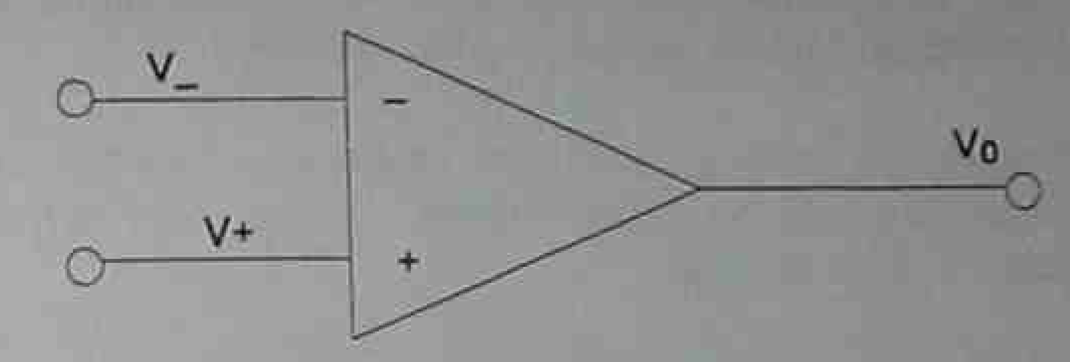


Fig. 1 Operational amplifier

The op amp is capable of working right down to DC, which is zero-frequency. This is possible because there are no blocking capacitors in the op amp.

Op amps come in many types, some having better specifications than the others. Some are general purpose, some optimised for good DC performance, some optimised for use as switches, some for high frequency etc. They are available in many packages such as 8-pin mini dual-inline-package (DIP), 10-pin flatpack, 8-pin round metal can or dual packages (two op amps in one package). General purpose types 741 and 301 are used for work in this module.

Absolute maximum ratings
Op amps are very rugged and easy to use, but you must check and stay within their absolute maximum ratings. If these are exceeded, the op amp may be destroyed. The most important ones are:

- maximum supply voltages. For the type 741, these are ±18V (for dual supply operation) or 36V (for single supply). Other types may have slightly different specifications.
- maximum input voltage at each input terminal. Typically these can range within 1 volt of the DC supply voltages. (Most op amps have built-in protection against too much input voltage.)
- maximum output current. The load resistance must be large enough to limit the current drawn from the output terminal to a safe value. (Many op amps have output current limiting protection, but this will still distort the output.)

With all the built-in protection, about the only ways to destroy an op amp are to interchange the + and - supply voltages or to connect the output terminal to a supply rail or the input signal source.

Ideal operational amplifier

For a quick understanding of op amp circuits, it is useful to suppose that the op amps are ideal. The properties of an ideal operational amplifier are that:

- the op amp has infinite gain ($A_{VOL} = \infty$)
- the op amp has infinite input resistance, between the input terminals and ground, as well as between the two input terminals themselves
- the output resistance of the op amp is zero.

Real op amps come close to the ideal. For practical op amps, the voltage gain may be ≈ 200 000 at DC and drop to a few hundred at upper audio frequencies (e.g.20 kHz). The input resistance may range from a few MΩ to a few GΩ depending on the type. The output resistance in typical circuits may be under 1Ω.

The ideal op amp is also supposed to be noiseless, its parameters independent of frequency, and not introduce unwanted phase shifts between the input and output. These issues will be dealt with in later sections.

In the study of this topic, theoretical work assumes that the operational amplifiers are ideal. Only dual supply operation is considered.

Linear amplifiers and negative feedback

Op amps have very large voltage gain. If we put the input signal directly between the + and - inputs, the output will tend to become so large that it will be distorted. For example, if the input voltage is 1 mV and the voltage gain is 50 000, the output will try to be 1 mV * 50 k = 50V, which is way beyond the voltage handling capacity of the op amp. So the output will not reach 50V, and will be distorted.

A *linear amplifier* has an output which has the same shape as the input without distortion (though phase shift is allowed). So an op amp cannot be directly connected as a linear amplifier.

Negative Feedback (NFB) is a method of reducing the effective voltage gain of a circuit in a controlled manner and thereby, linearize the amplifier circuit. With NFB, the output voltage (or part of it using a voltage divider) is connected back to the *inverting input.* If the output voltage gets too big, then the - input voltage also increases due to NFB. So the effective input voltage for the op amp is decreased, and the output also falls.

The voltage gain of the op amp, after using NFB, is called the *closed loop gain* (A_{VCL}) of the amplifier. This is usually much less than the open loop gain of the op amp itself.
Note carefully that when we talk about the 'gain of an amplifier', we usually mean the closed loop gain (unless stated otherwise).

Analysis of ideal linear (negative feedback) op amp circuits

The analysis of all negative feedback op amp circuits follows a few simple rules. These rules apply if the op amp is working as a linear amplifier, i.e. when there is NFB and there is no distortion. The rules are as follows.

- Because of the large open loop gain, a negligible voltage difference between the + and - inputs is sufficient to produce a sizeable output.
  ∴ $V_+ = V_-$ for normal operation.
- Because of the large input resistance, no current flows into the input terminals of an op amp. (Note that current may flow into the other circuit components around the op amp.)
- Because of the low output resistance, regardless of the load resistance connected to the output of the op amp, the output voltage does not vary. (Note that the load current must be less than the maximum output current rating of the op amp.)

Non-inverting amplifier

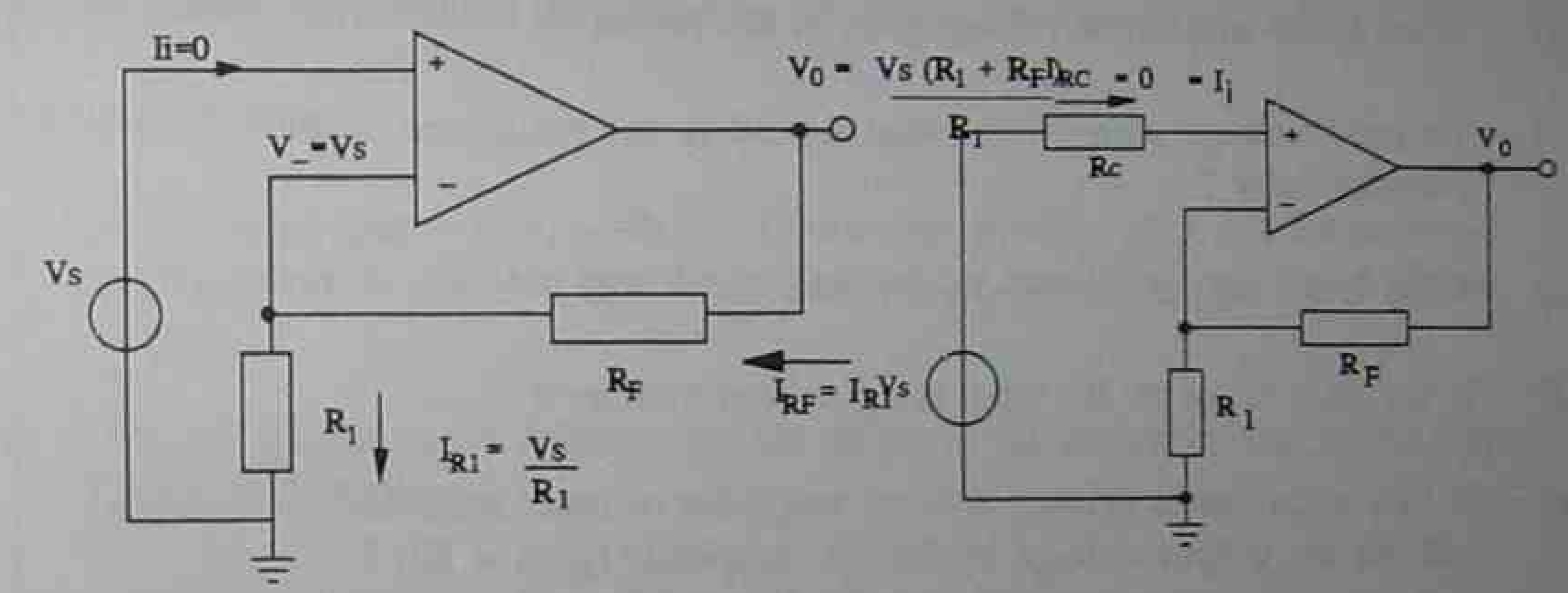


Fig. 2 *Fig. 3*

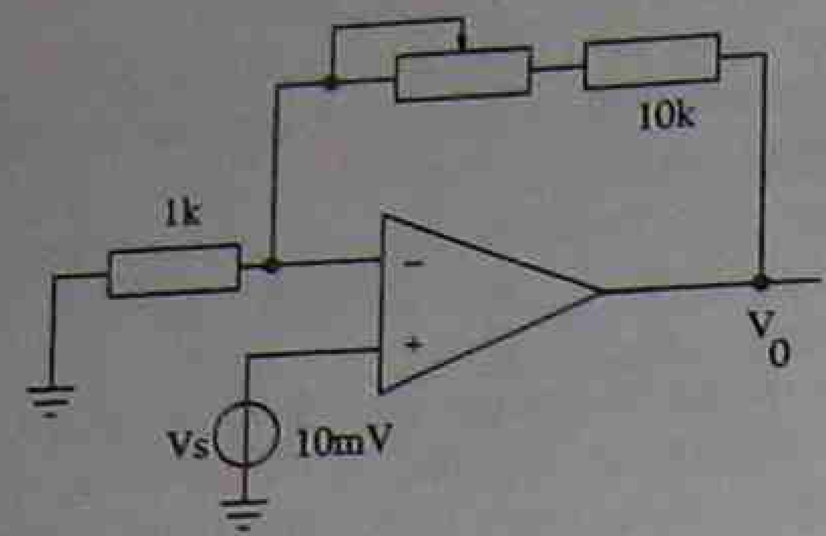
Non-inverting amplifier configurations

The circuits of Figures 2 and 3 are both non-inverting amplifiers. The input source is wired to the +input, which makes the amplifier non-inverting. The DC supply connections are not shown, even though they must be present.

If the op amp is ideal :

- the (closed-loop) voltage gain $A_V = v_o/v_s = (1 + R_F/R_1)$
- the input and output voltages are in phase
- the input resistance seen by the source $\frac{V_s}{I_i} = \infty$ (Note : it is not R_c)
- the output resistance = 0
- R_c makes no difference to the properties mentioned above. (The reason for including it is given in the next section.)

Example 1 : Non-inverting amplifier



Questions

(a) What is the minimum voltage gain of the circuit ?

(b) To get a voltage gain of 61, what should be the resistance of the potentiometer ?

(c) What is the signal current drawn from the source ?

(d) If the gain is set to 30, what is the output voltage ?

(e) If this amplifier is driving another amplifier of input resistance 5kΩ, what will be the output voltage of the first amplifier (gain = 30) ?

(f) If the signal source has internal resistance of 1 kΩ, what will be the output voltage (gain = 30) ?

Solutions

(a) $A_{v\,min}$ = (1 + 10k/1k) = 11 (when the pot is set to 0Ω).

(b) For A_v = 61, R_F = (61-1) * 1k = 60kΩ
Since 10 kΩ is already present and fixed, required pot resistance = 50 kΩ

(c) 0

(d) v_o = 30 * 10 mV = 300 mV

(e) unchanged

(f) unchanged

Inverting amplifier

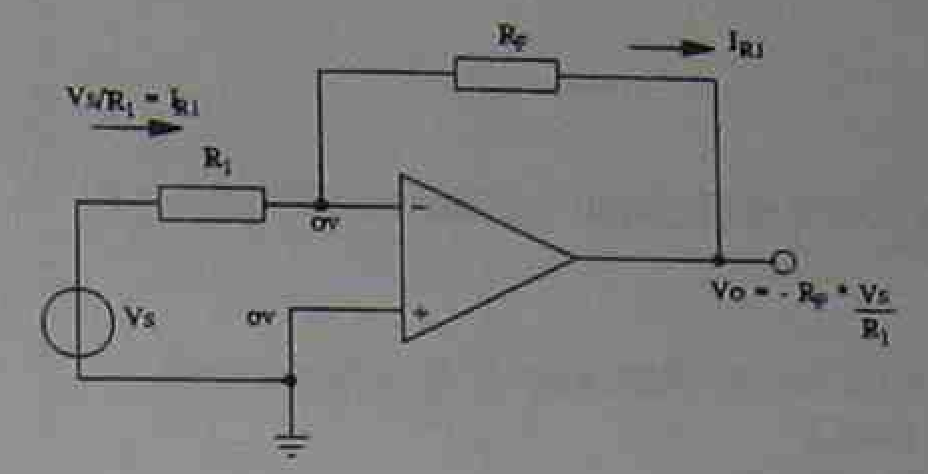


Fig. 4

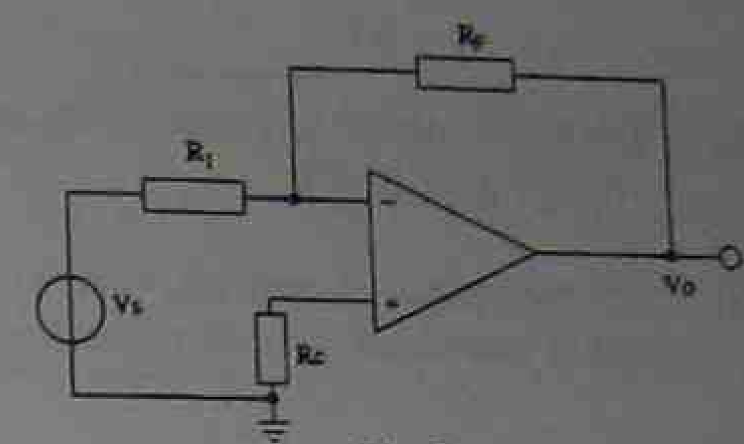


Fig. 5

Inverting amplifier configurations

The circuits of Figures 4 and 5 are both inverting amplifiers. The input source is wired to the -input, which makes the amplifier inverting. The DC supply connections are not shown, even though they must be present.

For ideal op amps:

- the voltage gain = $v_o/v_s = -R_F/R_1$ (Note : the voltage gain, for the same components, is one less than for non-inverting amplifiers, and has a minus sign.)
- the output and input voltages are out of phase (as shown by the minus sign in the gain formula)
- input resistance of the amplifier = R_1 (Note : it is not ∞.)
- output resistance = 0
- R_c has no effect on the properties of the amplifier.

Example 2 : Inverting amplifier

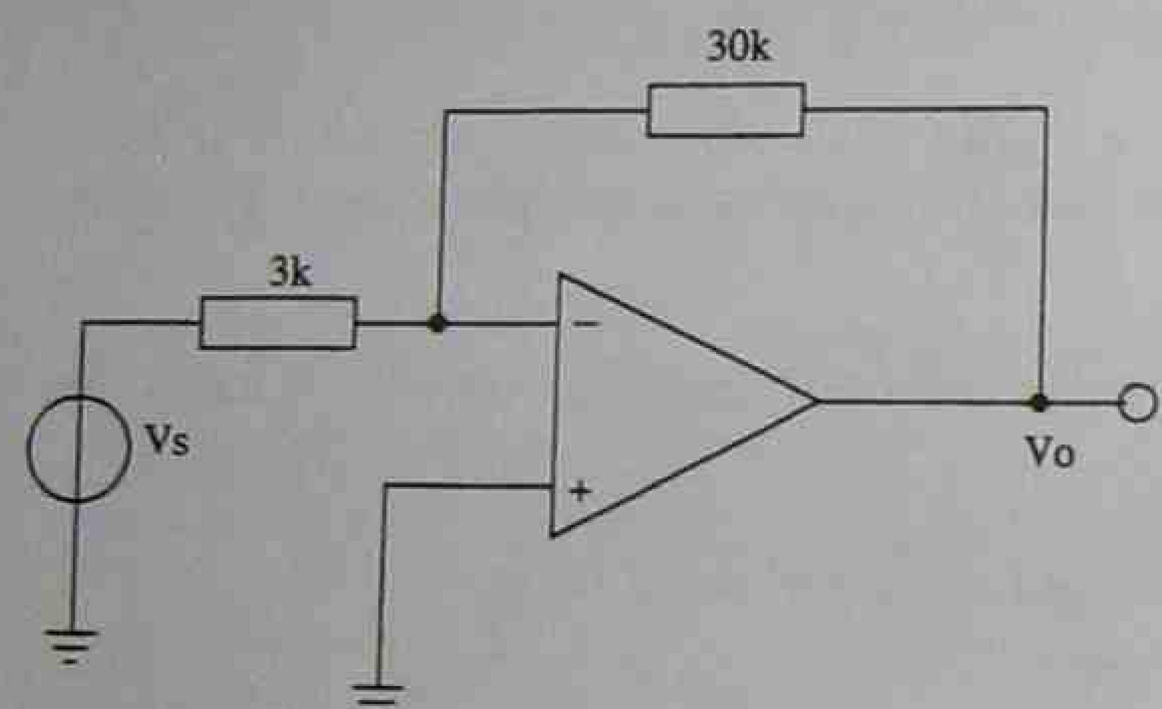
Questions

(a) Draw the circuit of an inverting amplifier with input resistance of 3 kΩ and voltage gain of 10.

(b) For the circuit in part (a), what will be the voltage gain if it is used with a signal source of internal resistance 600Ω.

Solutions

(a) R_1 = input resistance = 3 kΩ
$R_F = 10 * R_1 = 30$ kΩ
The circuit for part (a) is shown below without the power supply connections.



(b) $R_1 = 3$ kΩ as before, but
A_{VCL} = 30 kΩ / (3kΩ + 600Ω) = 8.333

Voltage follower

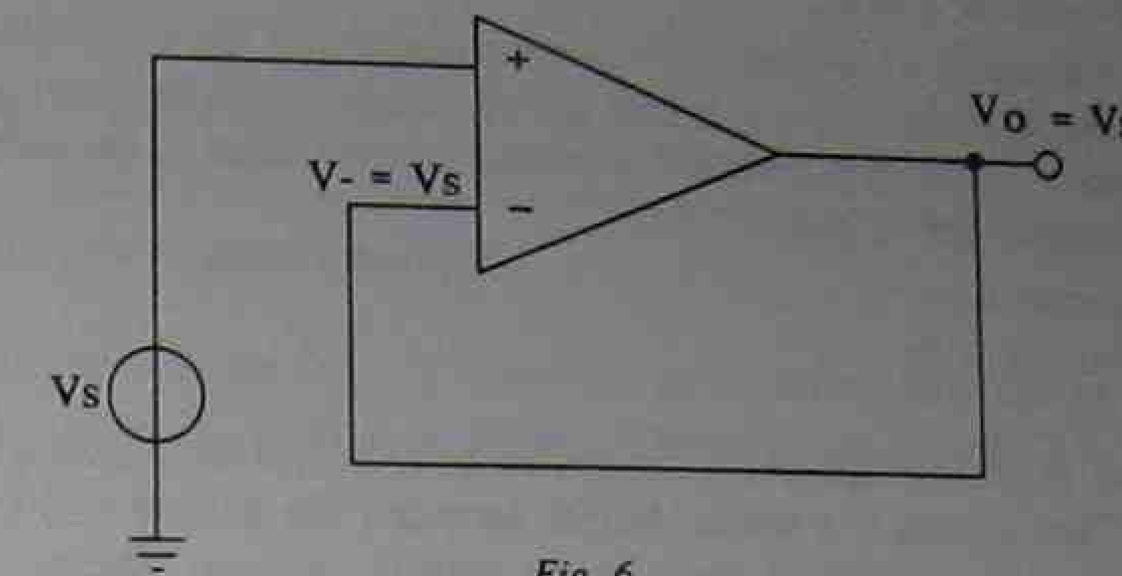


Fig. 6

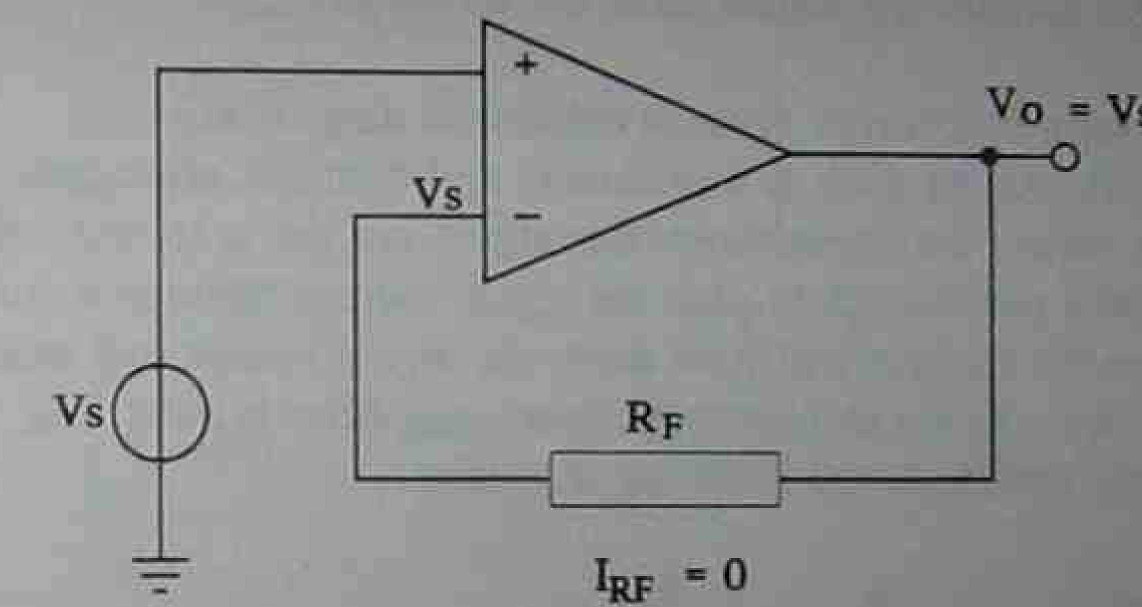


Fig. 7

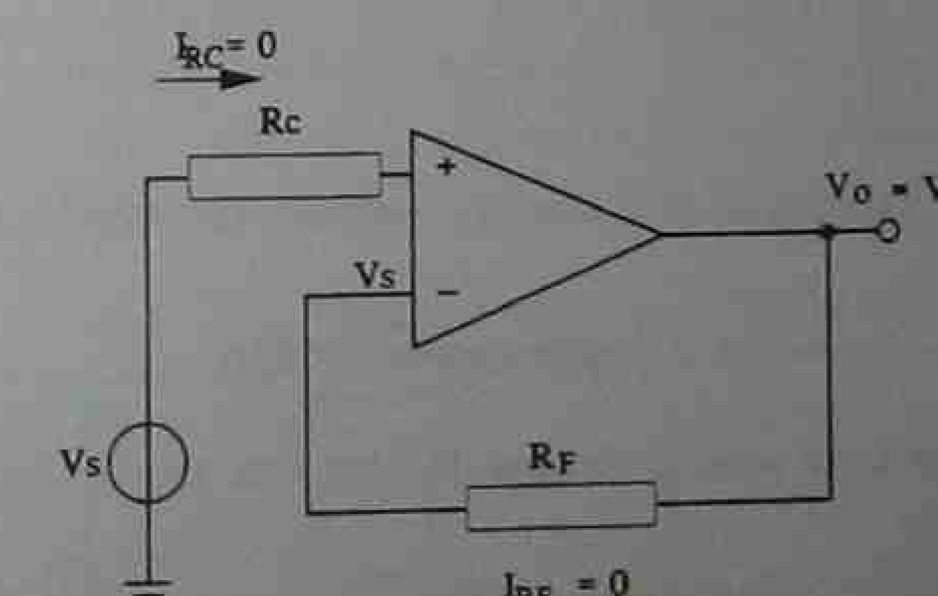


Fig. 8

Voltage follower configurations

The circuits in Figures 6, 7 and 8 are all voltage followers, though circuit 6 is the simplest and most common. In all the voltage follower circuits, the resistance R_1 from -input to ground is absent.

With ideal op amps :

- voltage gain = 1 (i.e. input signal = output signal; there is no voltage gain.)
- output and input are in phase
- input resistance = ∞
- output resistance = 0.

Even though there is no voltage gain, the useful property of this circuit is its high input resistance. This makes it a useful buffer between the source and the amplifier. The gain of the inverting amplifier can change with the source internal resistance. (There are other circuits such as the differential amplifier where the same problem arises.) The buffer isolates the source from the amplifier.

Current to voltage converter (transresistance amplifier)

Many useful signal sources such as transducers and light detectors give a current as their output. The easiest way to measure this signal current is to convert it to a voltage. It is usually not enough to pass the signal current through a resistor to make a voltage, because the resistor will load down the signal source and reduce the signal current. In such cases, an op amp transresistance amplifier is used (Fig. 9). This circuit is very similar to the inverting amplifier.

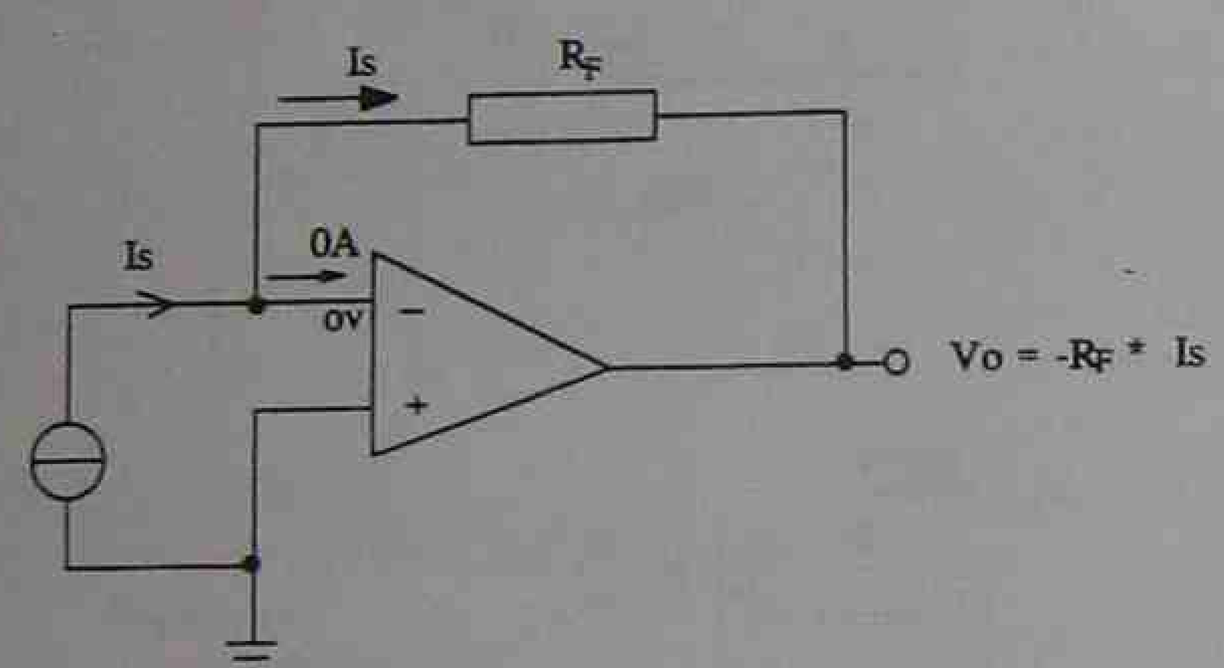


Fig. 9 Transresistance amplifier

The input current must flow around through R_F, because the op amp input does not draw any current. Also,

$V_- = V_+ = 0V$

$\therefore V_o = 0 - R_F * I_s = -R_F * I_s$

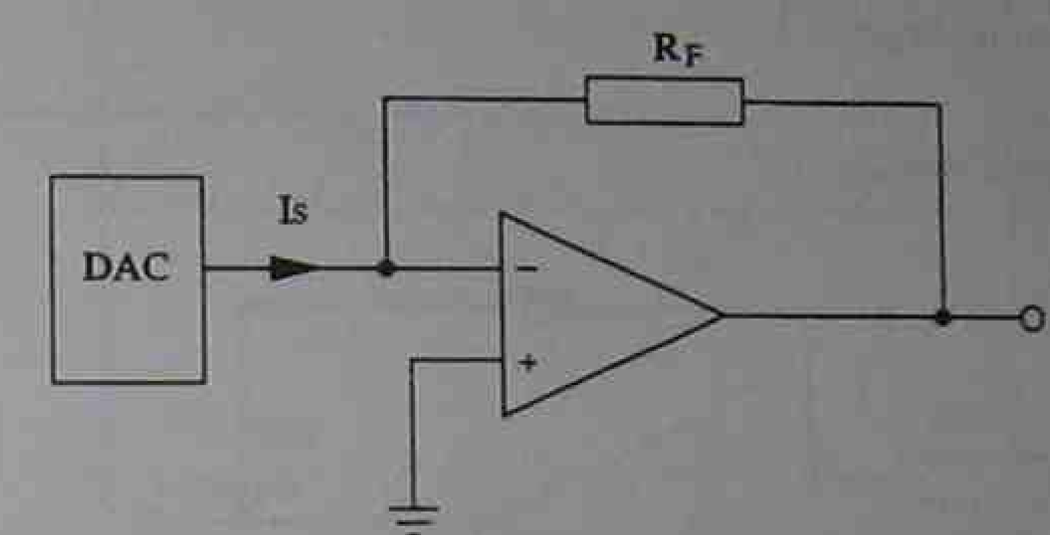
This means that $V_o \propto I_s$. The ratio V_o/I_s is called the transresistance R_T of the amplifier.

In this circuit, the -input is called 'virtual ground' because it is held at 0V, even though it is not physically connected to ground.

The signal source drives all its current into R_F. The output is measured at the op amp output, which has 0 output resistance, so the meter is not loaded, even if R_F is large.

Example 3 : Transresistance amplifier

In the following circuit, the D to A converter gives an output current in the range 0 to 1.992 mA. Select R_F to give an output voltage range of 0 to 5V.



Solution

$R_F = 5V / 1.992mA = 2.51\ k\Omega$

Voltage to current converter (transconductance amplifier)

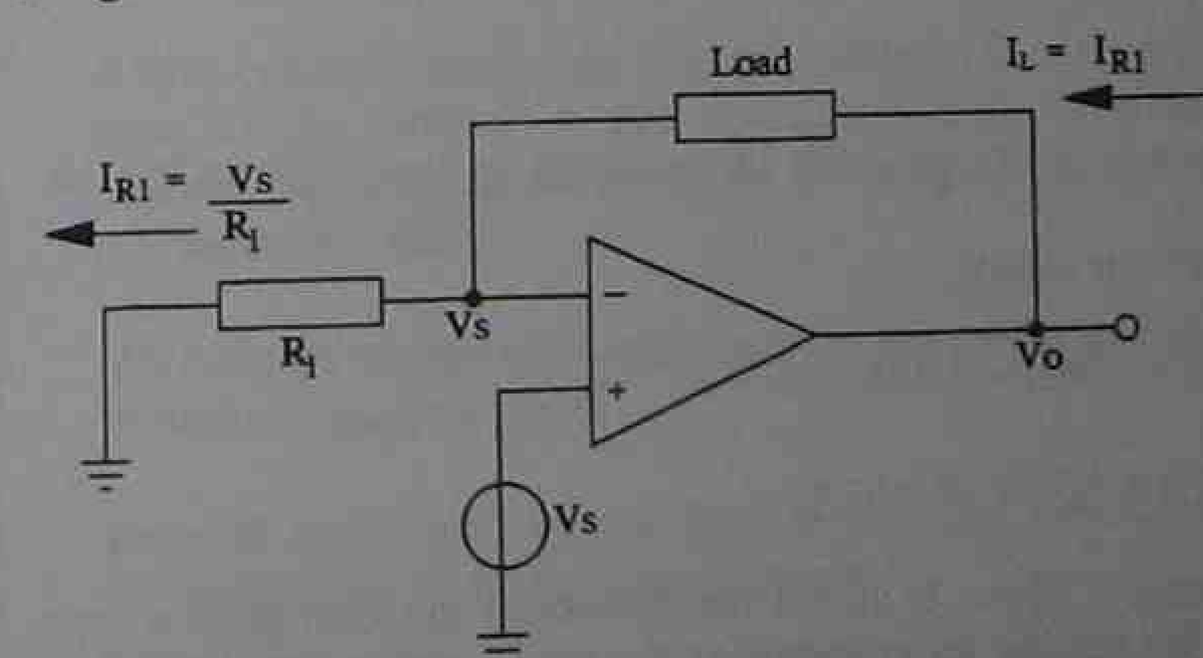


Fig. 10 Transconductance amplifier

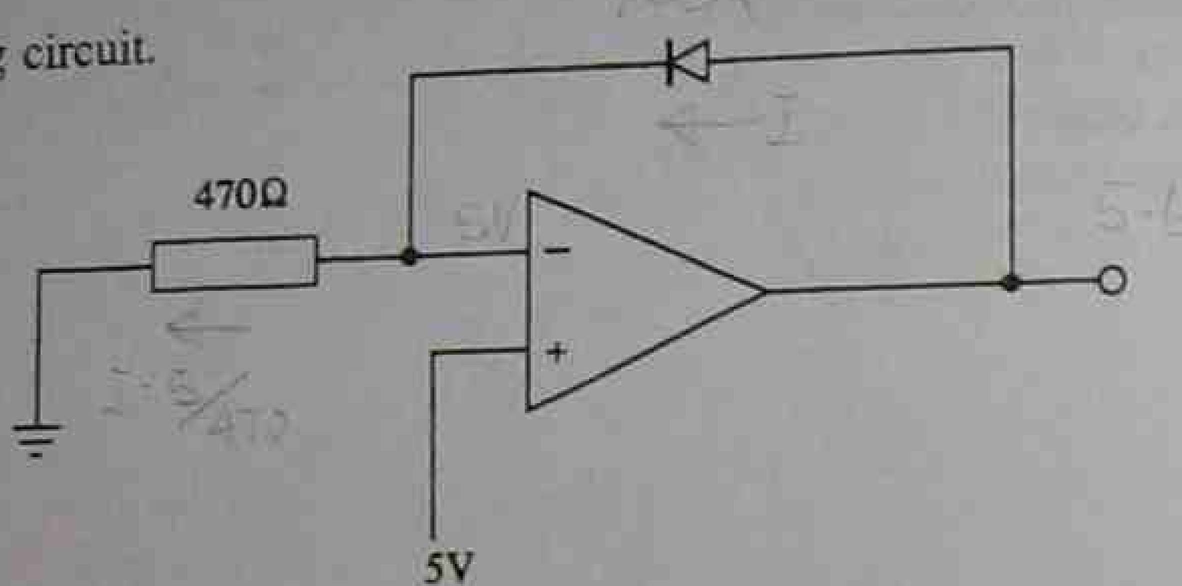
The circuit in Figure 10 is very similar to the non-inverting amplifier. Here, our aim is to pass a known current through the load, not amplifying the input voltage. The load can be any circuit components, possibly even diodes, rectifiers or meters. The purpose of the op amp is to isolate the source from the load. The only conditions are that the load must allow voltage to feed back from the output to the -input, and the output is not saturated (clipped).

Using the circuit rules for negative feedback op amp circuits:

- $V_- = V_+ = V_s$
  $\therefore I_{R1} = V_s / R_1 = I_{Load}$
- $\therefore I_{Load} = V_s / R_1$, independent of the load characteristics, *the load current being set by the value of R_1*
- I_{Load}/V_s is called the transconductance g_m of the circuit
- $V_o = V_s$ + (voltage drop in the load.)

Example 4 : Transconductance amplifier

Study the following circuit.



Questions

(a) Is the diode forward biased or reverse biased ?

(b) Calculate the diode current.

(c) Calculate the output voltage, if the diode voltage drop is 650 mV.

Solutions

(a) The direction of current in R_1, and also in the diode, is from right to left i.e. anode to cathode.
$\therefore$ The diode is forward biased.

(b) Diode current = 5V / 470Ω = 10.63 mA

(c) Output voltage = 5V + 0.65V = 5.65V

Note: The circuit in example 4 above is useful for measuring the voltage of a high impedance source (by putting an ammeter as the load), and to get the V-I characteristics of non-linear devices (by varying R_1 to change the current, without loading the source).

Maximum output swing in op amp circuits

We have mentioned that the output voltage capability of op amps is limited by the power supply. In practice, the maximum output swing (or peak-to-peak voltage) from an op amp circuit is a few volts less than the DC supply voltage differential. If we try to make the output bigger than the maximum swing, it just clips (becomes flat-topped).

The maximum output swing decreases with the load current. For example, a 741 with ±15V supply has a maximum swing of ±14V (or 28V p-p) for a 10 kΩ load, but only ±13V swing with a 2 kΩ load.

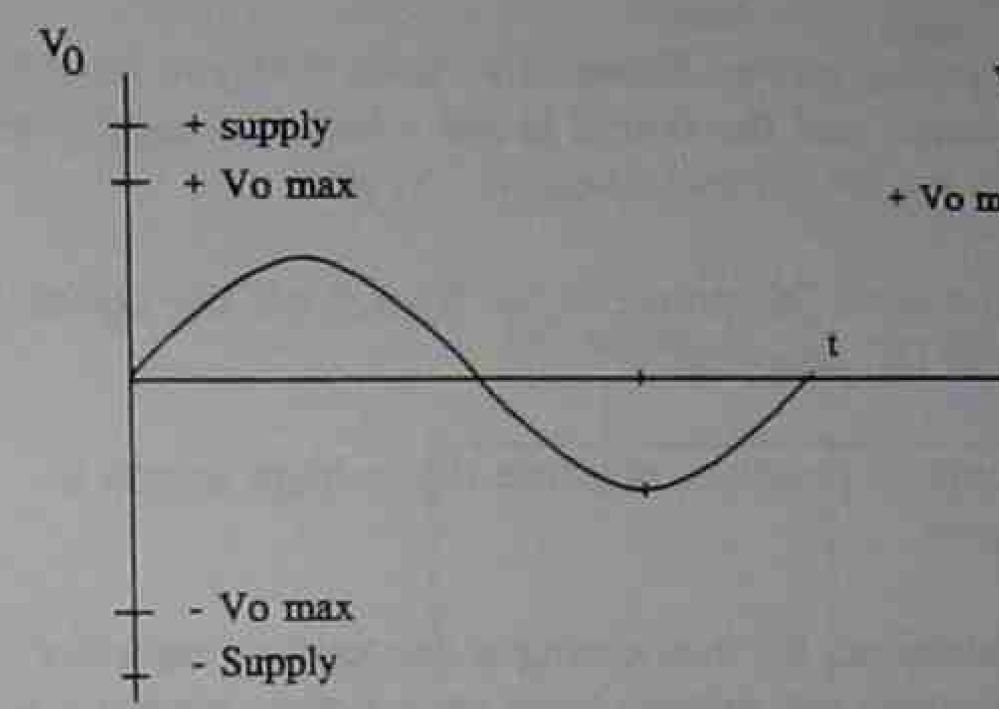


Fig. 11 Normal output

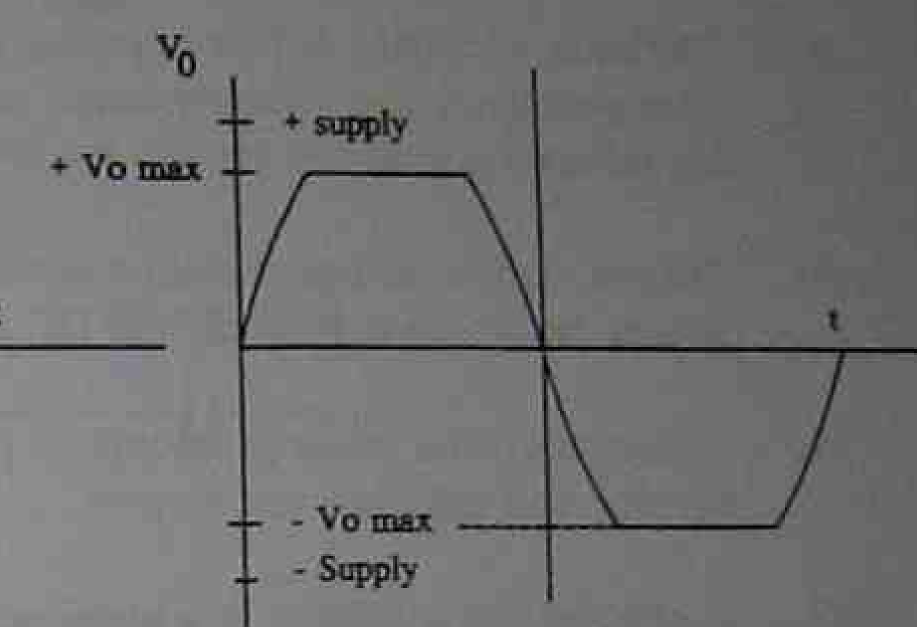


Fig.12 Clipped output

Breadboarding practice

The following tips will help you to do your practical work in this subject with the minimum of hassle and frustration, and to reduce the chances of blowing your ICs.

1. Do any wiring, or changes in wiring, with the DC power supplies switched off.

2. Set up the power supply voltages before connecting them to the circuit. Especially check that the supply voltages are within the op amp ratings.

3. Loose connections are the biggest problems to isolate in test circuits. Make sure that the sockets are mounted tightly to the breadboard and wires are inserted into sockets securely and not wrapped around leads. Alligator clips are not recommended.

4. Wire the power supply leads first. Do not forget to connect the power supply ground to circuit ground.

5. Keep all leads and wires as short as possible. Use shielded cables if possible for connection to external instruments.

6. Do not try to force thick component leads into the holes in the protoboard. This will damage the board, and the next time you use the board, you will be mystified why the circuit does not work.

7. Take care in inserting and removing ICs into boards. It is easy to bend an IC pin and very hard to see it. (It is possible to get a special tool to remove ICs from boards.)

8. Try to connect all ground connections to a single point rather than use a long ground rail, which could pick up noise.

9. Recheck all wiring before applying power. Especially check that you have not swapped the + and - supply leads, and the output is not wired to a supply or signal source lead.

10. Do not switch on signal source until DC power is on. Switch off the signal source before switching off the DC supplies.

11. Ammeters often cause problems. If possible, measure the voltage across a resistor and calculate the current.

12. If external noise pickup is a problem, try connecting a decoupling capacitor (10 nF to 100 nF - the exact value does not matter) from each power supply pin to ground. If you have to shield the whole circuit, a simple idea is to place it in a closed biscuit tin with holes cut out to bring out the leads.

13. Do not measure a voltage directly at an IC pin. You may accidentally short two adjacent pins together.

Power supply connection for op amps

For proper operation, the op amp needs DC power supply connections. Usually, we use two DC voltages — a positive supply and a negative supply of equal value. The maximum supply voltages are given in the op amp data sheets, e.g. ± 22V for general purpose op amps such as type 741. In circuit diagrams, usually the power supply connections are not drawn, though they are supposed to be there. It is important that you connect the DC supplies to your op amp properly, especially taking care to connect the power supply ground to circuit ground.

The pin connections for the type 741 op amp and the method of connecting two separate DC supplies to make a single +, - and ground supply are given below. Many other types of op amp such as 301 and LF351 also have the same pin connections as the 741 (i.e. they are pin compatible) but have improved specs.

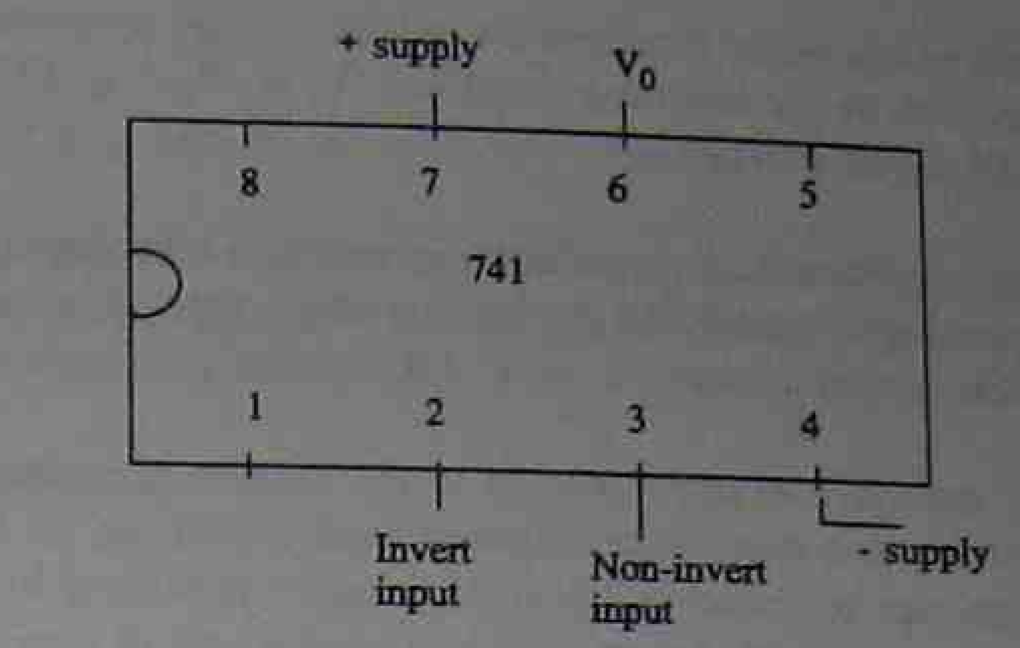


Fig.13 Pin connections for 741 and 351 type op amps

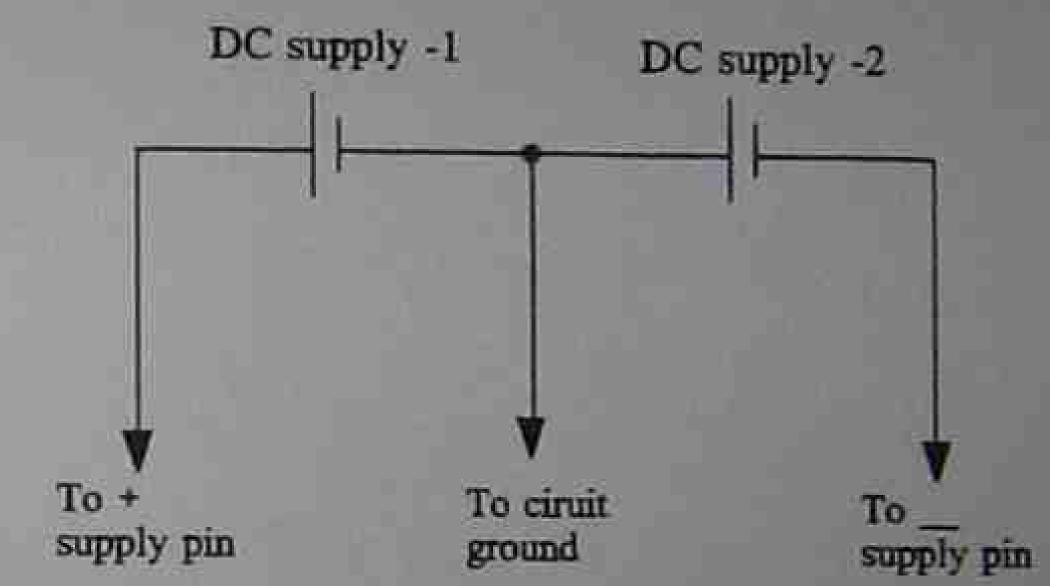


Fig.14 Dual power supply connections for op amps.

Summary

The ideal op amp has infinite gain, infinite input resistance and zero output resistance. Real op amps are close to ideal.

In non-inverting amplifiers, the input signal has a direct connection to the +input. For these amplifiers, $A_v = 1 + R_F/R_1$; $R_i = \infty$, $R_o = 0$; output and input in phase.

In inverting amplifiers, the input signal has a direct connection to the -input. For these amplifiers, $A_v = - R_F/R_1$; $R_i = R_1$, $R_o = 0$; output and input out of phase. For the same components, the inverting amplifier has a smaller voltage gain than the non-inverting amplifier.

In voltage voltage followers, there is no R_1 and the signal is connected to the +input. The $A_v = 1$ i.e. input voltage = output voltage; $R_i = \infty$, $R_o = 0$; output and input in phase.
This circuit does not amplify the signal, but is used to isolate the source from the load to prevent loading errors.

The current to voltage converter produces an output voltage proportional to the input current. It is a variation of the inverting amplifier. $V_o = -R_F * I_s$. This circuit is good for measuring small signal currents.

The voltage to current converter is a variation of the non-inverting amplifier. It is used to drive a current proportional to the input voltage through a load, which can be a linear or non-linear circuit element. $I_L = V_s / R_1$

For safe operation, you have to pay attention to the maximum supply voltage, maximum input voltage and maximum output current which can be handled by the op amp. Their values can be found in linear data books published by manufacturers. The maximum output voltage of the op amp is typically one or two volts below the power supply voltage.

Review questions

These questions will help you revise what you have learnt in Section 1.

1. An inverting amplifier works with ±12V supplies. It has a gain of -10 and an input resistance of 10kΩ. The input signal is 500 mV sine wave peak to peak, and has no internal resistance.

   (a) Sketch the circuit.

   (b) Sketch output voltage. Show important amplitude values and phase relations.

(c) Sketch the output voltage if the input voltage is 3V p-p. Show important amplitude values and phase relations.

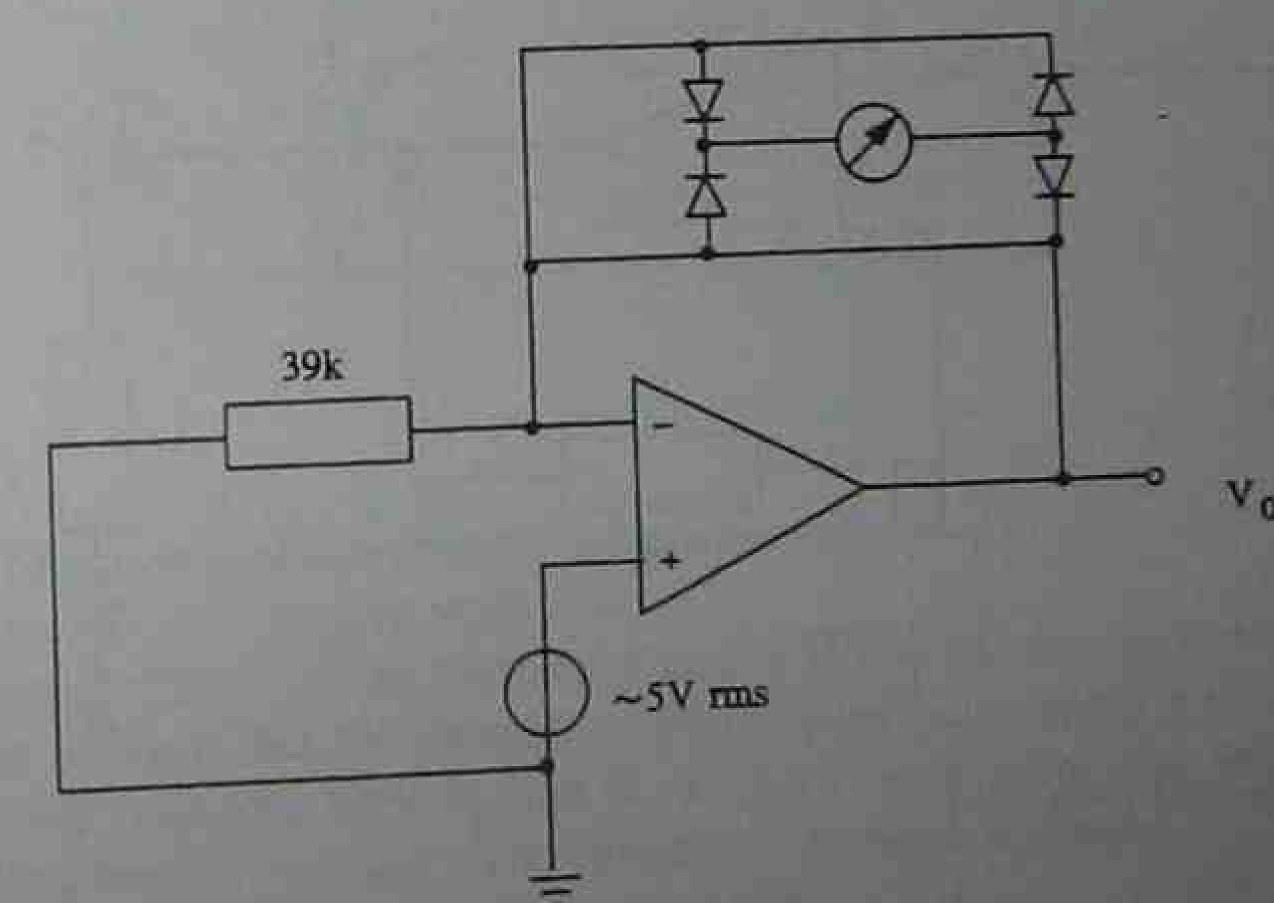
2. (a) in Q.1 how will the output voltage change if the signal source has a significant internal resistance?

(b) Sketch an addition to the circuit in Q.1 to overcome the problem mentioned in part 2(a) above.

3. In the following circuit,

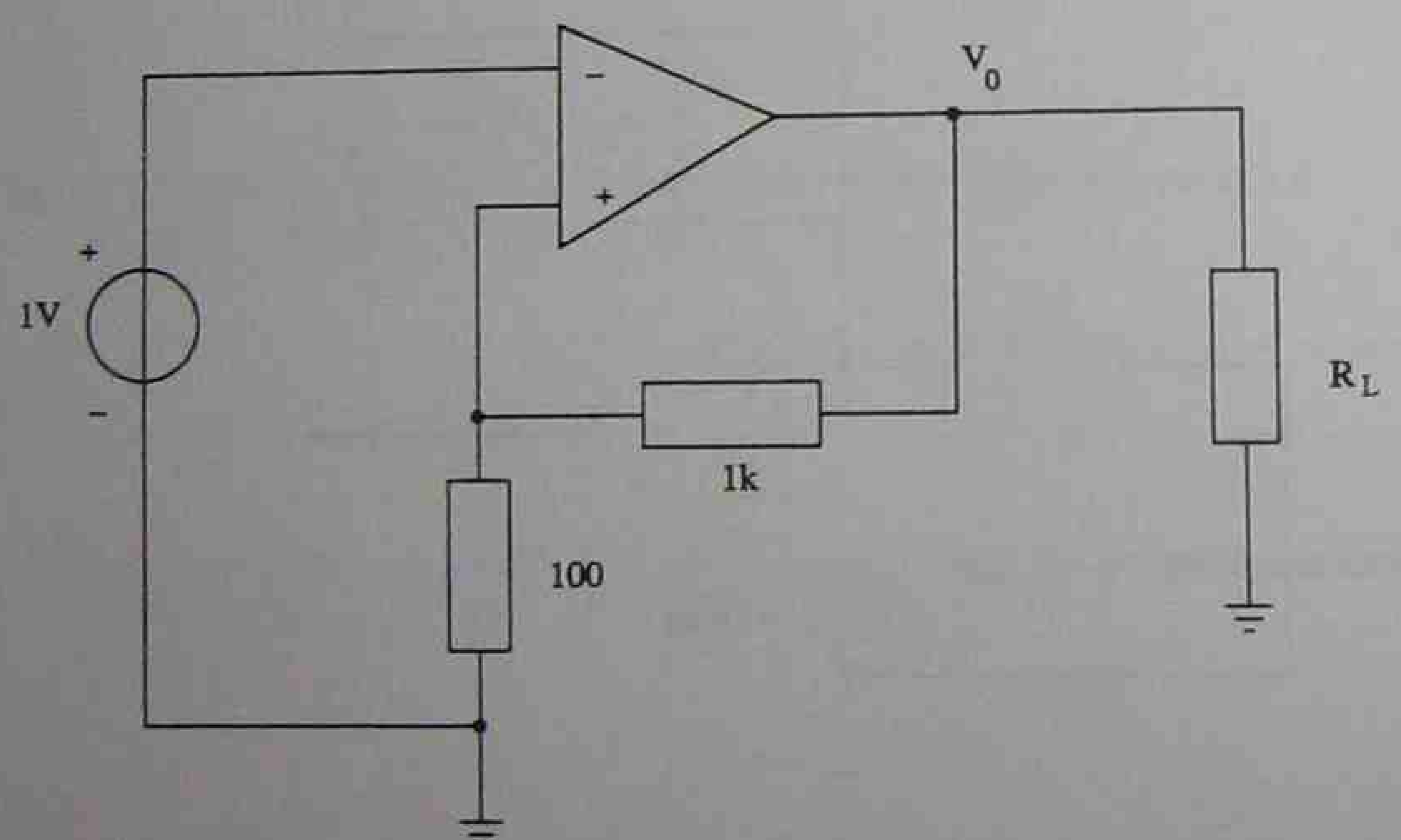
(a) What will be the average meter current if the input voltage is 5V rms sine wave? (Note: For sinewaves, full wave rectified average = 0.9 * rms)

(b) If the diodes have 0.6V forward voltage drop each and the meter has zero resistance, what is the maximum output voltage (assume no clipping)?



4. A transducer output has a temperature sensitivity of 150 nA/°C. Draw a circuit to change this to a sensitivity of 180mV/°C.

5. In the following circuit, the op amp has a maximum output current of 25mA. What is the minimum value of load resistance?



Skill practice 1

Suggested duration

1 hour 15 minutes

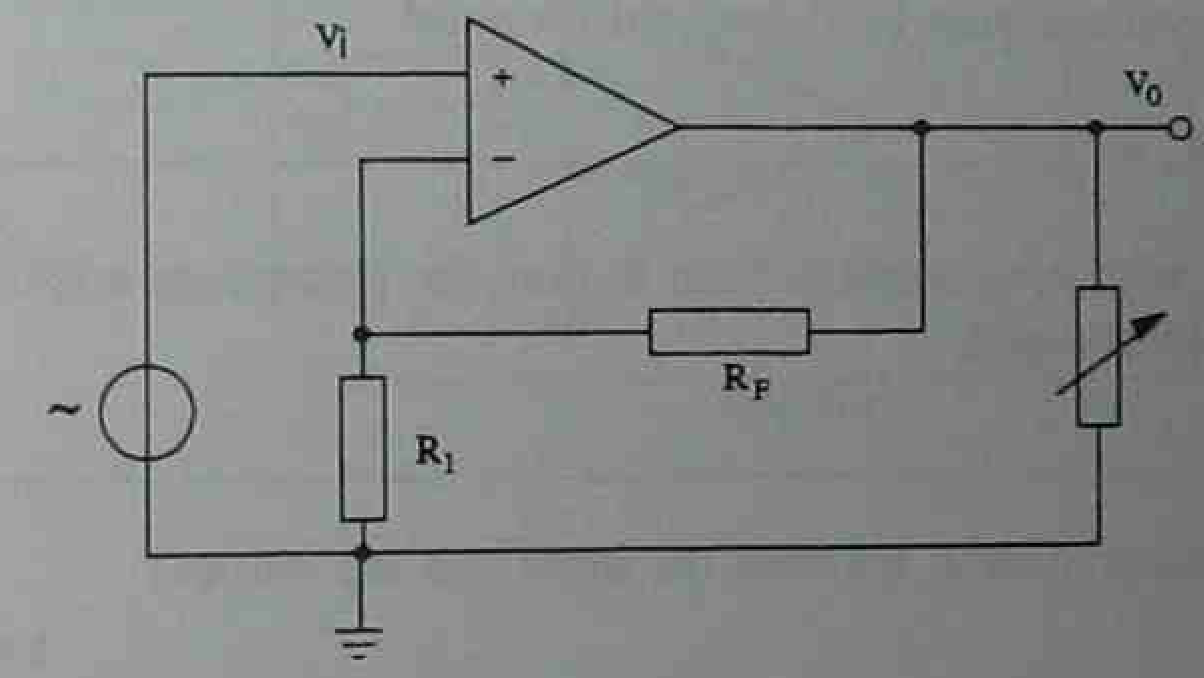
Tasks

- To measure the voltage gain of an amplifier
- To measure maximum output voltage swing of an amplifier
- To measure maximum output current from an amplifier

Equipment

- One type 741 op amp or similar (e.g. LF351)
- Sine wave generator (general purpose audio frequency function generator is adequate)
- 15 MHz dual trace oscilloscope
- ±15V DC supplies
- Decade box (range at least as wide as 100Ω to 100kΩ)
- Selection of resistors

Circuit diagram



Procedure

Step 1 *Setup and observation of output waveforms*

- Use ±15V DC supply voltages.
  Connect the circuit as shown, referring to the pin connections diagram of your op amp and the method of power supply connection in the previous section. You can vary R_F and R_1 within a wide range, for example R_F = 10 kΩ and R_1 = 1 kΩ. The procedure remains the same.
  Make the decade box resistance 10 kΩ.
- Set the signal generator to 1 kHz.
  Observe the input and the output on two channels of a CRO and adjust input voltage till the output is reasonably big (i.e. well above the noise level and not clipped).
- Observe the phase relation between the input and the output.

Step 2 Measurement of voltage gain

- Measure V_o and V_i using the same instrument (i.e. both can be measured p-p using a CRO, or both rms using a voltmeter).
  V_o =
  V_i =
- Calculate the experimental voltage gain = V_o/V_i =

Step 3 Maximum undistorted output voltage swing

- Connect a CRO channel to the output and increase the input signal until the output just begins to clip.
- The maximum undistorted output peak-to-peak voltage swing =

Step 4 Maximum output current

- Keep the output signal just below clipping. Decrease R_L until the output just begins to get distorted. Measure the **peak** output voltage and divide it by R_L, which gives the maximum output current.
  v_o (peak) = R_L =
- Maximum output current = $V_o/R_L - V_o/(R_F+R_1)$ =

Discussion questions

1. Calculate the voltage gain by theoretical formula:

   $A_v = 1 + R_F/R_1$ = ______

   Compare this with your result in Step 2. Find the percentage error in your experimental result.

   ______

2. What is the phase relation between the input and the output?

   ______

3. How many volts is your result for the maximum output voltage swing below the power supply differential?

   ______

4. Refer to the databook for your op amp and find out the specification for the maximum output current of your op amp.

   ______

   What is the percentage difference from your result in Step 4?

   ______

Skill practice 2

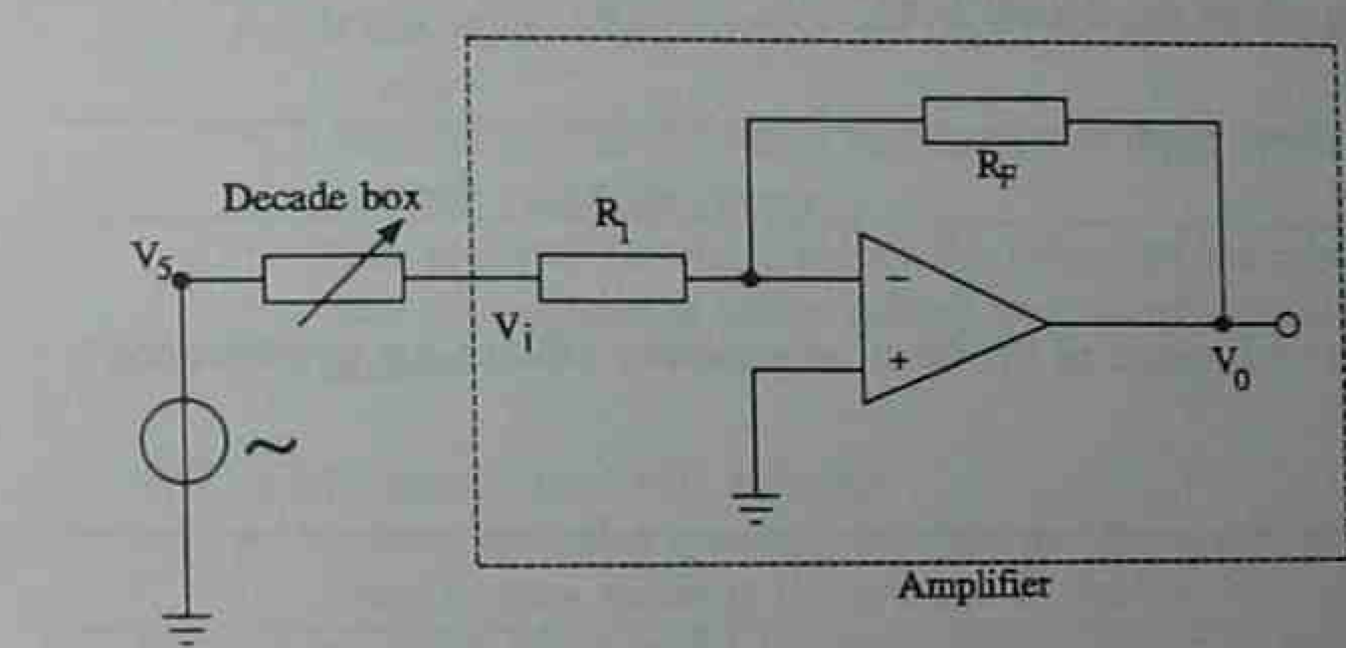
Suggested duration
1 hour 15 minutes

Task
To measure the input resistance of an amplifier.

Equipment

- One type 741 op amp or similar (e.g. LF351)
- Sine wave generator (general purpose audio frequency function generator is adequate)
- 15 MHz dual trace oscilloscope
- ±15V DC supplies
- Decade box (range at least as wide as 100Ω to 100kΩ)
- Selection of resistors

Circuit diagram



Procedure

Step 1 Setup

- Use ±15V supplies.
- Connect the circuit with suitable values of R_F and R_1 (e.g. 10 kΩ and 1 kΩ respectively).
- Make the decade box resistance = 0.
- Set generator to 1 kHz.
- Observe the output on a CRO and adjust input signal amplitude to get a good output (as in skill practice 1).

Step 2 Measurements

- Observe V_i and V_s on two channels of a CRO.
- Increase the decade box resistance until V_i is as close as possible to 50% of V_s. (This may not be always possible.)
- At this point, measure V_i and V_s and note the decade box resistance, R_D.

Step 3 Calculation of input resistance

- Calculate the input resistance, $R_i = R_D * V_i / (V_s - V_i)$

Discussion questions

1. For your amplifier circuit, what is theoretically the input resistance?

____

2. What can you say about the input resistance if V_i remains very nearly V_s, no matter how you adjust the decade box?

____

____

3. What can you say about the input resistance if V_i remains close to zero, no matter how you adjust the decade box?

____

____

4. Does the internal resistance of the function generator affect your experimental result? Why?

____

____

Section 2: DC Non-idealities

SUGGESTED DURATION	PREAMBLE
4 hrs 30 mins	To define the DC non-idealities of amplifiers: input bias current, input offset current, input offset voltage and their drift with temperature; predict their effect on the output of common amplifier circuits; and explain methods to nullify the effects.

Objectives

At the end of this section you should be able to:

- ☐ in relation to input offset voltage:
  - define input offset voltage and read typical values for common types of op amps
  - calculate output DC offset caused by the input offset voltage for common amplifier circuits
  - state practical means to reduce the effects of the input offset voltage
  - state the purpose of offset nulling and recognise common offset nulling circuits
- ☐ in relation to input bias currents:
  - define input bias currents and state that the input bias current values given in data sheets is the average of the two bias currents
  - calculate the output DC offset voltage caused by the input bias currents without compensation
  - state practical means to reduce the effects of the input bias currents
  - state the purpose of bias compensation; calculate and place the correct bias compensation resistor for common operational amplifier circuits
- ☐ define input offset current; calculate the output DC offset voltage caused by input bias currents for compensated circuits
- ☐ in relation to the effect of DC offsets on output:
  - calculate the total DC output offset due to input offset voltage and bias currents for both compensated and uncompensated circuits
  - state and sketch the general effects of input offset voltage and bias currents on AC signal outputs

- ☐ in relation to drift in DC offsets:
  - calculate the effects of drift in input offset voltages and current due to temperature change and due to power supply variations on the output DC offset voltage
  - state practical means to reduce the effects of drift
- ☐ measure the input offset voltage, input bias currents, and input offset current for an op amp
- ☐ demonstrate nulling the output DC voltage of any common type of op amp (eg. 741).

References

The following references deal with topics in this section.

1. Jacob (1993), pp 171-186
2. Rutkowski (1994), pp 58-67
3. Gayakwad (1993), pp 157-194
4. Coughlin & Driscoll (1993), pp 231-248.

Introduction

Until now we have studied ideal operational amplifiers. Real, practical operational amplifiers come close to the ideal, but not quite. For quick and approximate work, we can indeed consider the op amps to be ideal and get useful results. But if we want to get the best out of our op amps, or we work with very small or very fast signals, we have to understand the kinds of errors introduced by real amplifers, and hopefully correct for them. This is the theme of this and the next three sections.

In this section, we study the DC errors in operational amplifiers. These errors are due to input offset voltage, input bias currents and input offset current. In the next three sections, we look at problems in working with AC signals.

The DC errors studied here are usually quite small, and they do not affect the AC signals greatly. However, in many measurement and testing applications e.g. light levels, temperature, pressure, the signals are very small and very slowly varying, so the DC errors in the amplifiers may be significant.

Input offset voltage (V_{io})

If we connect an op amp as a voltage follower and ground the input, we would expect zero output voltage. But practically there would be a small DC output voltage. This is called the input offset voltage.

The op amp has a differential amplifier at the input stage, consisting of one transistor for the inverting input and another for the non-inverting input. These two transistors have slightly different DC voltage and current characteristics, causing a small DC offset voltage at the output even if there is no input. That is, the input offset voltage is caused by small DC imbalances at the input stage of the amplifier. Typical values may range from a few mV to a few tens of μV.

Notes

- The input offset voltage does not directly affect AC signals.
- The input offset voltage does not actually exist at the input terminals and cannot be measured there. You can only see its effect at the output.
- Values of V_{io} vary from type to type and from device to device for the same type. Typical values of V_{io} for any type can be read from data sheets.
- The input offset voltage can be positive or negative depending on the internal details of the op amp. The data sheets always give the magnitude of V_{io} as a positive number.

Effect of input offset voltage on output DC voltage

The output DC voltage (due to the input offset voltage)

$$= V_{io} * (1 + R_F/R_1) \qquad \text{Equation 1}$$

- V_{io} is the input offset voltage of the device
- R_F is the negative feedback resistor
- R_1 is the total (effective) resistance from the inverting input to ground. If more

than one resistor is in the path from the inverting input to ground, you must take the series or parallel effective resistance appropriately.

You can recognise the second factor in equation 1 above as the gain of the non-inverting amplifier.

You should note that the same formula applies whether the amplifier is wired as non-inverting or inverting.

In the circuit below, the op amp data specify maximum V_{io} to be 3 mV. The signal source has internal resistance of 600 Ω. The maximum output DC voltage caused by the input offset voltage is 3 mV * (1 + 100kΩ / (1kΩ + 600Ω)) = 187.5 mV DC

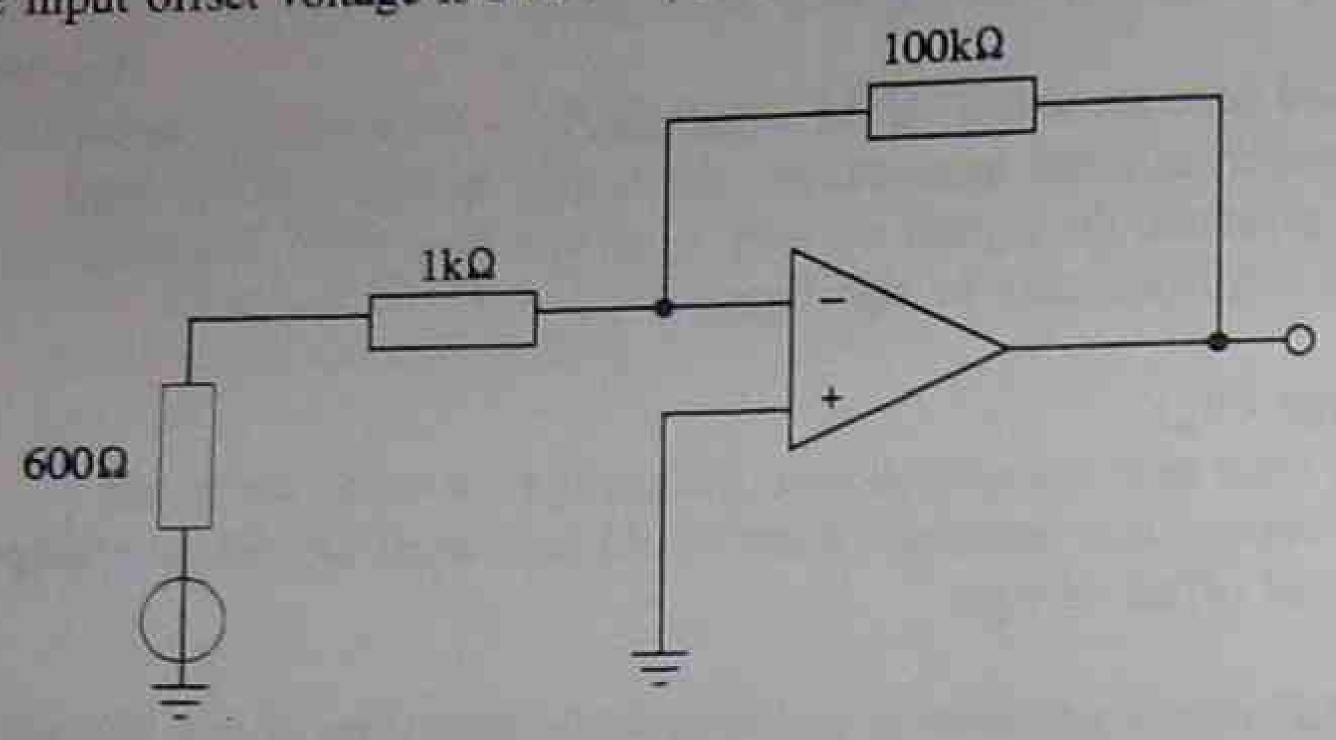


Fig. 1 Calculation of the effect of input offset voltage

Reducing the effects of input offset voltage

The effect of input offset voltage could be a nuisance in high gain amplifiers. If the problem needs attention, you can use some of the following techniques.

- Choose high quality op amps with low V_{io} specifications (Look through linear data books.)
- If possible, work with AC signals rather than DC. For example, a pulsed (square wave) light source may be used, rather than a steady (DC) one.
- Use an offset nulling circuit, as explained below.

Offset nulling methods

If the offset voltage is a significant problem, you will have to cancel out its effect by using an offset nulling circuit. The general idea is to inject a small voltage into the input stage, just enough to cancel out the DC imbalances. The data sheets for each type of device usually specify the best circuit to null the offset, for that device. For example, the recommended circuit for the 741 type op amp (taken from the National linear data book) is shown in Figure 2. (Connect the ends of a 10 kΩ pot between pins 1 and 5 and connect the centre lead to - supply).

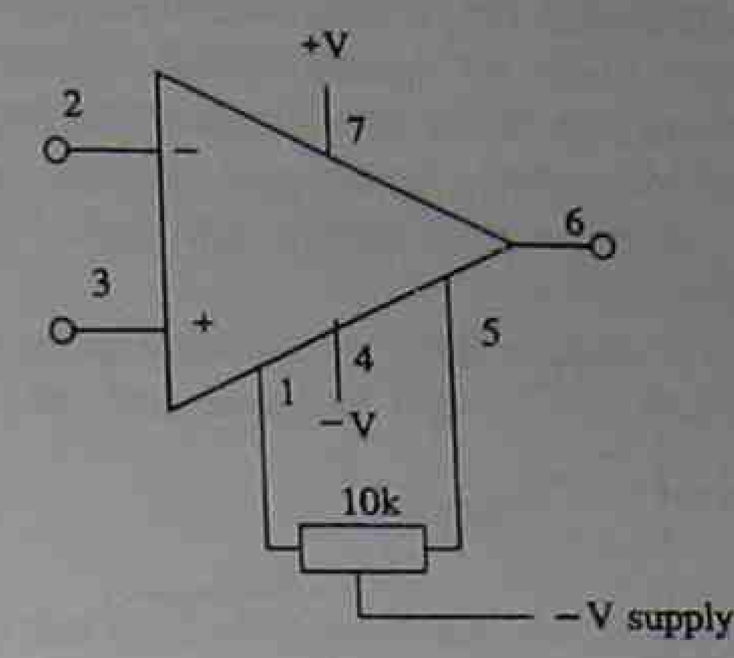


Fig. 2 Offset nulling for type 741 op amp.

After connecting the amplifier circuit and the offset nulling component, ground the input, observe the DC voltage at the output and adjust the 10 kΩ pot until output DC is zero.

Note that this circuit is not universal. It works for type 741 op amp and a few others. If you use other types of op amp, you have to check their data sheets to find the correct nulling circuit. Some common nulling circuits are given in Coughlin & Driscoll, p.244.

If the data sheets do not give any nulling circuit, you can use the following universal nulling circuit shown in Figure 3. This circuit effectively adds a small DC voltage at the input to oppose the internal DC offset voltage.

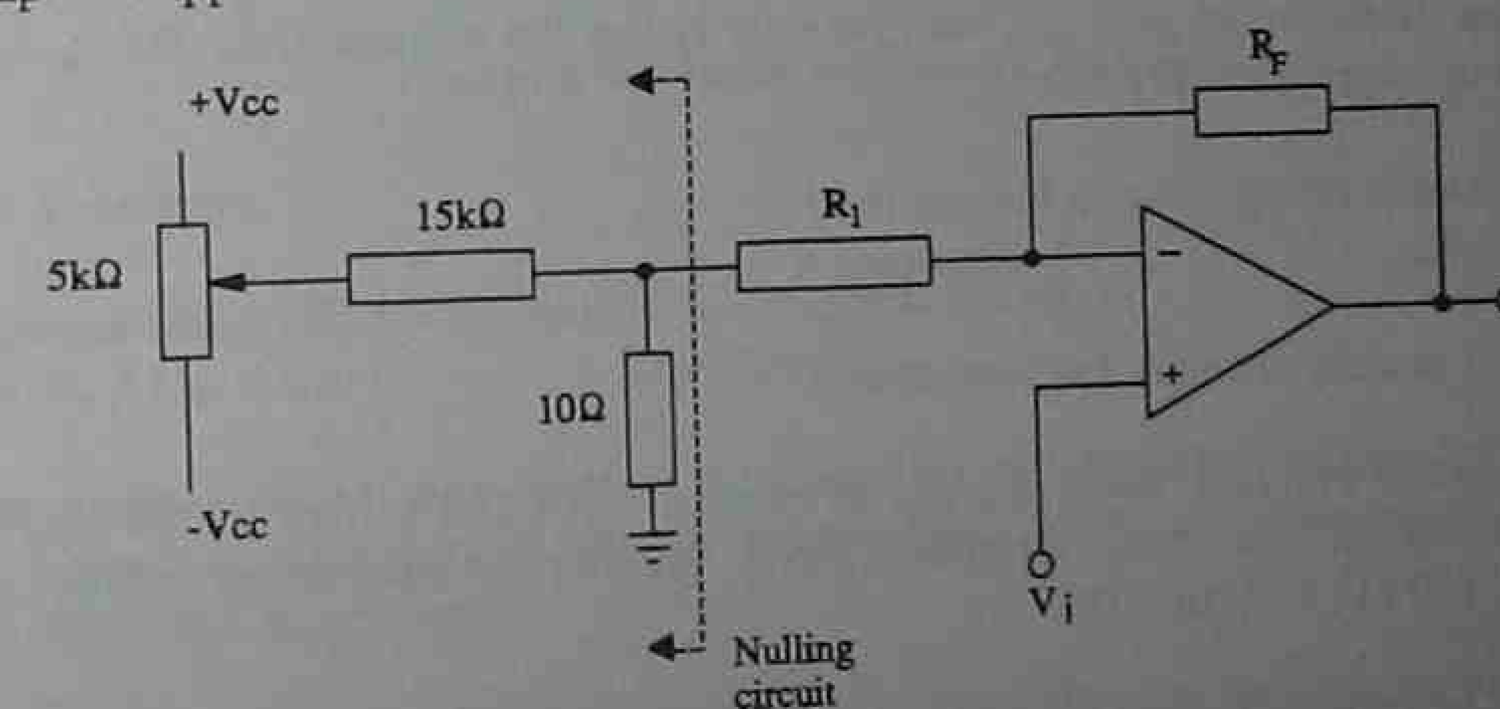


Fig. 3 Universal offset nulling circuit

Input bias current (I_B) and its effect on output DC voltage

Input Bias currents are another reason why the op amp may have a DC voltage at the output even if there is no input. Even though the ideal op amp is supposed not to draw any input current, the transistors at the input stage need small base biasing DC currents to work properly. These currents are called the input bias currents.

There are two input bias currents (one each for the inverting inputs) called I_{B-} and I_{B+} (as in Figure 4). I_{B-} and I_{B+} are nearly, but not exactly, equal. Data sheets usually give the average of the two bias currents and call it the 'Input bias current', I_B. It is defined as:

$I_B = (I_{B+} + I_{B-}) / 2$. Equation 2

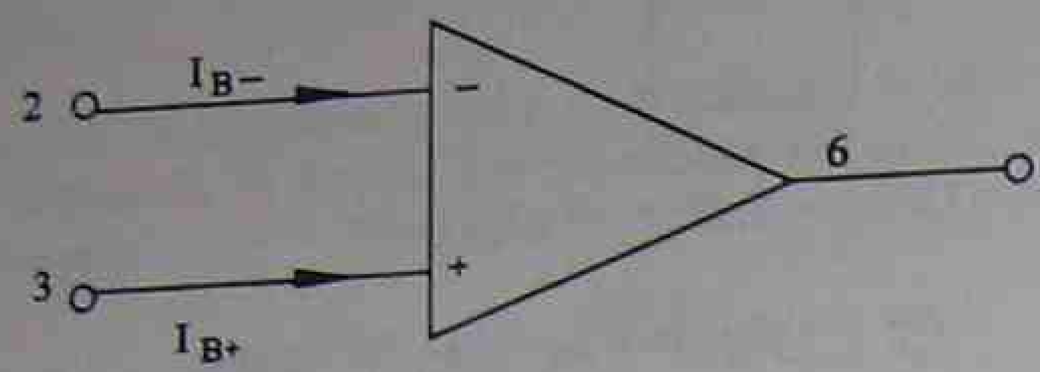


Fig. 4 Bias currents of an op amp

The value of I_B depends on the type of op amp. For general purpose op amps such as type 741, I_B may be about 100 nA. For FET input and 'superbeta' (i.e. Darlington) input op amps, I_B may be as low as 100 pA.

The input offset currents cause a DC voltage at the output because they flow through the external resistors in the circuit causing a DC voltage drop in them.

The DC output voltage (due to bias current)
$= I_{B-} * R_F$ Equation 3

If the data sheets do not give I_{B-} , but give only I_B (i.e. the average of I_{B-} and I_{B+}), then you can approximately take

The DC output voltage (due to bias current)
$= I_B * R_F$ Equation 4

Equation 3 is more accurate than equation 4.

In the circuit of Figure 1, supposing the op amp has input bias current of 100 nA, the output DC voltage due to the bias current alone, ignoring the input offset voltage, is 100 nA * 100 kΩ = 10 mV DC.

Methods to reduce the effect of input bias currents

- Choose an op amp with low bias currents specifications e.g. FET or superbeta input stages.
- Choose the smallest possible value of R_F. For example, you get the same gain by choosing R_F = 100 k and R_1 = 10 k, or by choosing R_F = 10 k and R_1 = 1 k. The second set of values give much less unwanted output DC offset voltage.
- Use bias current compensation as discussed below.

Bias current compensation and input offset current

We have seen that I_{B-} flowing through R_F causes an output DC offset voltage. This voltage can be partially cancelled by allowing the other bias current I_{B+} to flow through a resistor, thereby developing a voltage of the opposite polarity. This method of cancellation is called bias current compensation.

For bias current compensation, we need to place a bias current compensation resistor R_c in series with the non-inverting input. The value of R_c is calculated as

$R_c = R_1 \parallel R_F$. Equation 5

As before, R_1 is the total effective resistance between the -input and ground (including all series and parallel resistances in the path). Similarly, R_c includes the effect of any other resistance present between the +input and ground. After connecting R_c, no other circuit adjustment is necessary.
The value and location of R_c is the same for both inverting and non-inverting amplifiers (Figures 5 and 6).

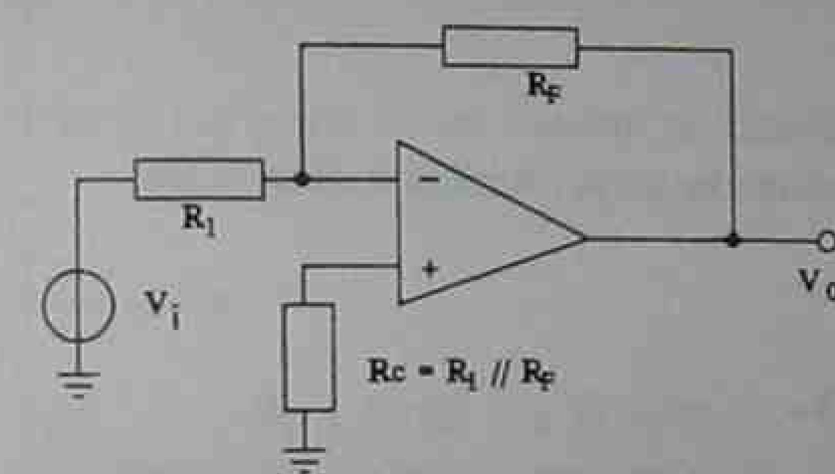


Fig. 5

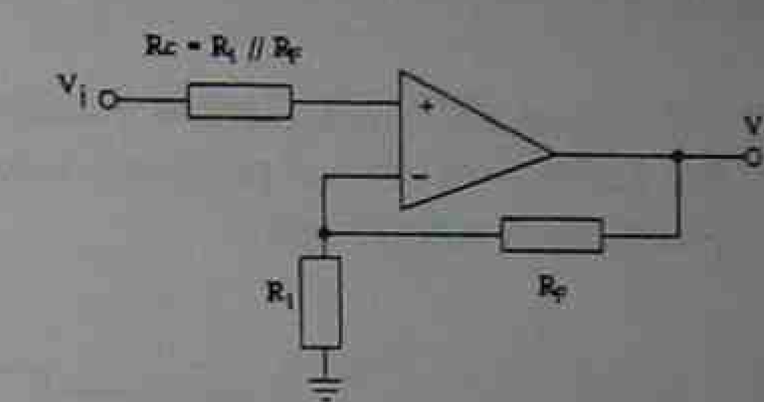


Fig. 6

Location of bias current compensation resistors

With R_c present,

the output offset DC voltage (due to bias currents alone)
$= R_F * (I_{B-} - I_{B+})$ Equation 6

Comparing equation 6 with equation 3, we see that the output DC voltage due to the bias currents is much reduced, because the two bias currents are nearly equal and their difference is very small.

The *input offset current* (I_{os}) is the magnitude of the difference between the two bias currents and its value is given in data sheets.
$I_{os} = |(I_{B-} - I_{B+})|$ Equation 7

Using the definition of the input offset current, the output offset DC voltage (due to bias currents alone, when bias compensation resistor is present)
$= R_F * I_{os}$ Equation 8

The input offset current is **not** a real current which flows anywhere. It is just a mathematical concept representing the imbalance of I_{B+} and I_{B-}, which is useful because bias compensation is commonly used to minimise the effect of DC input bias currents.

You are reminded once more that equation 3 and equation 6 calculate the same thing. Equation 3 is used when bias compensation is not present and equation 6 used when it is present.

Example 1 : Bias current compensation and its effect

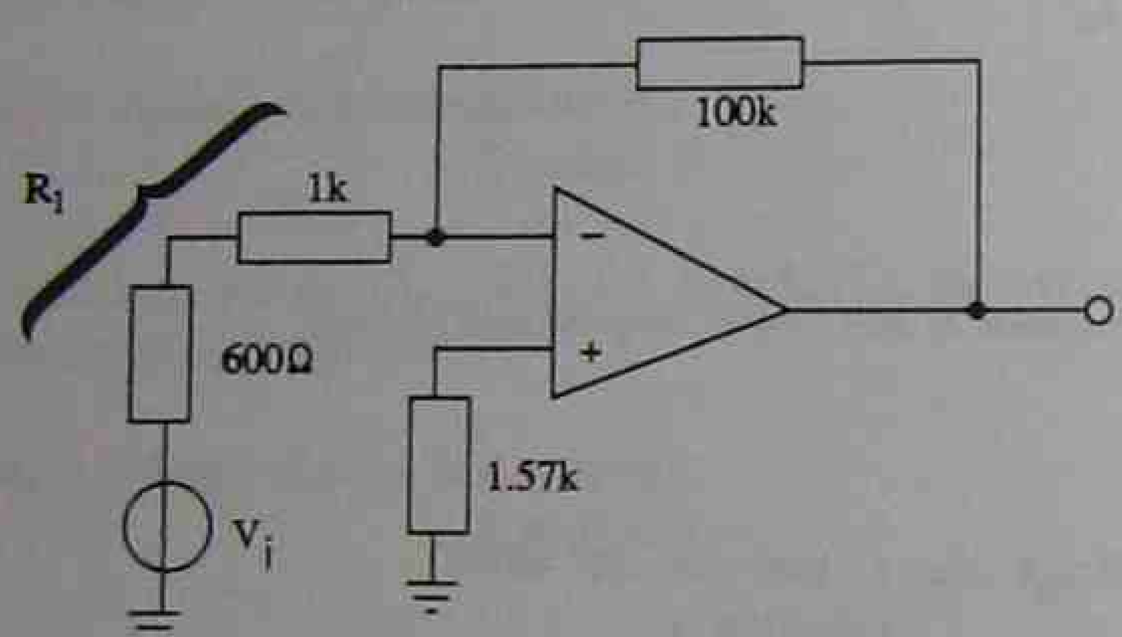
(a) For the circuit of Figure 1 design a bias current compensation.

(b) If the op amp has input offset current of 20 nA, what is the output DC offset voltage due to bias currents for the compensated circuit ?

Solutions

(a) $R_c = 100\ k\Omega \parallel (1k\Omega + 600\Omega) = 1.57\ k\Omega$

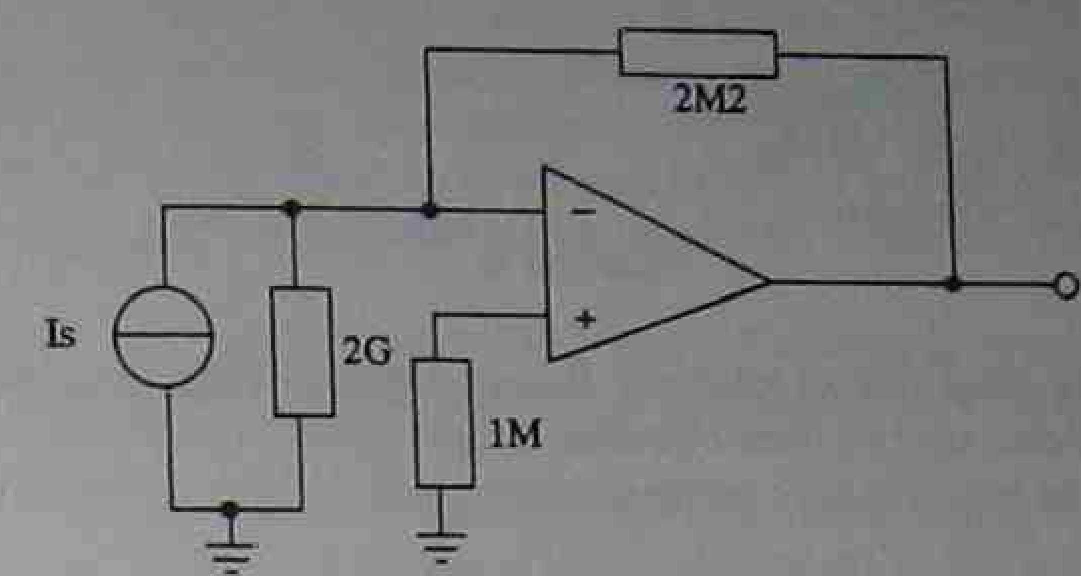
Put 1.57kΩ (or nearest preferred value) resistor in series with the +input for bias current compensation.



(b) $V_{o(DC)}$ due to bias currents with compensation =
$100\ k\Omega * 20\ nA = 2\ mV$

Example 2 : Bias current compensation and its effect

(a) Design a bias compensation circuit for the following circuit.



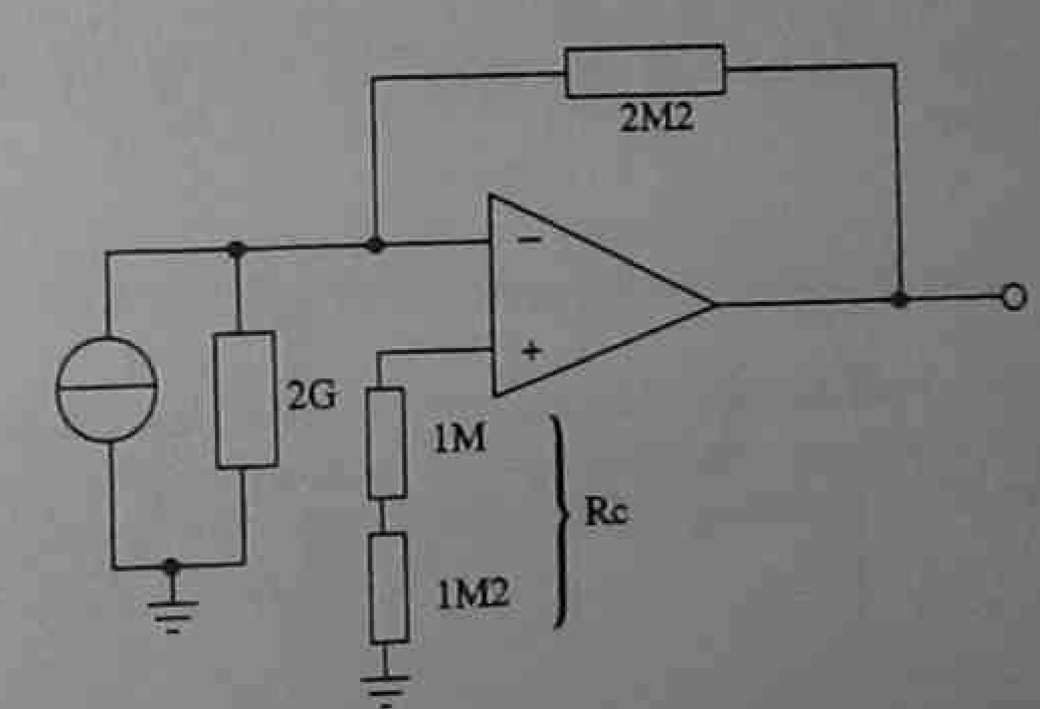
(b) If the input offset current is 10 nA, what is the output DC offset voltage due to bias currents after compensation ?

Solutions

(a) $R_c = 2M2 \parallel 2\ G = 2M2$

Resistance already present from +input to ground =
1M

∴ Place a resistor 2M2 - 1M = 1M2 in series with the +input as shown.



(b) $V_{o(DC)}$ due to bias currents = 10 nA * 2M2 = 22 mV

Note: R_c has no effect at all on the voltage gain or input resistance of the amplifier.

Total output DC offset voltage due to input offset voltage and input bias currents

The input offset voltage and input bias currents both add unwanted DC voltages at the output. These two effects are independent of each other, and both may be + or - depending on the individual device. With luck, these two effects may partly cancel each other, but in the worst case, the two effects will be additive.

Worst case total output DC offset voltage =
output DC due to input offset voltage +
output DC due to input bias currents
Equation 9

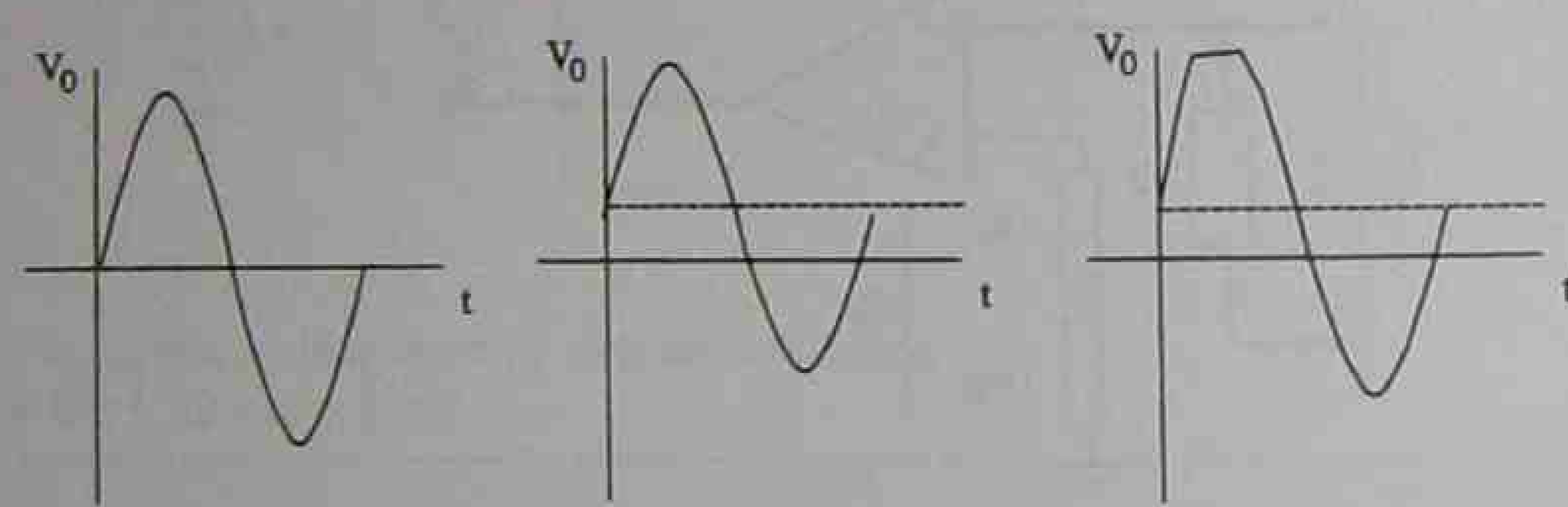
The first term on the right hand side of equation 9 is calculated using equation 1. The second term is calculated using either equation 6/equation 8 or equation 3/equation 4, depending on whether bias compensation resistor is used or not used.

For the circuit in Figure 1, supposing that V_{io} = 3 mV and I_B = 100 nA, and there is no bias current compensation, the worst case DC output offset voltage is 187.5 mV + 10 mV = 197.5 mV.

For the circuit in Example 2, supposing that V_{io} = 0.5 mV, I_{os} = 10 nA and I_B = 40 nA, and the circuit is bias compensated, the worst case DC output offset voltage is 0.5 mV * (1 + 2M7/2G) + 10nA * 2M2 = 22.5 mV.

Effect of input offset voltage on AC signals

Input offset voltage and current are DC effects and have no direct effect on AC signals. The output AC signal will be level shifted by an offset, as given by equation 9. The output DC offset could cause problems in further processing unless blocked out suitably (remember that the op amp can amplify DC), and also, reduce the maximum output AC voltage swing because one half of the AC will clip before the other half, as shown in Figure 7.



(a) ideal output *(b) level shifting due to offset* *(c) reduced swing due to offset*

Fig. 7 Effect of DC offsets on AC signals

Drift in offset voltage and offset current

The input offset voltage and bias currents vary with temperature and with supply voltage. Such variation is called drift. Because of drift, the output DC offset voltage also changes with temperature or supply voltage changes.

Data sheets specify temperature drifts for various device types (μV/°C for input offset voltage; pA/°C for input offset current or input bias current). Usually V_{io} increases with temperature for all types of op amps. I_B and I_{os} decrease significantly with temperature for BJT amps and increase rapidly with temperature for FET amps. The significance of this is that even if we carefully null out the output DC offset at room temperature, it won't stay nulled when the device temperature changes.

Change in input offset voltage = Drift in V_{io} * change in temperatureEquation 10
Change in input offset current = Drift in I_{os} * change in temperatureEquation 11

The power supply rejection ratio (PSRR), also called supply voltage rejection ratio, is sometimes specified in data sheets as the change in V_{io} for a 1V change in supply voltage e.g. 50μV/V. In other data books, PSRR is given as so many dB's e.g. 96 dB. These two definitions are unfortunately inverses of each other. (PSRR of 96 dB = antilog(-96/20) = 15.8μV/V). The significance of PSRR is that if the power supply is not well stabilized, both V_{io} and the output DC offset will drift.

Change in input offset voltage = change in supply voltage * PSRR (following the first definition)
Equation 12

Note that if there is AC ripple in the power supply, or if the supply leads pick up noise, it will show up as extra noise in the output. This is why you are advised to put bypass capacitors from supply pins of IC to ground.

Change in output offset voltage can be calculated using equations 1, 6 and 9 except that we must use changes in V_{io} or I_{os} instead of V_{io} or I_{os}.

Example 3 : Drift in output DC offset due to change in temperature

A bias compensated amplifier has R_F = 100 kΩ and R_1 = 1600 Ω. The maximum drift in input offset voltage is 30 μV/°C and the maximum drift in input offset current is 300 pA/°C. If the circuit is offset nulled at 20°C, what is the worst case output DC offset voltage at 80°C ?

Solution

Change in V_{io} = 30 μV/°C * (80° - 20°) C = 1.8 mV
Change in I_{os} = 300 pA/°C * (80° - 20°) C = 18 nA

∴ worst case change in output DC offset
= 1.8 mV * (1 + 100k/1.6k) + 18 nA * 100kΩ
= 116.1 mV

Example 4 : Drift in output DC due to change in power supply voltage

In example 3, the op amp has PSRR of 95dB. If the supply voltage changes by 2V, what will be the change in the output DC voltage ?

Solution

change in V_{io} = 95dB down from 2V = 2V/antilog(95/20) = 2V/56234
= 35.6μV

∴ change in output DC offset
= 35.6μV * (1+100k/1.6k)
= 2.3 mV

The best protection against drift problems is to keep the op amp as cool as possible (by heat sinking, fan cooling, keeping power transistors and regulators away from the op amp etc.), to use stable power supplies and bypass supply pins to ground with capacitors. For precision work, very low drift op amps are available. If you can manage to work with purely AC signals, the whole issue is not very significant.

Summary

1. The input offset voltage and input bias currents cause an unwanted DC offset voltage (DC level shifting) at the output.

2. The output DC voltage due to input offset voltage increases with the voltage gain of the circuit. The effect is the same for both inverting and non-inverting amplifiers (equation 1).

3. The effect of input offset voltage can be nullified by using the recommended offset nulling circuit for the device.

4. The output DC voltage due to input bias currents increases with the value of the feedback resistor (equation 3 or 4).

5. The effect of input bias currents can be minimized by using small value resistors and by using a bias compensation resistor in series with the +input (equation 5).

6. The input offset current is the difference between the two bias currents. Its value is much less than those of the bias currents. With bias current compensation, the output DC voltage due to bias currents depends on the input offset current and so is much reduced (equations 6 and 7).

7. The total output DC offset voltage is the sum of the effects due to the input offset voltage and bias currents.

8. Drift refers to the variation in the input offset voltage and input bias currents with temperature or supply change. Its effect on the change in output DC voltage is calculated in a similar way to equation 9.

Review questions

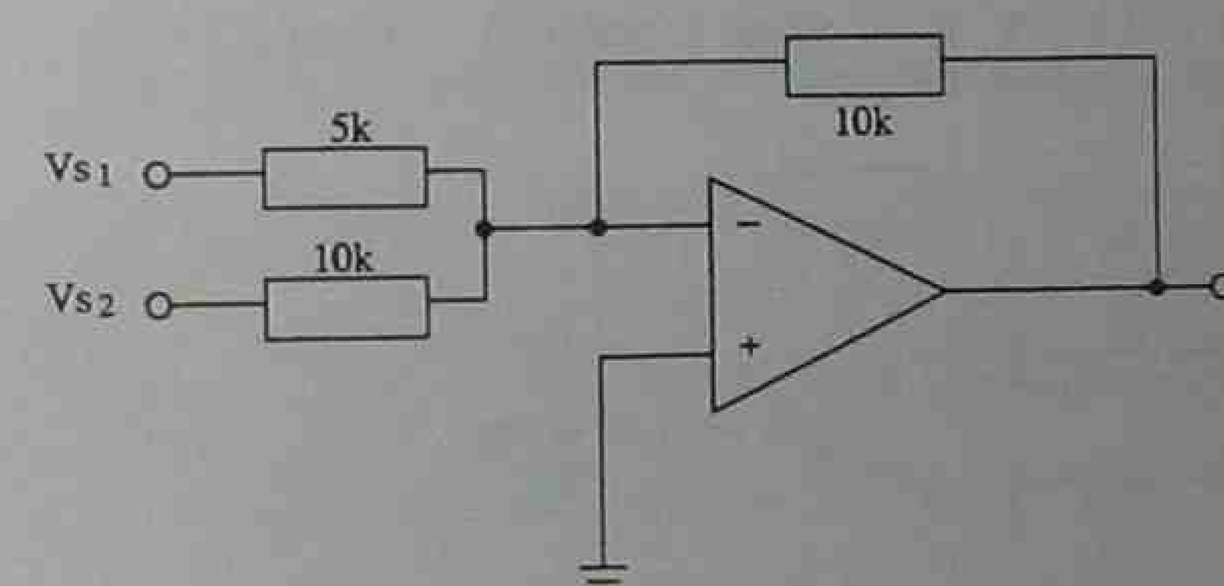
These questions will help you revise what you have learnt in Section 2.

1. For the summing circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

   (a) Calculate the worst case output DC offset voltage.

   (b) Sketch a suitable bias compensation for this circuit and calculate the added component value.

   (c) Calculate the worst case output offset DC voltage after bias compensation.

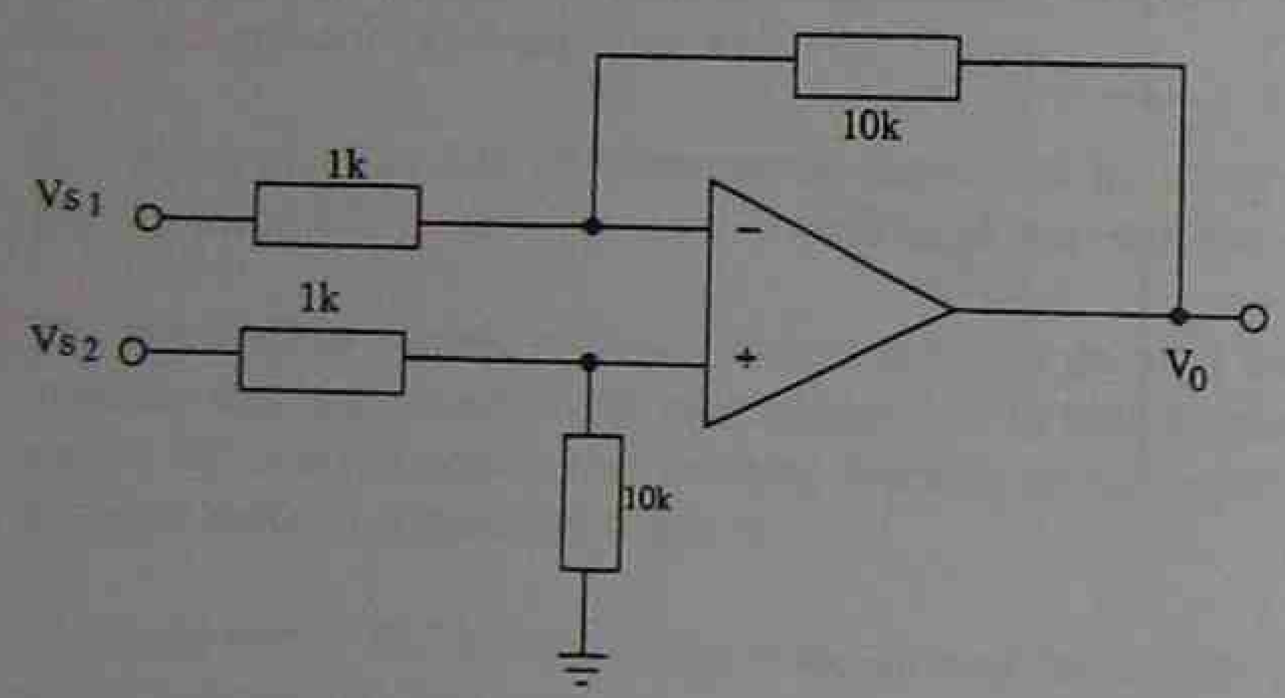


2. For the differential amplifier circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

(a) Show that the circuit is properly bias compensated.

(b) Calculate the worst case output DC offset voltage.

(c) The drift in the input offset voltage is 6 μV/°C and the drift in the input offset current is 100 pA/°C. If the circuit is nulled at 20°C, calculate the output DC offset voltage at 70°C.



3. Briefly explain the effect of DC offsets on (i) DC input signals and (ii) AC input signals.

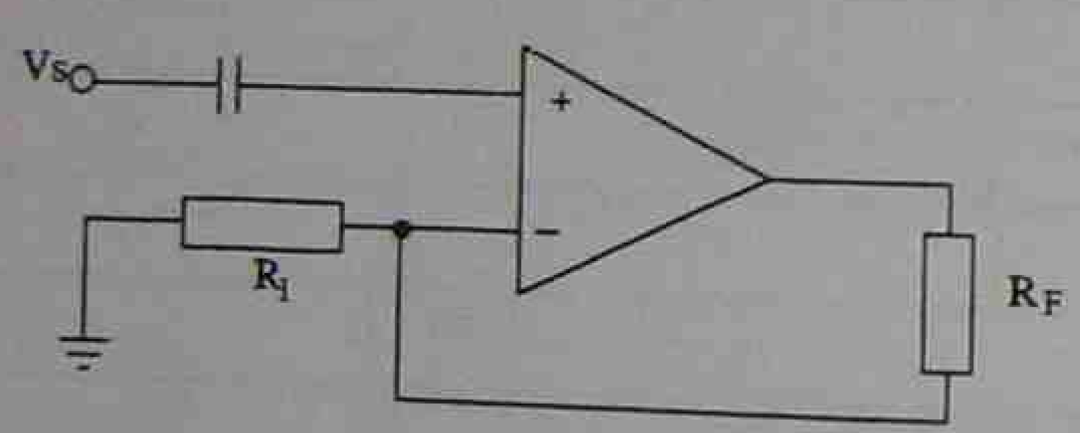
4. What are common ways to avoid errors due to drift in DC offsets in amplifiers?

5. What are the common methods to reduce the errors due to input offset voltage ?

6. What are the common methods to reduce the errors due to input bias currents?

7. From a linear data book, find (i) an op amp type with $V_{io} \le 1$ mV and (ii) an op amp type with $I_{os} \le 1$ nA.

8. The following circuit will not work. Why?



9. Refer to the circuit of Q.2 above. If the PSRR of the op amp is 30μV/V, and the power supply has unfiltered ripple of 4V p-p, what will be the output ripple?

Skill practice 3
DC offset effects

Suggested duration
1 hour 30 minutes

Tasks
- To measure the input offset voltage of an op amp
- To measure the input bias currents of an op amp
- To measure the input offset current of an op amp
- To null the output DC offset voltage of an op amp

Equipment
- 1 of type 741 op amp (Other types can be used. Check their data sheets for pin connections, recommended offset null circuit, and whether an external compensating capacitor is needed.)
- ±15 V DC power supplies
- resistors : 2 of 47Ω, 1 of 4k7, 2 of 1M, 1 of 10k potentiometer (multiturn preferred)
- DC voltmeter

Additional Information

The input offset voltage and offset current cannot be measured directly. The bias currents, being very small, cannot be measured using common laboratory instruments. These parameters must be calculated by measuring the DC output voltage and working backwards to the input.

For the pin connections of the 741 op amp and the method of connecting the dual power supply, check the skill practice exercises of Section 1. Though circuit diagrams usually do not show power supply connections, they are supposed to be there.

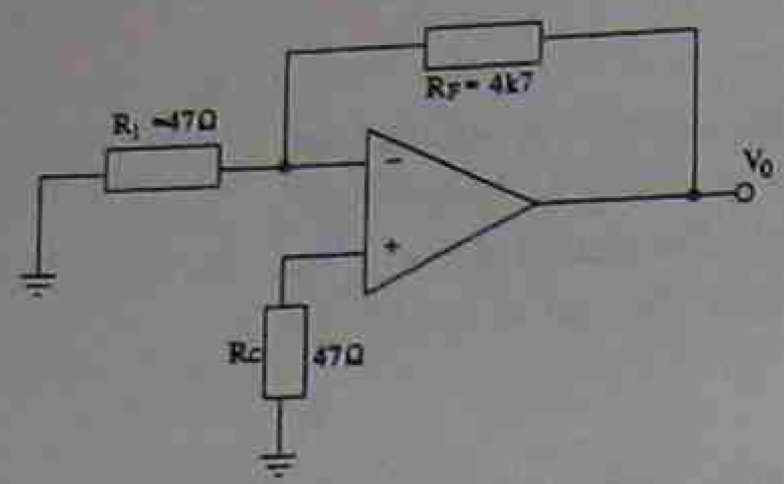
No external signal source is necessary. In the following circuits, the input is connected to ground.

(For Steps 1, 2, 3 and 4, if you do not have the recommended resistors, use others of nearby value. For Step 5, you must use a 10 kΩ pot.)

Procedure

Step 1 *Measurement of input offset voltage, V_{io}*

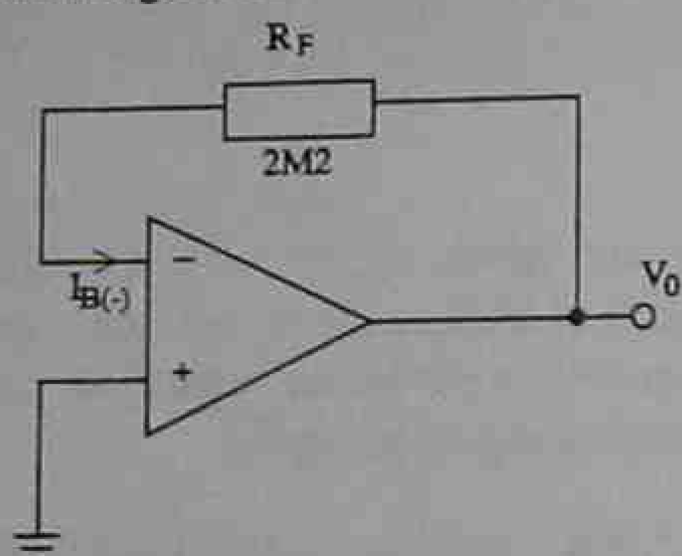
- Connect the following circuit and measure the output DC voltage V_o.



- Calculate $v_{io} = V_o / (1 + R_F / R_1) =$

Step 2 *Measurement of bias current I_{B-}*

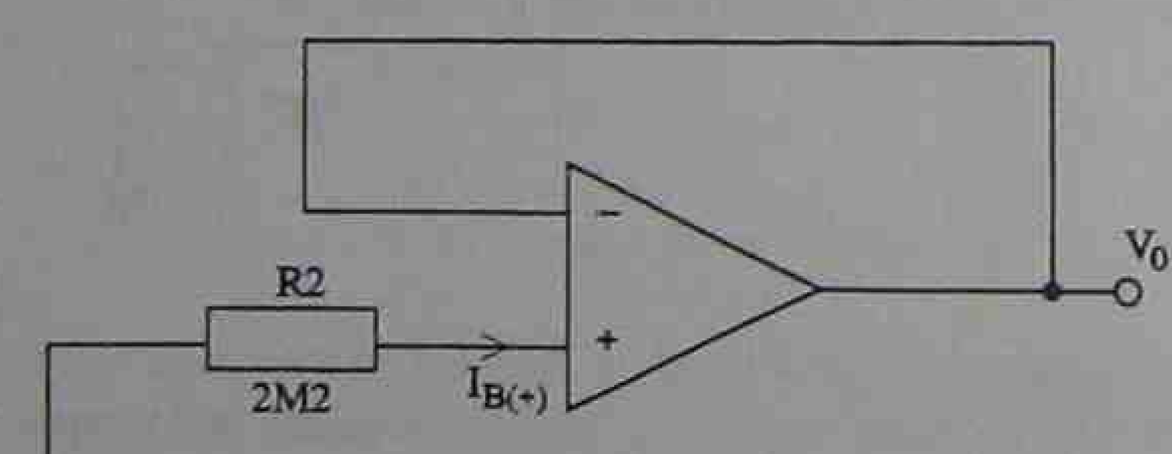
- Connect the following circuit and measure the output DC voltage V_o.



- Use the value of V_{io} from Step 1 and calculate
  $I_{B-} = (V_o - V_{io}) / R_F =$

Step 3 *Measurement of bias current I_{B+}*

- Connect the following circuit and measure the output DC voltage V_o.

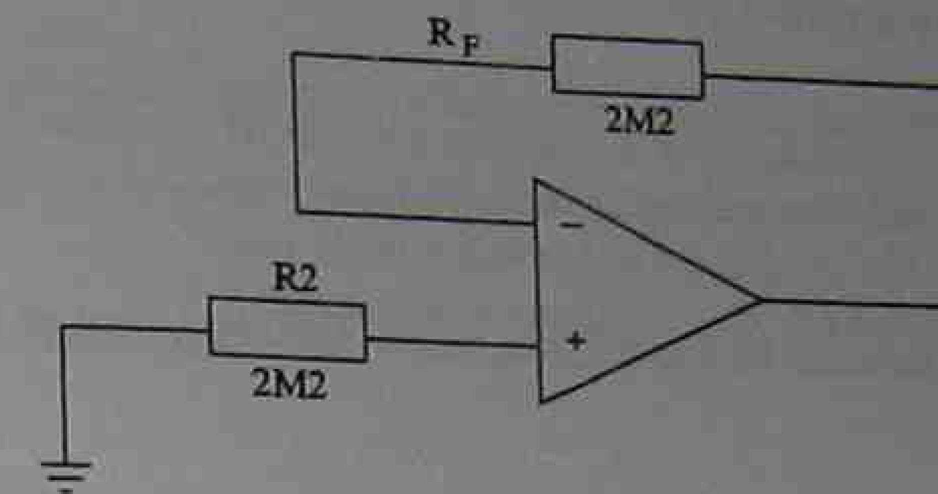


- Use the value of V_{io} from Step 1 and calculate

  $I_{B+} = -(V_o - V_{io}) / R_2 =$

Step 4 *Measurement of input offset current I_{os}*

- Connect the following circuit and measure the output DC voltage V_o.



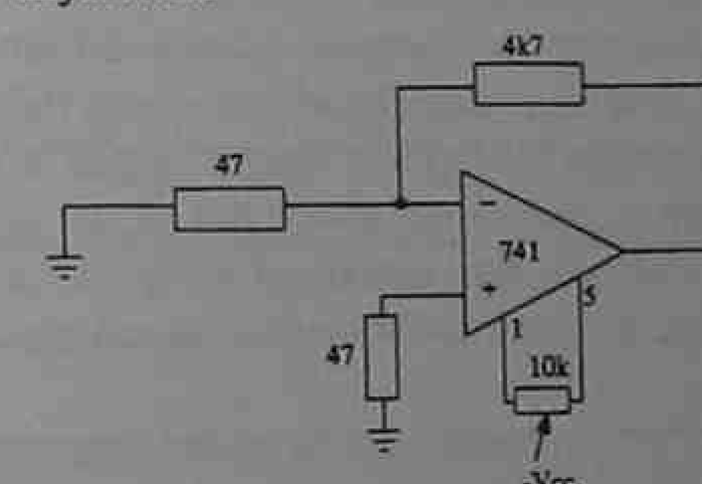
- Use the value of V_{io} from Step 1 and calculate

  $I_{os} = |(V_o - V_{io})| / R_F =$

- You can also use the values of I_{B-} and I_{B+} from Steps 2 and 3 and calculate

  $I_{os} = |I_{B+} - I_{B-}| =$

Step 5 *Offset null adjustment*



- Connect the circuit. (This circuit only works for 741 op amps. If you use any other types, check the data sheets for the correct circuit. The procedure is the same for all types of op amp.)

- Connect a DC voltmeter or CRO in DC mode to the output.

- Adjust the 10 k pot until the output DC voltage is zero.

Discussion questions

1. Compare your experimental results for V_{io}, I_B and I_{os} with data sheets specifications for your op amp.

2. Compare the output DC voltages in Steps 2 and 4 and state the effect of bias compensation.

3. Why are the resistors in Step 1 chosen to be very small?

4. Which of the two methods for measuring I_{os} (Step 4) is more accurate? Why?

Section 3: Slew rate

SUGGESTED DURATION	PREAMBLE
4 hrs 30 mins	To develop qualitative and quantitative understanding of the distortion caused by slew rate limitation; to interpret data sheet information relating to slew rate; and select components to minimise this distortion.

Objectives

At the end of this section you should be able to:

- ☐ understand slew rate and its effect on amplifier performance, and in particular:
  - define slew rate and describe its significance in applications
  - state the conditions under which slew rate distortion is prominent
  - sketch the effect of slew rate on square wave outputs of amplifiers
  - state and use the formula relating output voltage step change, rise time and slew rate
  - sketch the effects of slew rate on sine wave outputs of amplifiers
  - state and use the formula relating the output voltage sine amplitude, maximum undistorted frequency of operation and slew rate
  - define full power bandwidth and calculate
  - given the output voltage swing vs frequency graph of an amplifier, read the value of the full power bandwidth, maximum available voltage swing at any given frequency and calculate the slew rate.
  - state some common methods to improve the slew rate performance
- ☐ measure the slew rate of an amplifier
- ☐ observe the improvement in slew rate by varying the compensation capacitor in an externally compensated op amp.

References

The following references deal with topics in this section.

1. Jacob (1993), pp 195-199
2. Rutkowski (1994), pp 97-101
3. Gayakwad (1993), pp 225-231
4. Coughlin & Driscoll (1993), pp 260-264

Introduction

In many applications, amplifiers work with step changes in the input voltage e.g. sharp changes in music levels for audio amplifiers; switching the output from off to on or the other way in amplifiers used for interfacing to digital circuits. If we apply a step input, we would expect a step output (Figure1). In reality, the output of the amplifier does not respond instantly to the input (Figure 2).

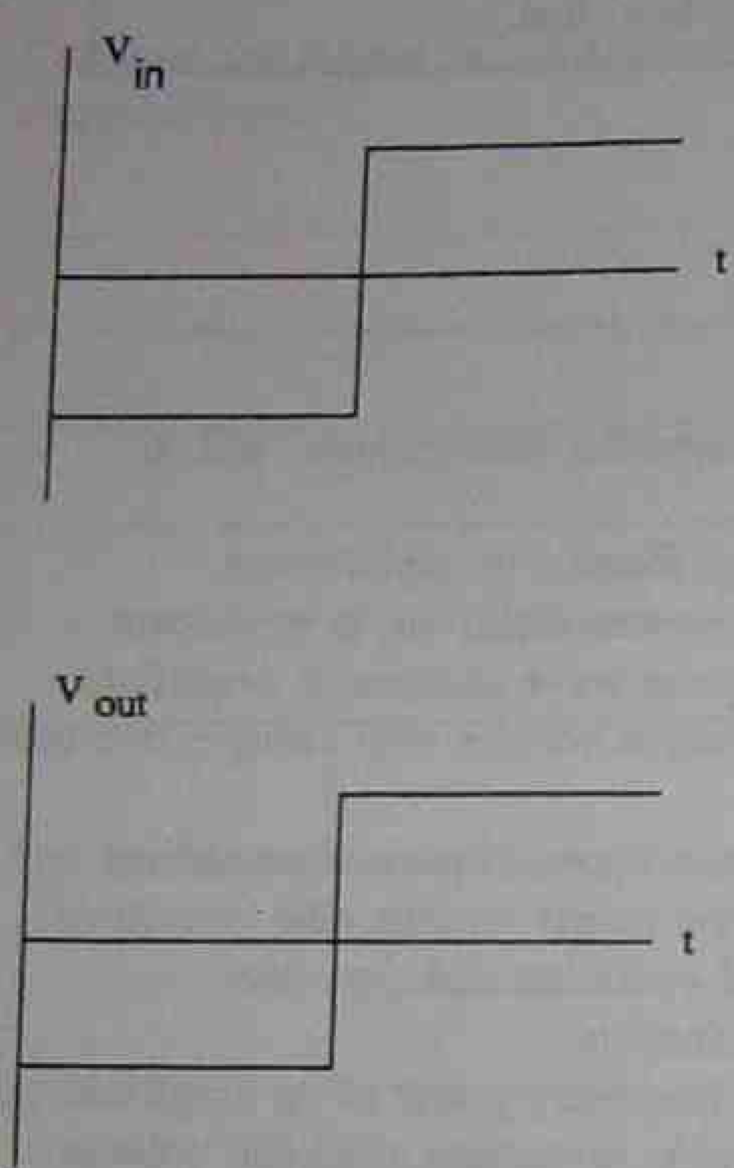


Fig. 1 (Ideal output)

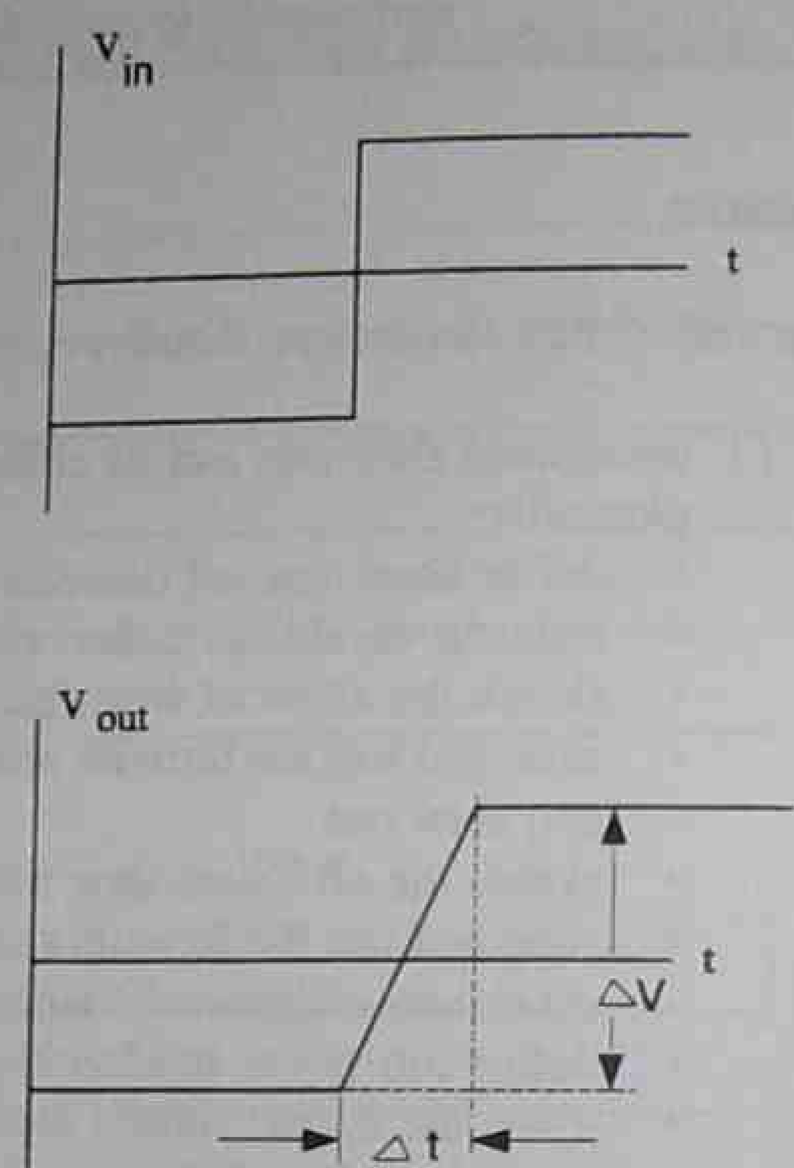


Fig.2 (Slew rate limited)

The non-zero 'catch-up' time required by the output is due to a characteristic called the *slew rate*. The slew rate of the amplifier is the maximum rate of change of the output voltage. In other words, the output cannot rise or fall faster than the slew rate.

In Figure 2, the output is rising as fast as it can, so

the slew rate $= \Delta V / \Delta t$ (Volts / μs) Equation 1
= change in output voltage / rise time

Note that usually the time unit in the denominator is μs, not seconds. For example, usually we write 0.5 V/μs rather than 500 000 V/s.

Cause of slew rate

There are some small internal capacitances in any amplifier. In some amplifiers, we have to connect small external capacitors to ensure proper operation. These capacitances have to charge up to the correct voltages before the output can settle down. It is the non-zero charging time of the capacitances that causes the slew rate.

Methods to improve slew rate

A high slew rate is usually desirable, because it means that the output can respond quickly to a change in the input. However, you should be aware that a high slew rate is sometimes accompanied by ringing. This means that the output overshoots and undershoots around the final value for a while before settling down. Methods which eliminate ringing usually also reduce the slew rate.

One obvious way to have a high slew rate is to choose an amplifier with a high slew rate specification. The common 741 op amp has a rather poor slew rate of only 0.5 V/μs. Other direct replacements for the 741 such as the LF351 have much higher slew rates (about 10 V/μs). There are high speed op amps which have slew rates ranging from 50 V/μs to more than 1000 V/μs.

In some 'externally compensated' op amps, the 'compensating' capacitor C_c (which affects the slew rate) can be chosen by the user, subject to some criteria which we discuss in a later section. With such op amps, quite large slew rates can be obtained at small expense.

Effect of slew rate on square wave response

Figure 2 shows the effect of slew rate on a square wave output, which is distorted and becomes a trapezoid shape. If the input frequency is high enough, the slew rate limited output never actually reaches the flat part of the square wave before the input steps to the other level; so the result is a triangular wave. However, even if the square wave frequency is low, it will still have two sharp edges and these will be slew rate limited at the output, though the distortion will be less noticeable if you see it in the CRO.

It is important to understand that slew rate limitation has its greatest effects only on large and/or high frequency output signals. Small output signals also have a rise time but this is due to bandwidth limitation, which we study later. We will postpone discussion of what is meant by 'large' or 'small' signals. As a rule of thumb, if the output waveform due to a step looks like Figure 3 or Figure 4, then it is small signal (bandwidth limited). If it looks like Figure 5, then it is a large signal (slew rate limited).

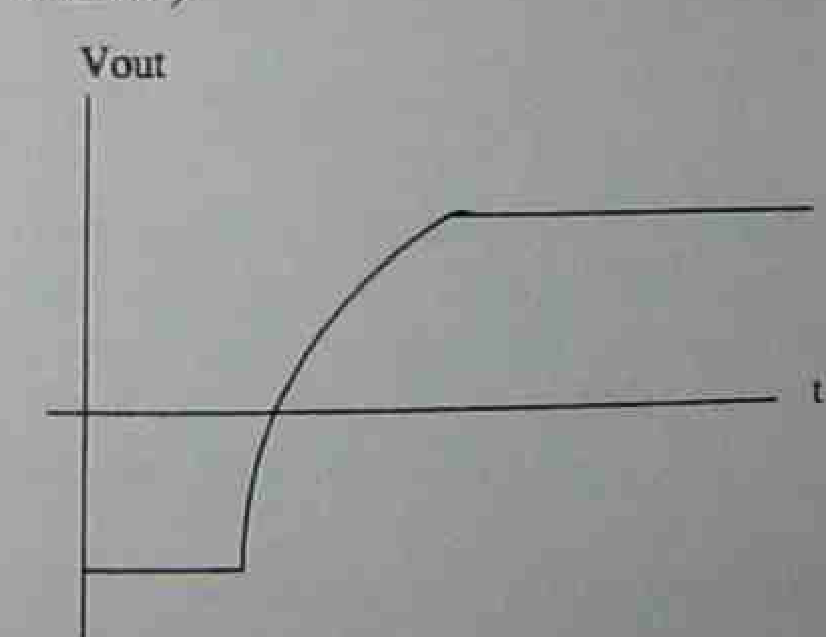


Fig.3

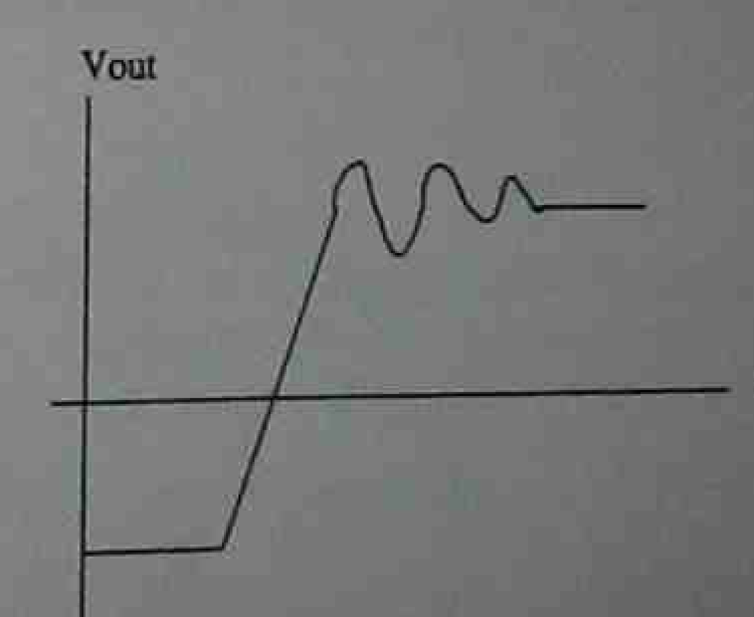


Fig.4

Small signal (bandwidth limited) square wave outputs

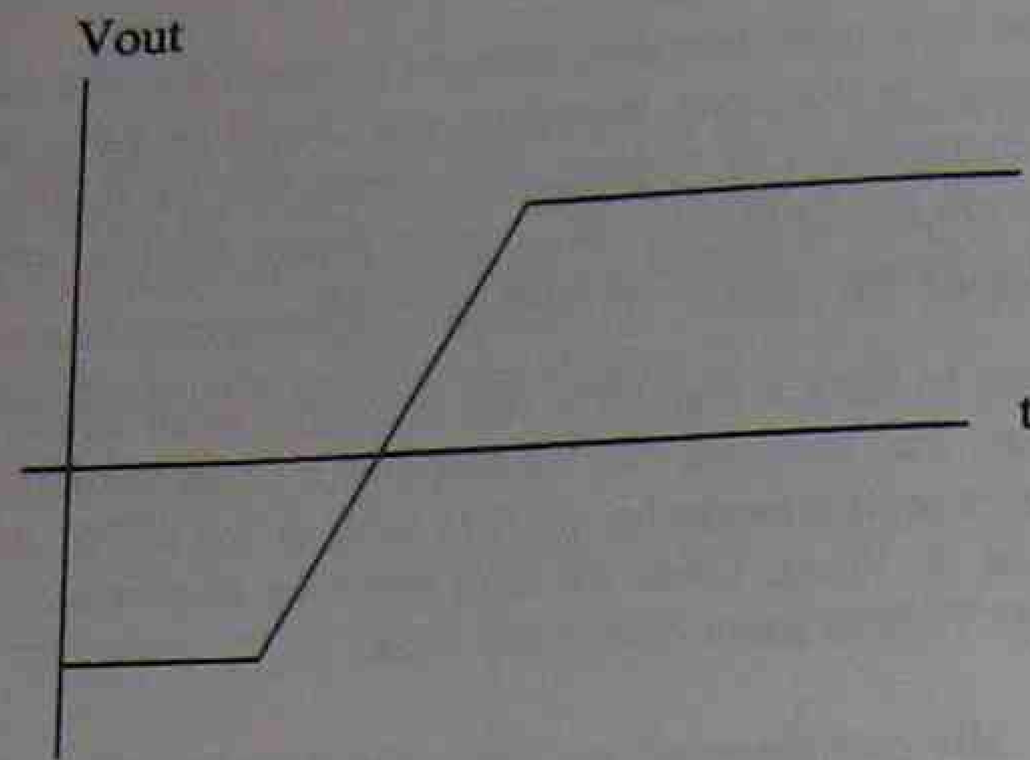


Fig.5. Large signal (slew Rate limited) square wave output

Effect of slew rate on sine wave response

If a sine wave has large amplitude and/or frequency, it is possible that the rate of change of some parts of the sine wave, especially near the zero crossing, will exceed the slew rate and thus cause distortion. This is shown in Figure 7 and 8. Figure 6 shows the undistorted sine wave output.

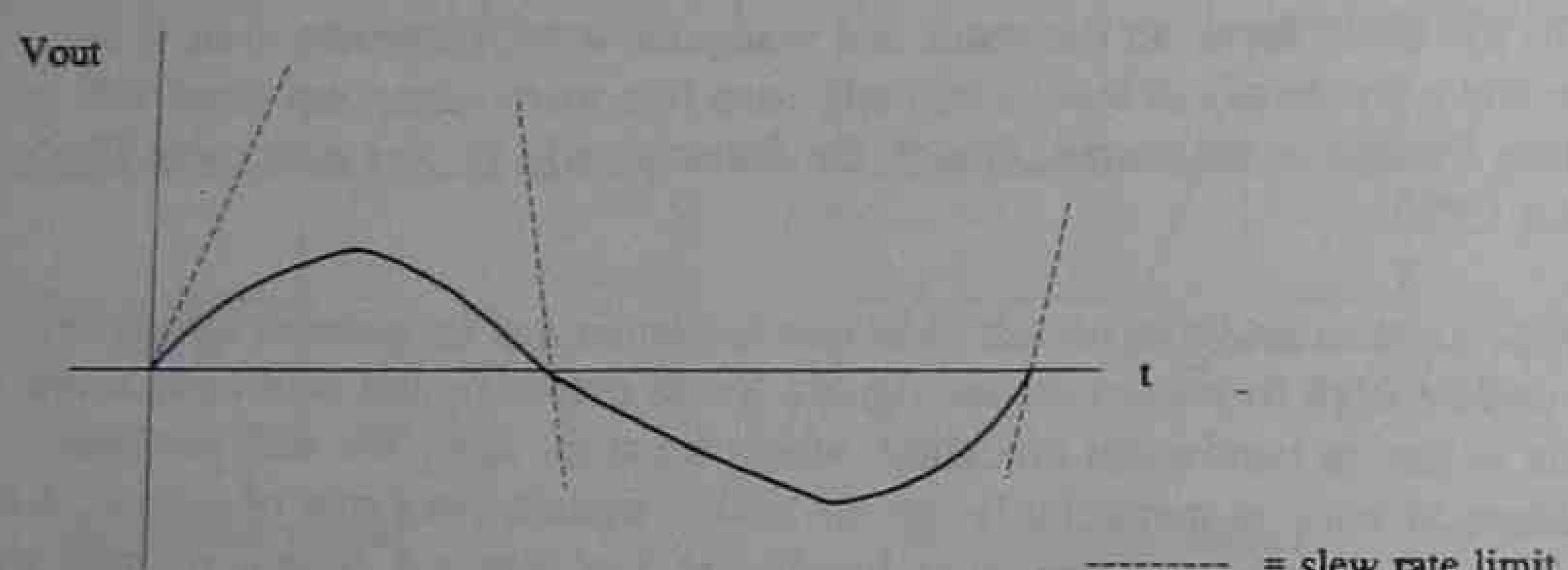


Fig.6 Undistorted sine wave output

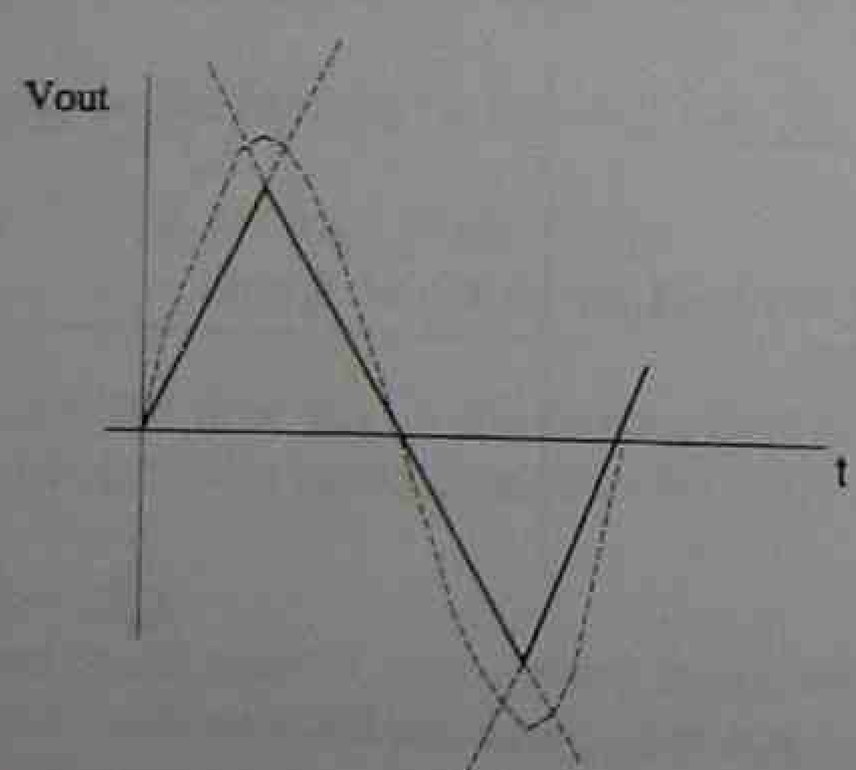


Fig. 7 High frequency sine waves are slew rate distorted

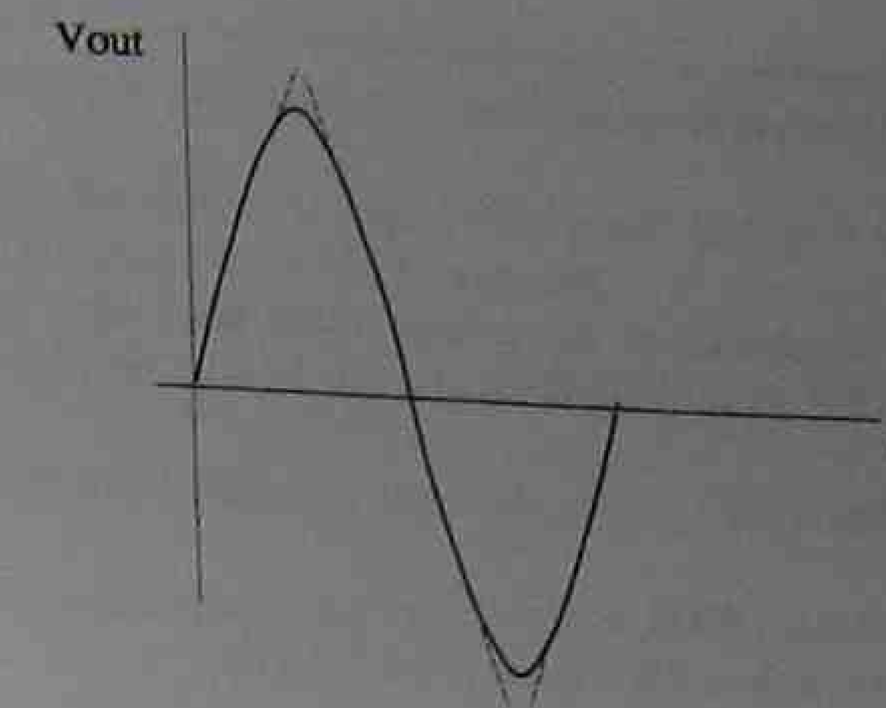


Fig.8. High amplitude sine waves are slew rate distorted

Again, as in the case of square waves, we see that slew rate is a large signal and high frequency problem. The formula relating the slew rate SR, the output *amplitude* (i.e. 0 to peak) $V_{o\ amplitude}$ and the maximum allowable frequency without suffering slew rate distortion f_{max} is:

$$SR = 2 * \pi * V_{o\ amplitude} * f_{max} \qquad \text{Equation 2}$$

This equation simply shows that the maximum achievable amplitude and maximum achievable frequency, without slew rate distortion, are inversely related. If you need a large output voltage swing, you have to keep the frequency low. If you have to work with large frequencies, you have to keep the amplitude low.

If an amplifier is required to output sine waves of 12 V rms at 20 kHz the minimum slew rate it must have is

$$2 * \pi * 12\sqrt{2}\ V * 20\ kHz = 2.13\ V/\mu s$$

The time taken by a comparator with a slew rate of 1.2 V/µs to switch from +12 V to -12 V is:

$$24\ V / (1.2\ V/\mu s) = 20\ \mu s$$

Full power bandwidth and output voltage swing graph

The full power bandwidth (FPBW) of an amplifier is the largest sinewave frequency without causing slew rate distortion, when the output amplitude is the maximum unclipped value (about one volt less than the supply voltage). The FPBW can be calculated using equation 2.

$$FPBW = SR / (2 * \pi * v_{o\ max\ ampl}) \qquad \text{Equation 3}$$

Example : Full power bandwidth

Question

Calculate the full power bandwidth of an op amp which has a slew rate of 0.8 V/μs and works with ± 15V power supplies.

Solution

Max output amplitude = 15-1 = 14V

Full power bandwidth = 0.8 V/μs /(2 * π * 14V)

= 9.1 kHz

Some op amp data sheets give a graph of 'maximum output voltage swing as a function of frequency'. A typical graph (for the 741 op amp) is shown in Figure 9.

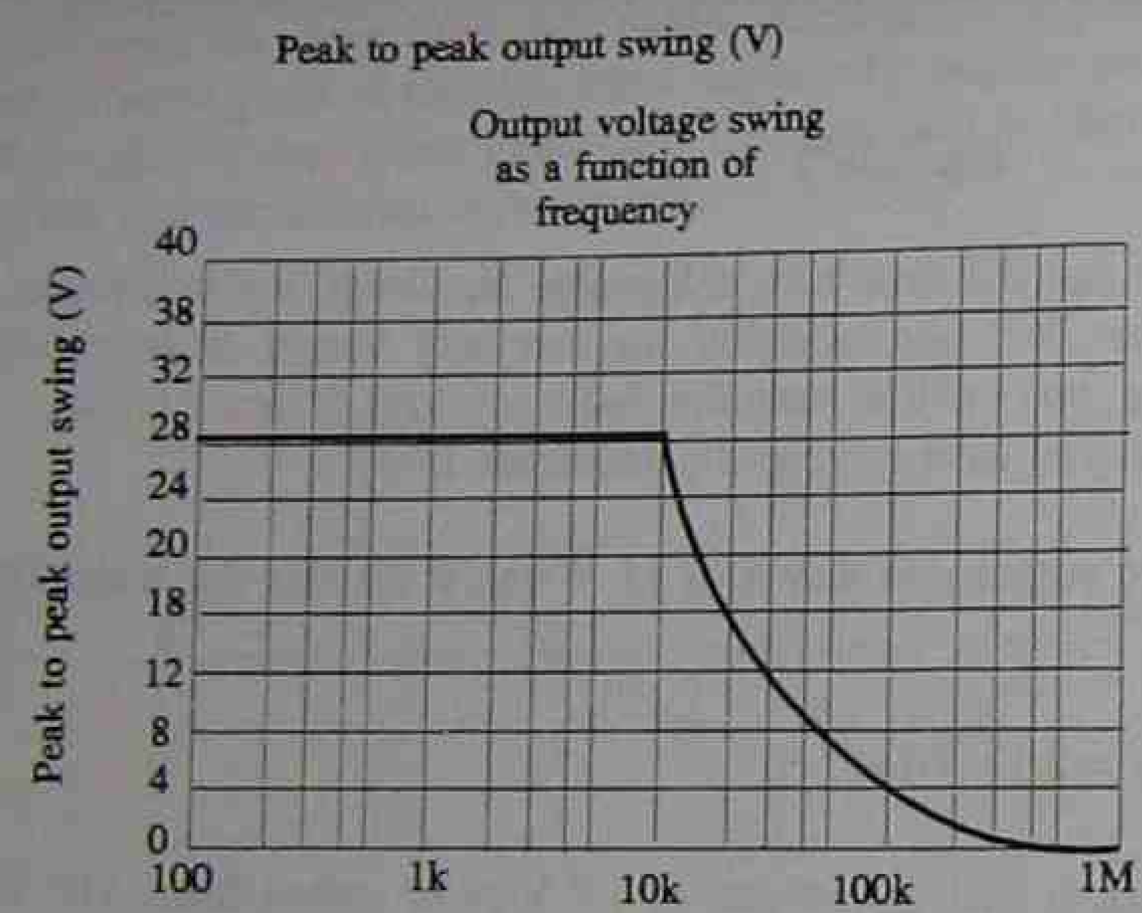


Fig. 9. Output voltage swing graph of the 741 op amp

The flat part of the graph (up to 10 kHz) shows that the maximum output (about 28V peak to peak or 14V amplitude) is limited by the DC supply voltage (clipping limited). At higher frequencies, slew rate limitation takes effect and the maximum undistorted swing is reduced (as in equation 2). The frequency where the two curves meet (10 kHz in Figure 9) is the full power bandwidth.

Take care in reading the graph in Figure 9. In some data sheets, the Y-axis may give the peak-to-peak output voltage; in others, the amplitude. The X-axis is a log frequency scale, whose markings may go 1,2,3,4,5,10,20..... The markings for 6,7,8,9 may not be shown, to keep the graph uncluttered.

Example 2 : Maximum undistorted output voltage calculations

An amplifier with the output voltage swing graph in Figure 9 is wired up with a voltage gain of 10, using ± 15 V DC supplies.

(a) What is the slew rate of the amplifier ?

(b) What is the maximum rms input voltage at 1 kHz without output distortion ?

(c) What is the maximum rms input voltage at 20 kHz without output distortion ?

Solutions

(a) FPBW = 10 kHz

∴ SR = 2 * π * (28V/2) * 10 kHz

= 0.88 V/μs

(b) From the graph, at 1 kHz, max. $v_{o\,p\text{-}p}$ = 28V

∴ max v_i rms = 28V / (2 √ 2) / gain = 1 V rms

(The output is clipping limited below FPBW)

(c) From the graph, at 20 kHz, max. $v_{o\,p\text{-}p}$ = 14V

max v_i rms = 14V / (2 √ 2) / gain = 0.5 V rms

The output is slew rate limited above FPBW

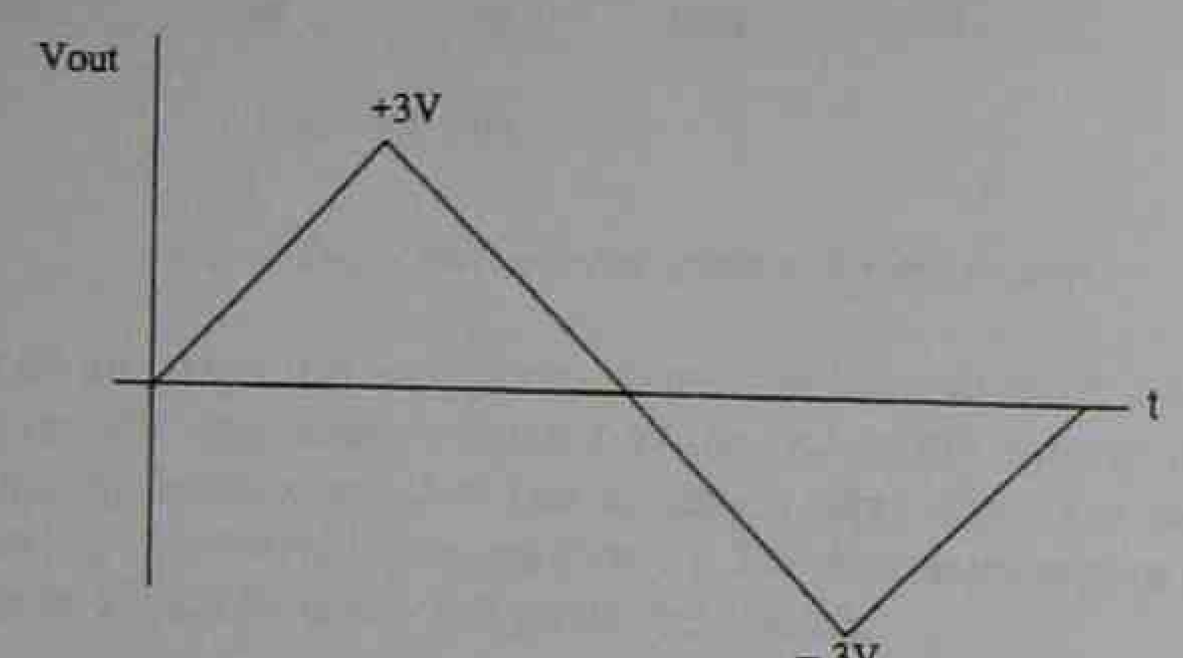
(Note : you could have calculated the answer to part (c) using the slew rate and equation 2. The answer would be about the same, except for the human error in reading graphs.)

Review questions

These questions will help you revise what you have learnt in Section 3.

1. An operational amplifier with a slew rate of 0.5 V/μs is connected as a voltage follower and a square wave input at 62.5 kHz, 6V peak to peak is applied. Sketch the output waveform, carefully marking important time and voltage values.

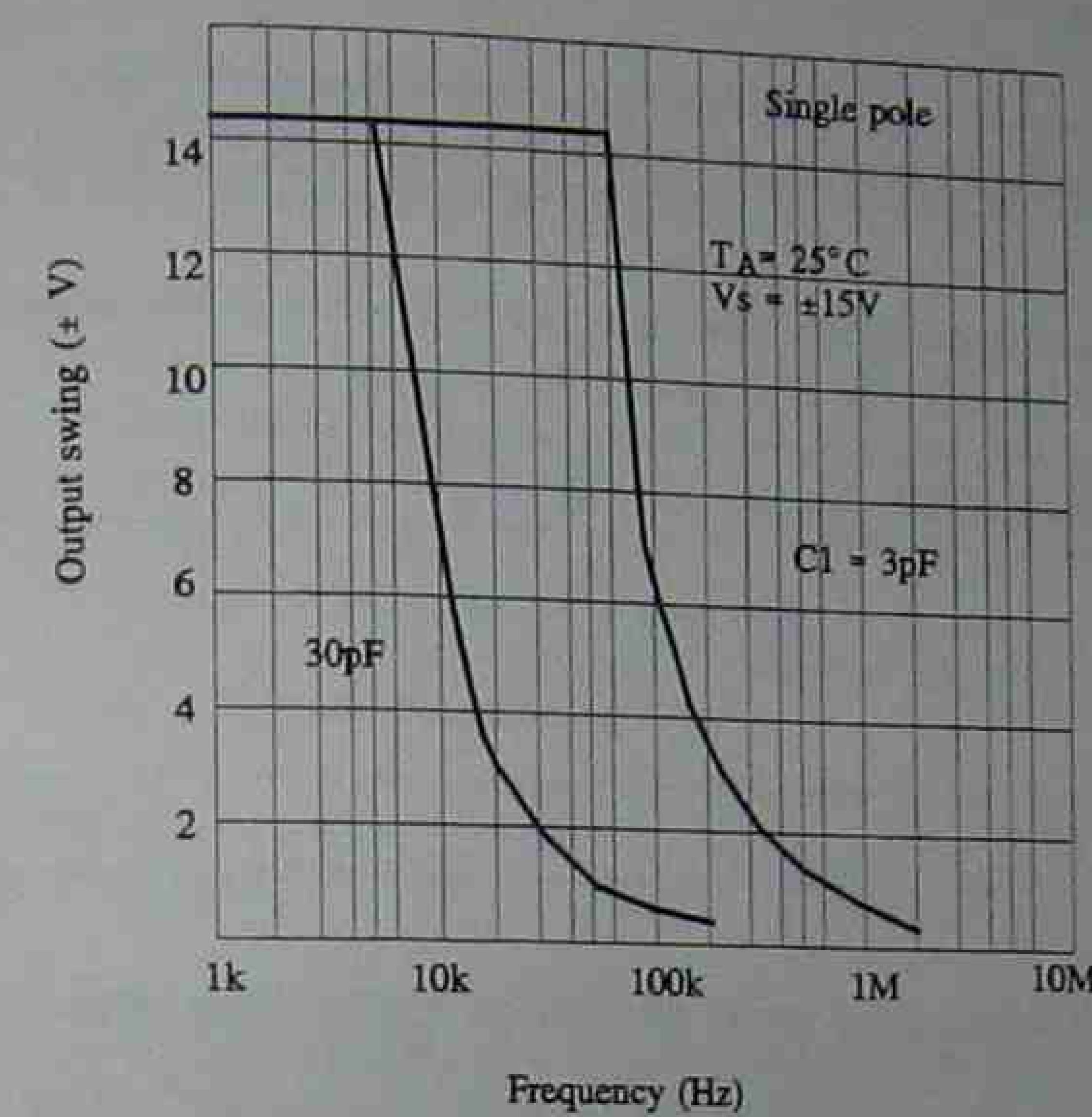
2. An amplifier has a sinewave input at 100 kHz and a triangular wave output as shown below. What is the slew rate of the amplifier ?



____

____

3. The output voltage swing graph of an externally compensated operational amplifier is shown below.



(a) What is the FPBW and slew rate for C_1 = 30pF?

____

____

(b) What is the FPBW and slew rate for C_1 = 3 pF?

____

____

(c) If C_1 = 3 pF is used, determine from the graph the maximum undistorted output amplitude for a sinewave input at 300 kHz.

____

____

(d) Using equation 2 and the slew rate calculated in part (b), verify your answer to part (c) by calculation.

4. Explain the significance of the terms 'slew rate' and 'full power bandwidth'.

5. State any **two** methods which can improve the slew rate performance of an amplifier circuit.

-
-

6. Check the data sheets of several op amps and name one type which has a slew rate of at least 50 V/μs.

Skill practice 4
Slew rate

Suggested duration

1 hour 30 minutes

Tasks

- To measure the slew rate of an op amp.
- To observe the effect of changing the compensating capacitance on slew rate in an externally compensated op amp.

Equipment

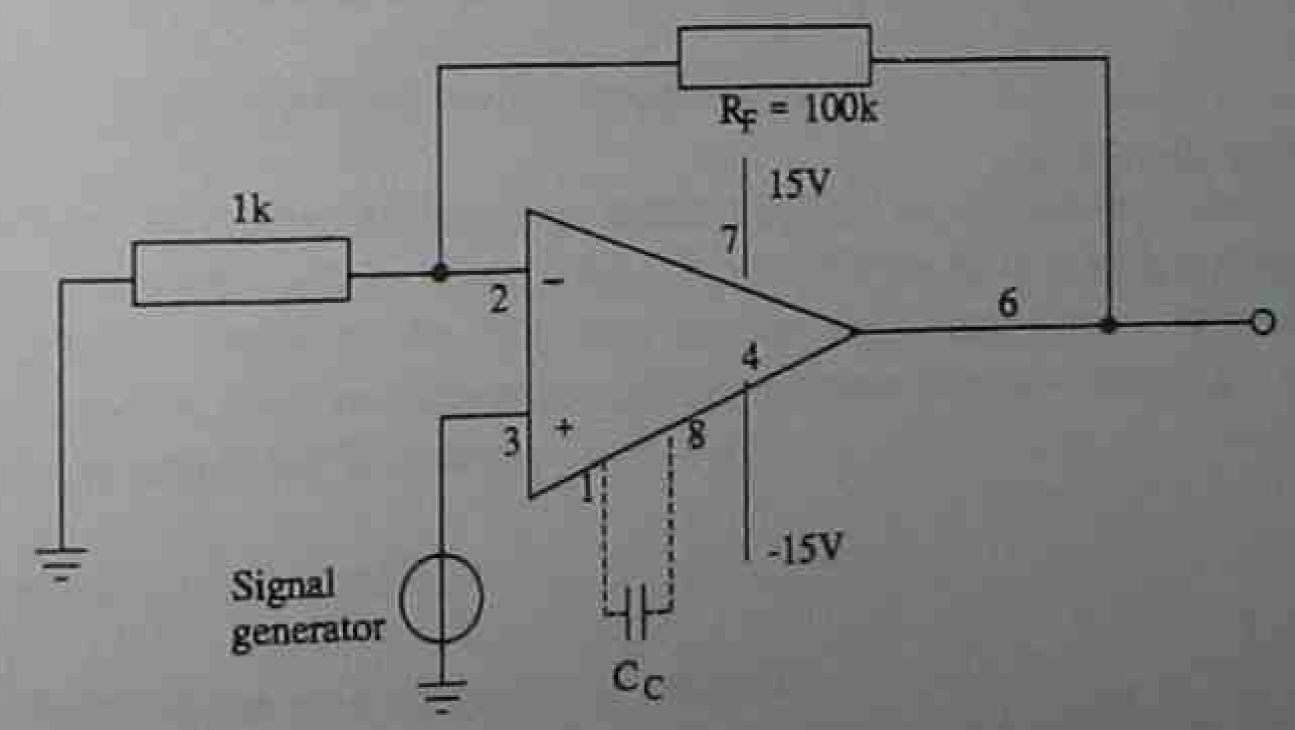
- Either one op amp type LM301 and a 30 pF capacitor OR one op amp type 741. (741 is not suitable for Step 6.)
- General purpose 15 MHz oscilloscope (one with time-base x5 or x10 switch preferable)
- AC signal source (e.g.function generator)
- Dual DC power supply (fixed ±15V supplies can be used)
- Connectors, breadboard, sockets etc.

Procedure

Step 1 Setup

- Set the power supplies to ± 15V.
- Set the function generator to square wave, about 5 kHz, about 2 V peak to peak.
- Signal frequency and voltage are not critical.

- Connect the circuit shown below. (Do not forget to connect the power supply common to circuit ground.)
- If you are using 301 op amp, connect a 30 pF capacitor across pins 1 to 8. This is the compensating capacitor C_c mentioned in the lesson. If you are using the 741 op amp, this capacitor is not necessary.



Step 2 Output waveform

- Sketch the output waveform.

Step 3 Measurements

- Measure the rise in voltage and rise time. (If your CRO has a time-base x5 or x10 switch, rise time measurement gets more accurate - ask your teacher to show you how).

Rise in voltage =

Rise time =

Step 4 Slew rate calculation

- Calculate the slew rate (equation 1) and compare it with the data sheet specification.

Slew rate (experimental) =

Data sheet specification :

Step 5 Effect of changing the compensation capacitor

- If you are using the 741 op amp, skip this step. If you are using the 301 op amp, remove the 30 pF capacitor and R_F and repeat steps 2, 3 and 4 above.
- Sketch of output waveform:

Rise in voltage =

Rise time =

Slew rate (experimental) =

Step 6 Full power bandwidth measurement

If you are using the 301 op amp, reinsert R_F and the 30 pF capacitor. Change the signal generator to sine wave. Adjust input voltage until output just begins to clip. Observe the output on the CRO and vary the frequency from 1 kHz upwards until it starts to show slew rate distortion (flattening near the zero crossing - see Figure 8 in notes). Measure the following.

Output sine wave amplitude :

Maximum undistorted frequency of operation :

Theory calculation of f_{max} (equation 2) =

Percentage error in measurement :

Discussion questions

1. How does your measured slew rate compare with the data sheet value ? What is the percentage error?

2. If you did Step 5 above, what is the effect of reducing the compensation capacitor on the slew rate?

Section 4: AC Noise

SUGGESTED DURATION	PREAMBLE
9 hrs 30 mins	To enable you to analyse the noise performance of op amp circuits and recognise methods for improving it.

Objectives

At the end of this section you should be able to:

☐ understand the noise model of op amps and in particular:

- state the meaning of 'noise' as applied to electronic circuits and various common sources of noise
- state the meaning of the terms 'noise spectrum', 'noise density', 'white noise' and 'pink noise'
- interpret the input noise model for an amplifier and state the meaning of 'total input equivalent noise voltage'

☐ calculate the source noise resistance of common amplifier circuits (including those studied in Section 1, as well as summing amplifier and differential amplifier configurations)

☐ use the broadband noise graph of an operational amplifier to estimate the total equivalent input noise voltage

☐ use the noise rms addition formula to find the total equivalent noise rms voltage or current given several independent noise voltages or currents

☐ calculate the noise rms voltage or current over a given bandwidth, given the noise density specification (either as volts squared per Hz or volts per square root Hz, and similarly for current)

☐ write the formula for the total equivalent input noise voltage and calculate it

☐ calculate the noise gain of common amplifier circuits and the output noise

☐ define 'signal to noise ratio' (SNR); calculate input SNR and output SNR given a circuit, the amplifier's noise specifications and the input signal

☐ calculate the effective noise bandwidth when bandlimiting is done by a first order (RC) or by a high order (order ≥ 5) low pass filter

☐ recognise common methods to minimise external and internal noise in amplifiers

- ☐ measure/estimate a noise voltage using a
  - true RMS voltmeter
  - digital multimeter
  - CRO
- ☐ measure the input noise voltage of an amplifier for a given bandwidth and source resistance.

References

The following references deal with topics in this section.

1. Jacob (1993), pp 199-203
2. Rutkowski (1994), pp 106-108
3. Gayakwad (1993), pp 197-199
4. Coughlin & Driscoll (1993), pp 264-267

Noise

'Noise' is any unwanted voltage or current at the output of the system. Commonly, noise is thought of as added to the signal and unrelated to it.

Sources of noise in amplifiers and noise model

External noise

The output noise in amplifiers can be due to external or internal sources. External noise can be picked up by incoming signal leads (or communication channels) - for example, the 50 Hz hum from AC power lines, cosmic noise, broadcast signals, DC power supply variations, switches, fluorescent lamps, motors. External noise can sometimes be minimized by careful circuit construction, shielding and earthing. Though external noise is important, we shall not consider it further in this section.

Internal noise

The amplifier also generates noise internally. There are four main sources of internal noise.

1. Noise voltage caused by external resistors (e_R).

   You have learnt Ohm's Law, which says that the current I in a resistor is equal to the voltage difference across the resistor V divided by the the resistance R. However, the electron flow which makes up this current is not quite smooth or uniform. At any instant, a few more or a few less electrons than average reach the positive voltage terminal. This random variation in electron flow shows up as a noise voltage.

   Resistive noise is also called 'thermal' or 'Johnson' noise.

   e_R is the total noise voltage generated by the external resistors which make up the amplifier (R_1, R_F, R_c etc). We shall soon present and use the formula for calculating e_R.

2. Similarly, the resistances inside the amplifier generate noise voltages.

3. The amplifier generates noise current called 'Schottky' or 'shot' or 'recombination' noise. This is caused by the non-uniform, random rate at which the positive and negative charge carriers in the semiconductor devices recombine.

4. The amplifier also generates noise current called 'flicker' noise. This noise is also called '1/f' noise because its power increases with decreasing frequency.

Noise model for operational amplifiers

Since the noise components generated by the operational amplifier are impossible or difficult to calculate, it is common practice to 'lump' or aggregate the internal noise into one single effective noise voltage source and one single effective noise current source, and bring them out to the input. This is very similar to the approach taken with the DC input offset voltage. This gives the noise model for the amplifier as shown below in Figure 1.

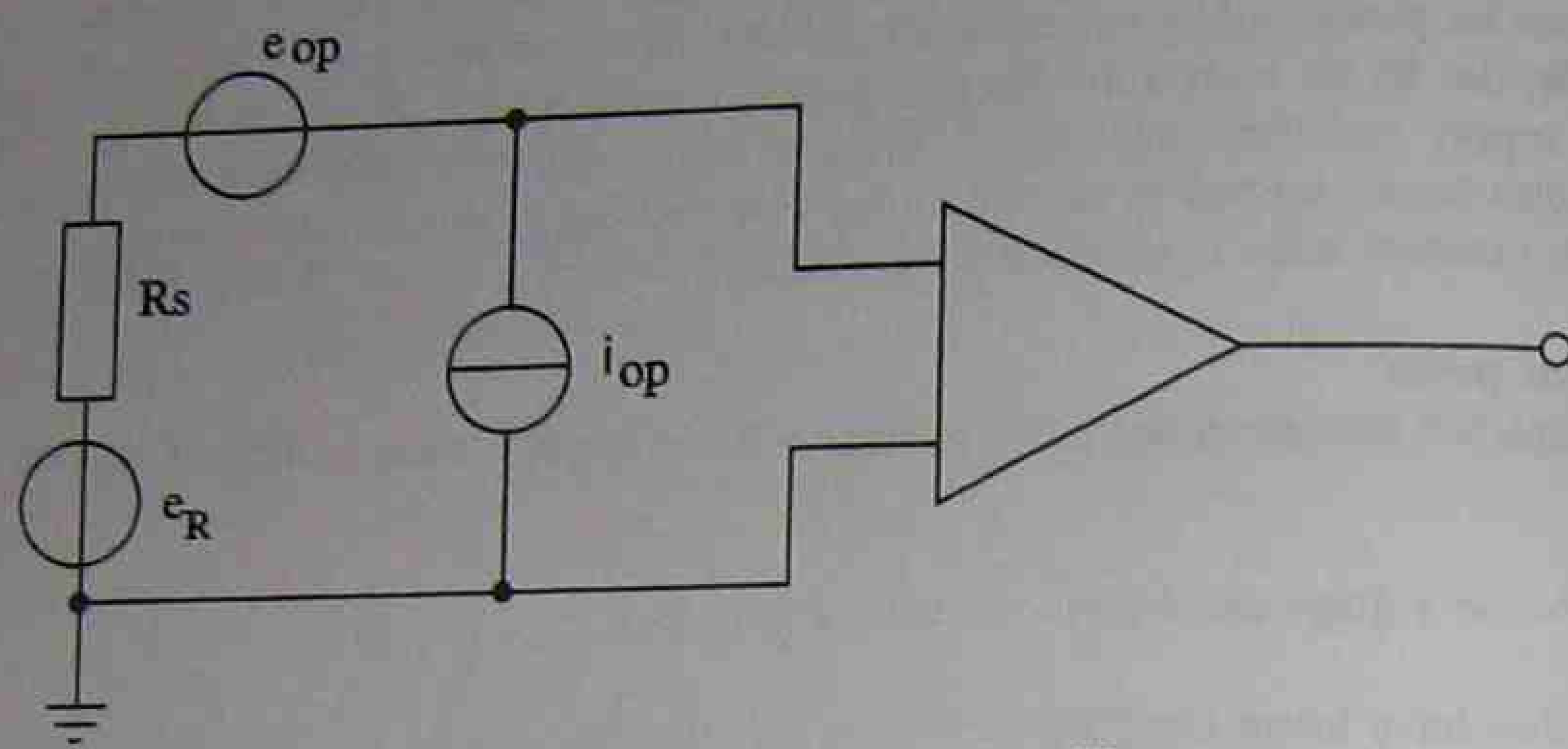


Fig. 1 Noise model for operational amplifiers

R_s = source noise resistance (equivalent to all the externally connected resistors in the circuit)

e_R = thermal noise voltage source due to R_s

e_{op} = equivalent input noise voltage due to the op amp

i_{op} = equivalent input noise current due to the op amp

Note that:

- there are only three sources in the circuit above (not four)
- we keep e_R and e_{op} separate, even though they are in series, because the former is affected by our choice of components, while the latter is a property of the op amp
- e_{op} and i_{op} do not really exist at the input or anywhere else. They only represent the effect of the actual noise inside the amplifier.

Next we study how to combine the effects of the three sources given above into a single equivalent input noise voltage source. Then we study the actual mechanics of calculating the equivalent input noise voltage using formulas and/or the data sheets of the op amp.

Description and calculation of noise

Since noise is usually random and unpredictable, we cannot write any equation for noise voltage or current, nor sketch it with reasonable accuracy. However, it is a fact that the average *power* in a load due to any realistic noise source is fixed. Since power (from any kind of voltage or current source) is proportional to (rms voltage)^2 or (rms current)^2, we can say that the rms voltage or rms current of a noise source (working into a given load) has a fixed and definite value.

Noise voltage or current is always expressed as a rms value.
Note that you cannot use the conversion factor of 1.414 to convert noise rms voltage to noise peak voltage. This factor applies only to sine waves and has no meaning for noise.

Though a single rms value is a convenient description of noise, it does not tell the whole story. Since noise varies randomly, it can be usually expected to have a large spread of frequency components. The *noise power density spectrum* is a graph showing the distribution of the total noise power over frequencies, or the power per Hz of bandwidth as a function of the centre frequency where the power is measured.

The shape of the noise power density spectrum depends on the physical origin of the noise. Some common shapes are:

- constant with frequency (called white noise)
- inversely proportional to frequency (called pink noise) and
- lines at specific frequencies.

Thermal noise and shot noise are white. Flicker noise is pink. The hum picked up from power lines has a line spectrum, since all their power is concentrated at 50 Hz and its harmonics.

An important point to note about noise power (and so, the noise rms voltage) is that it depends strongly on the circuit bandwidth, which is under the control of the circuit designer.

For white noise (which is the most common kind of noise), the noise power is proportional to the bandwidth. Therefore, data sheets often specify the noise density rms volts squared (or noise rms current squared) per Hz and leave it to you to calculate the total noise volts squared over your circuit bandwidth.

Other data sheets give noise density rms volts per square root Hz (or noise rms current per square root Hz). Of course, the square root of Hz is not a physically meaningful unit. It only serves to remind you that the noise rms voltage or current is proportional to the square root of the bandwidth.

Total noise volts over a bandwidth

$$= \sqrt{(noise\ density\ volts^2/Hz) * bandwidth} \qquad \text{Equation 1}$$

or

$$= (\ noise\ density\ volts/\sqrt{Hz}\ ) * \sqrt{bandwidth} \qquad \text{Equation 2}$$

The same rule works for noise current.

Note : This rule applies only to white noise. However, white noise is the most common kind in wideband amplifiers. The other common kind, pink noise, is important only at low frequencies, in which case noise is probably not a major problem anyway.

Some examples showing the calculation of noise voltage and current follow.

Example 1
If an op amp is stated to have a noise voltage density of 15 $nV/\sqrt{Hz}$, its rms noise voltage over a bandwidth of 30 kHz
$= 15\ nV/\sqrt{Hz} * \sqrt{30kHz} = 2.6\ \mu V$ rms.

Example 2
The noise current density of a 741 op amp is specified as $3 * 10^{-25}\ A^2/Hz$. Its rms noise current over a 25 kHz bandwidth = $\sqrt{3E-25A^2/Hz*25kHz} = 87\ pA\ rms.$

Example 3
The noise voltage of an amplifier is measured to be 6.5μV over a 40 kHz bandwidth. If the bandwidth is changed to 15 kHz, the noise voltage is estimated as
$6.5\mu V * \sqrt{15kHz/40kHz} = 4\mu V\ rms.$

Addition of noise voltages and currents

Normally the noise in any system results from the addition of noises from many different sources. The noises are random and without any relationship to each other. They are uncorrelated or non-coherent. Uncorrelated signal *voltages or currents* cannot be simply added to get the total. However using advanced mathematics, it can be shown that uncorrelated noise *powers* can be added. This means that noise volts squared (or noise currents squared) can also be added to get the total. After adding the squares, we can take the square root to get the rms value of the noise voltage (or current).

$$e_{total}^2 = e_1^2 + e_2^2 + e_3^2 + \ldots\ldots \qquad \text{Equation 3}$$

or

$$e_{total} = \sqrt{e_1^2 + e_2^2 + e_3^2 + \ldots\ldots}$$

If an amplifier has external noise voltage of 4.5μV and internal equivalent noise of 7μV at its input, the total equivalent noise at the input e_{total}
$= \sqrt{(4.5\mu V)^2 + (7\mu V)^2} = 11.5\mu V$

Note that the answer is *not* (7 + 4.5) μV = 11.5μV.

Ohm's Law is unchanged for noise. The noise voltage v_n caused by a noise current i_n flowing through a resistance R is given by $v_n = i_n * R$.

Total equivalent input noise of an amplifier

Refer to the noise model of the amplifier in Fig. 1. The total equivalent noise at the input is total of the effects due to (i) the thermal noise e_R of the source resistance R_s, (ii) the internal noise voltage of the op amp e_{op}, and (iii) the noise voltage caused by the internal noise current i_{op} flowing through R_s. Using the rules for adding noise voltages and Ohm's Law,

The total equivalent input noise voltage of an amplifier =

$$e_n = \sqrt{e_R^2 + e_{op}^2 + (i_{op}^2 * R_s^2)}\ V(rms) \qquad \text{Equation 5}$$

Example 4: Using the noise addition formula
In an operational amplifier circuit, the source noise resistance = 30 kΩ, the thermal noise due to the source resistance = 2.8μV, the internal noise current of the op amp = 60 pA and the internal noise voltage of the op amp = 4.1μV.

(a) What is the total equivalent input noise voltage e_n ?

(b) What will be the new value of e_n if the bandwidth is tripled ?

Solutions

(a) $e_n = \sqrt{(2.8\mu V)^2 + (4.1\mu V)^2 + (60pA * 30k\Omega)^2}$
$= 5.3\mu V$ (rms)

(b) Since $e_n \propto \sqrt{BW}$, new $e_n = 5.3 * \sqrt{3}\ \mu V$
$= 9.2\ \mu V$rms

Effective noise bandwidth

We have seen that the total noise voltage depends strongly on the circuit bandwidth. However, the bandwidth is just a convenient number, i.e. the frequency where the output has a relative attenuation of 3 dB, or 0.707 times the maximum output. Signals or noise at higher frequencies will also pass through, though at greater attenuation. This means that the total noise actually transmitted by the amplifier is more than what is calculated on the basis of the bandwidth. The situation is shown in Figures 2 and 3 below.

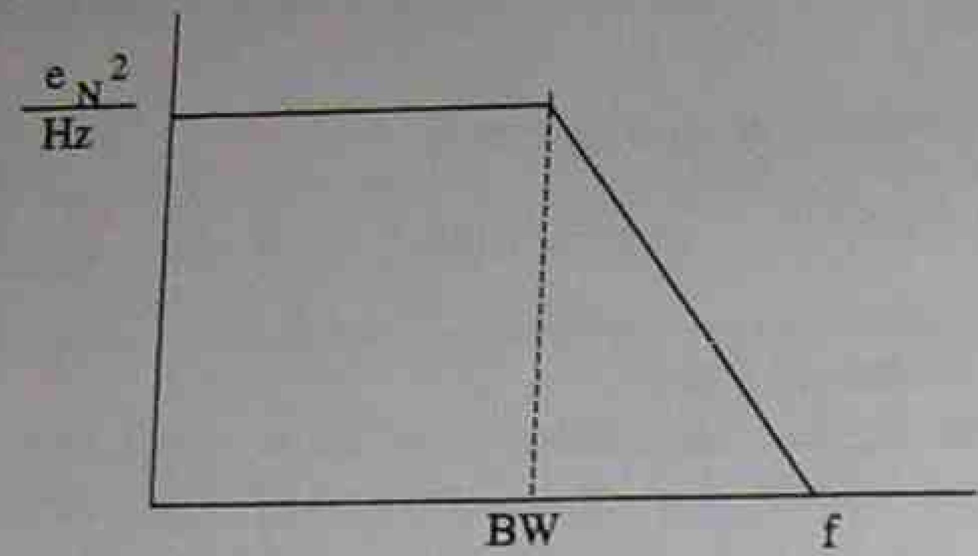


Fig. 2 Practical Bandlimiting Filter response

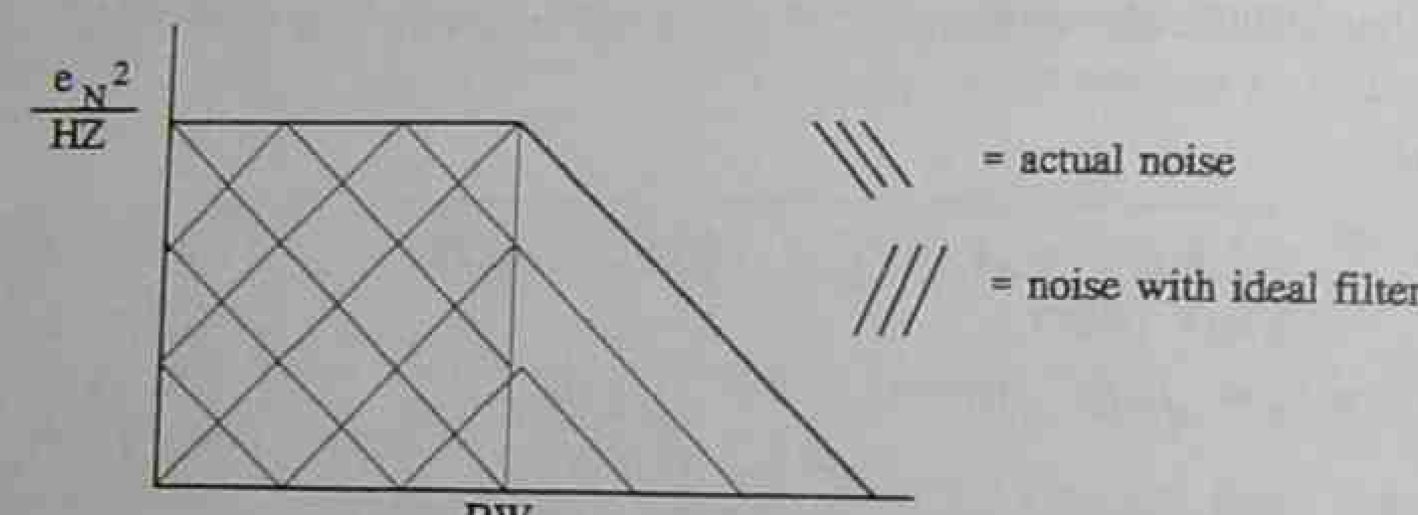


Fig. 3 Comparison of noise allowed in by ideal and practical bandlimiting filters

The extra noise due to the 'tail' of the filter response depends on its rate of roll-off. If the rate of roll-off is 20 dB/decade, then advanced mathematics shows that

Effective noise bandwidth = $\pi/2$ * (3dB BW) — Equation 6

Since noise voltage is proportional to the square root of the bandwidth,

$$\text{noise increase factor for bandwidth correction} = \sqrt{\frac{\pi}{2}}$$

Equation 7

Note that these formulae are valid only for first-order (or single pole, or simple RC) low pass filtered white noise. This is the most common situation. The filtering may be done by external components, or perhaps by the internal capacitances in the amplifier.

For high order filters (order ≥5) no correction is necessary because the rate of rolloff is steep and close to ideal.

Example 5 : Effective noise bandwidth

An amplifier has input noise of 15nV/√Hz. It is bandlimited by an RC filter with cutoff frequency of 20 kHz.

(a) What is the effective noise bandwidth?

(b) What is the actual input noise voltage?

Solutions

(a) Effective noise bandwidth = 20 kHz * $\pi/2$ = 31.4 kHz

(b) actual input noise voltage = 15 nV/√Hz * √31.4kHz
= 2.66 μV (rms)

Calculation of input noise resistance

Looking at the total equivalent input noise of an amplifier (equation 5), we see that we need the value of the input noise resistance of the amplifier, R_s.

R_s is the effective resistance (called the Thevenin resistance) between the +input and -input with all sources removed. This means that all voltage sources are short circuited, all current sources are open circuited and the output earthed. All the common negative feedback amplifiers can be simplified to the following form (Figure 4).

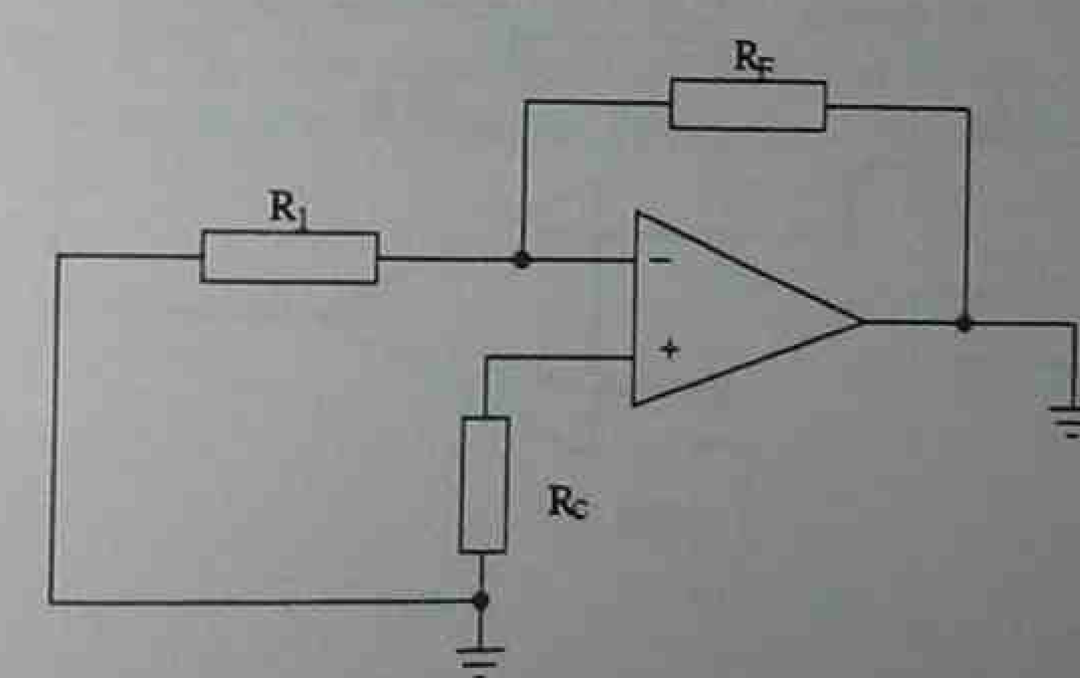


Fig. 4 Negative feedback amplifier

$\therefore R_s = R_c + (R_F \parallel R_1)$ — Equation 8

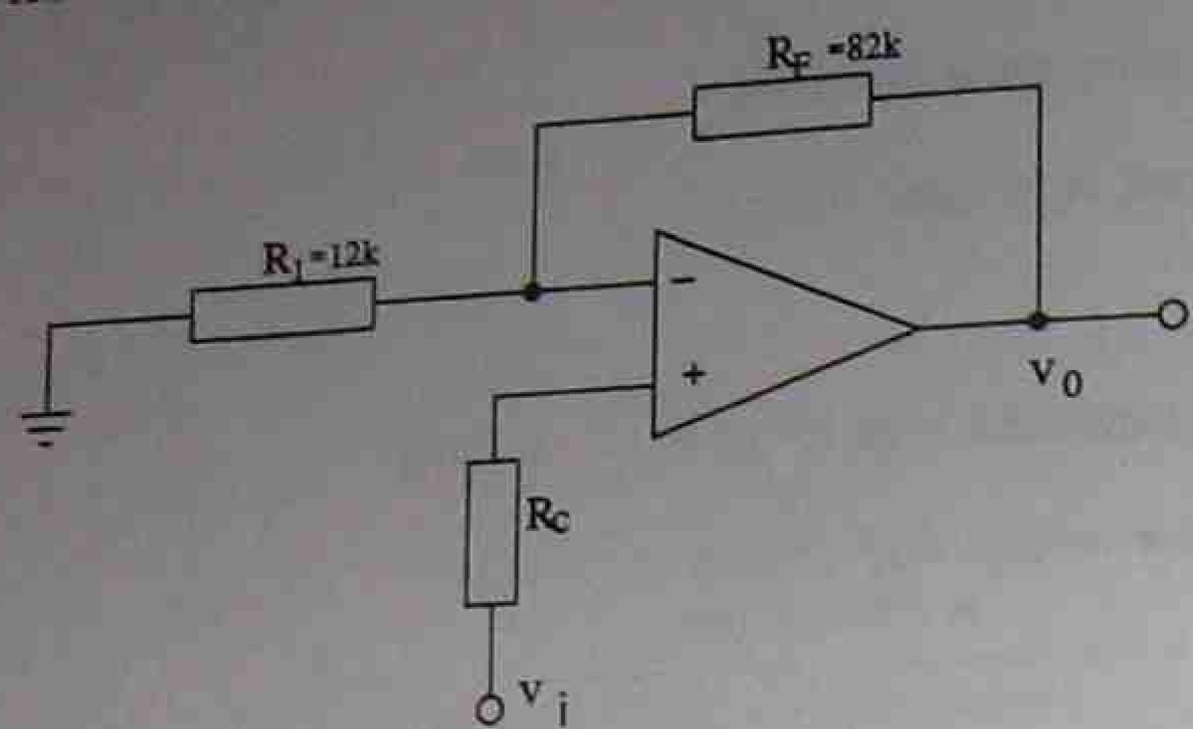
As before, R_c and R_1 are the effective resistances from the +input and -input to ground respectively, after simplifying any series and parallel combinations.(To be exact, the answer in equation 8 must be paralleled with the input resistance of the op amp. Practically, the input resistance is very large and does not affect the result.)

Example 6 : Source noise resistance

For the following circuit, calculate

(a) the value of R_c for bias current compensation

(b) the source noise resistance, R_s, for R_c = 10 kΩ



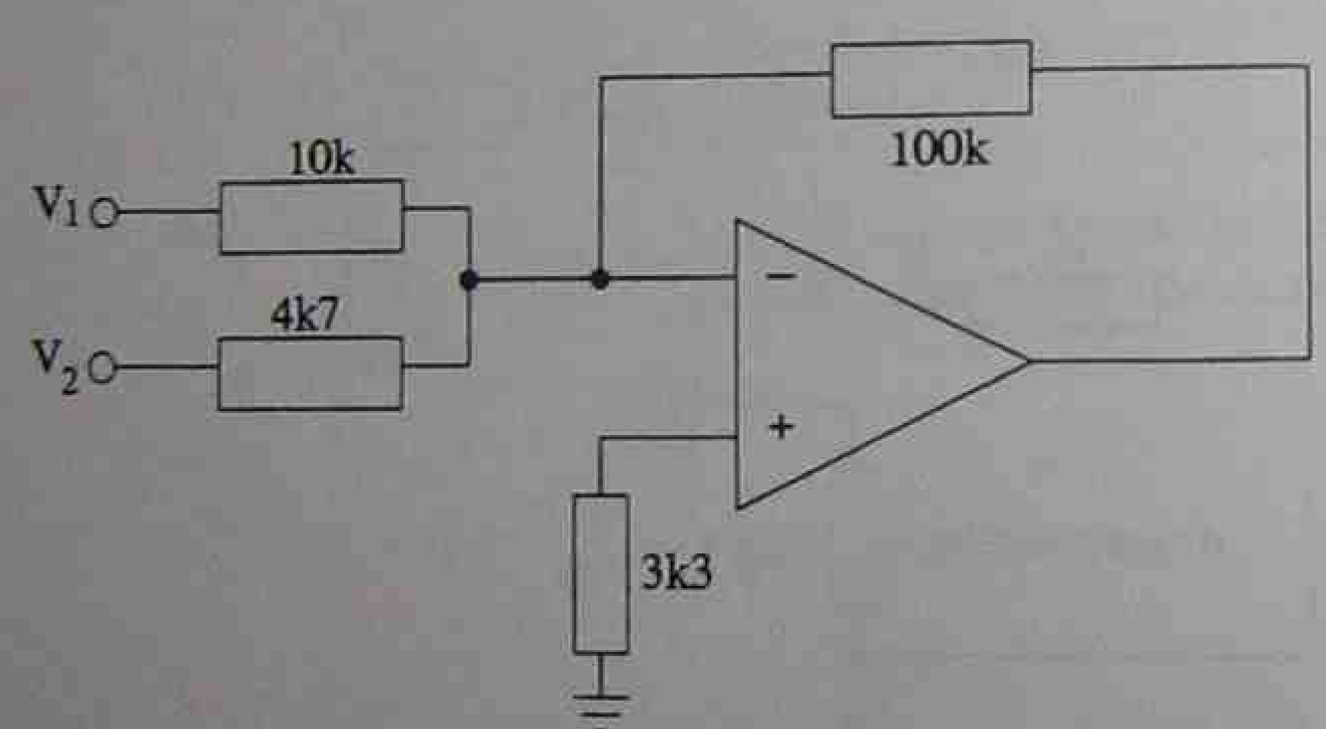
Solutions

(a) R_c = 12k ∥ 82k = 10.4 kΩ

(b) R_s = 10 k + (12k ∥ 82k) = 20.4 kΩ

Example 7 : Source noise resistance

For the following circuit, calculate the source noise resistance.



Solution

To calculate R_s, ground all inputs and the output. This makes 10k and 4k7 in parallel.

R_1 = 10k ∥ 4k7 = 3.2 k

∴ R_s = 3k3 + (3.2k ∥ 100k) = 6.4 kΩ

Determination of total equivalent input noise voltage using the broadband noise graph

The data sheets of many op amps give a 'broadband noise graph' which simplifies the calculation of the total equivalent input noise voltage (e_n.) This graph gives a set of curves giving e_n as a function of R_s for different bandwidths. So if we calculate R_s and know the effective bandwidth, we can read off e_n without further calculation. A typical broadband noise graph as found in the Fairchild linear data book appears below (Figure 5). This graph is for the type 741 op amp.

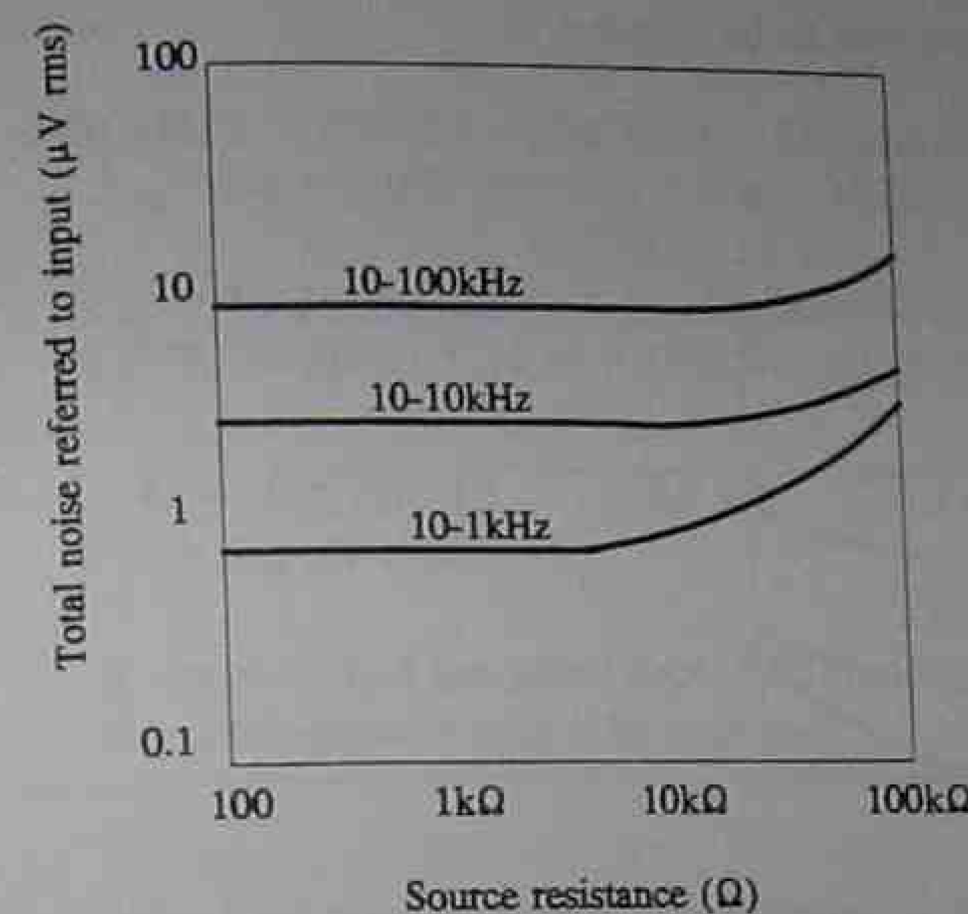


Fig. 5 : Broadband noise graph for 741 op amp

Some care is needed to use this graph. The X and Y axis are log scale and they go *1,2,3,4,5,10* (6,7,8 and 9 are not shown). The bandwidths are actual bandwidths, not ranges. For example, the curve for '10-100 kHz' means 'for cutoff frequencies 10 Hz and 100 kHz'. If the actual circuit has a bandwidth of 30 kHz, for example, you have to estimate the value of e_n between the graphs for 10 kHz and 100 kHz.

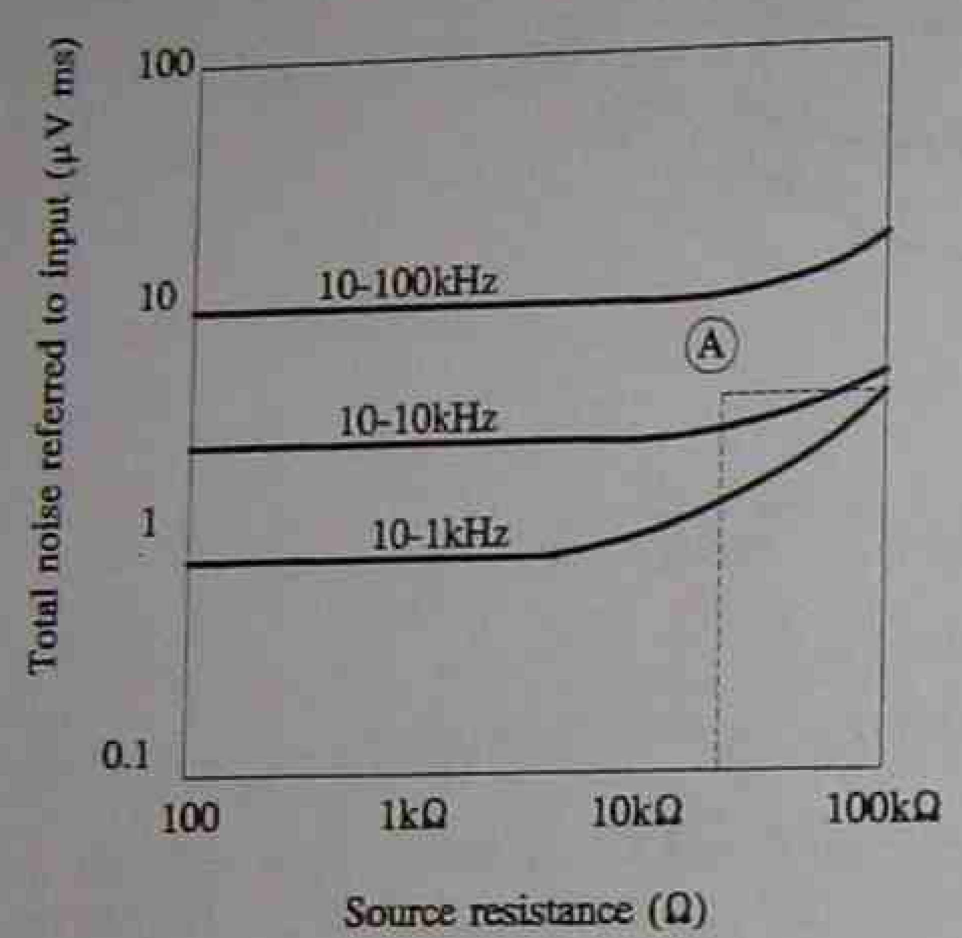
Example 8: Using the broadband noise graph

An amplifier using a 741 op amp has source noise resistance of 20 kΩ. It is bandlimited to 20 kHz using a RC filter. Use the broadband noise graph to obtain the total equivalent input noise voltage.

Solution

Effective BW = 20 kHz * π/2 = 31.4 kHz. Reading the graph for R_s = 20 kΩ and BW = 31.4 kHz, the answer is estimated to be at point A.

$\therefore e_n$ = 5 μV rms.



Calculation of e_n using equation 5

If the broadband noise graph is not available, we have to calculate e_n using equation 5. We have already seen how to calculate R_s, e_{op} and i_{op} if we know the BW and the noise V/√Hz and A/√Hz. The resistive noise e_R is calculated from the formula

$$e_R = \sqrt{4 * k\,T\,R_s * B} \qquad \text{Equation 9}$$

k = Boltzmann's constant = 1.38E-23 JK^{-1}
T = absolute temperature K = °C + 273
R_s = resistance
B = effective noise bandwidth

Now equation 5 can be used to calculate e_n.

Example 9 : Calculation of total equivalent input noise

An amplifier using type 741 op amp has source noise resistance of 20 kΩ. The effective circuit bandwidth is 31.4 kHz. From the data sheets, the 741 op amp has squared input noise voltage of 4E-16 V^2/Hz and squared input noise current of 3E-25 A^2/Hz. The temperature is 25°C.

Calculate the total equivalent input noise voltage (e_n).

Solution

R_s = 20 kΩ; T = 273+25 = 298°K; B = 31.4kHz

$\therefore e_R^2$ = 4 * 1.38E-23 * 298 * 20k * 31.4k
= 10.33E-12 V^2

e_{op}^2 = 4E-16 * 31.4k = 12.56E-12 V^2

$i_{op}^2 * R_s^2$ = 3E-25 * 31.4k * $(20k)^2$ = 3.77E-12 V^2

$$\therefore e_n = \sqrt{10.33E{-12} + 12.56E{-12} + 3.77E{-12}}\ V$$
= 5.2 μV rms.

Example 4 had the same data. The broadband noise graph gave nearly the same answer with a lot less work.

Noise gain and the output noise voltage

The output noise is calculated by the formula:

output noise = total equivalent input noise * noise gain

We have discussed the calculation of input noise e_n by two methods : using the broadband noise graph and using equation 5. The noise gain is just the *non-inverting* voltage gain of the circuit, whether the circuit is inverting, non-inverting or anything else. This is similar to offset voltage calculation.

Example 10: Output noise voltage calculation

The circuit of example 4 has an effective bandwidth of 100 kHz and uses a 741 op amp at 25°C.

(a) Use the broadband noise graph in Example 5 to calculate the total equivalent input noise voltage.

(b) Calculate the noise gain.

(c) Calculate the output noise voltage.

Solutions

(a) R_s = 6.4k (see example 4). Using the graph in example 5, for 100 kHz, we estimate e_n = 6.5μV rms.

(b) noise voltage gain = 1 + 100k/3.2k = 32.25

(c) output noise voltage = 6.5μV * 32.25 = 210 μV rms.

Signal to noise ratio (SNR): output SNR and input SNR

We see that, even with cheap general, purpose op amps, the output noise is quite small — some hundreds of microvolts. So why worry about it ?

In many circuits, especially those used in communications and measurements with transducers, the signal is also quite small. What really matters is how the signal voltage compares with the noise voltage. The simplest way to describe this is to use the signal to noise ratio (SNR) :

SNR = signal rms voltage / noise rms voltage   Equation 10

$(SNR)_{dB} = 20 \log (SNR_{ratio})$ in dB notation   Equation 11

Note that rms voltages have to be used. Noise voltage is already rms, so signal voltage also has to be made rms.

Equations 10 and 11 work in the normal way for output SNR. For input SNR, the noise should include *only the resistive noise,* not the total equivalent input noise e_n. This is because e_n includes e_{op} and i_{op} which are really added on after the input, so it is not correct to include them in the input SNR.

Input SNR = input signal rms / e_R Equation 12

The output SNR is always less than the input SNR, for two reasons : the former includes op amp noise and the latter does not; and the noise gain ≥ signal gain.

Example 11 : Input and output SNR calculations

Continuing examples 7 and 10, suppose the input signal is sine wave of 20 mV p-p and connected to v_2. (Ignore v_1)

(a) Calculate the output SNR

(b) Calculate the input SNR

Solutions

(a) output signal = -100k/4.7k * 20 mV p-p = 425.5 mV p-p
= 425.5 /(2√2) = 150 mV rms
output noise = 210 μV rms (from example 11)
∴ output SNR = 150 mV/210 μV = 716.4 = 57.1 dB

(b) $e_R = \sqrt{4*1.38E{-}23*298*6.4k\Omega*100kHz} = 3.24\mu V$

∴ Input SNR = (20/2√2mV)/3.24μV = 2182.4 = 66.8 dB

Measurement of noise

Since noise voltage is usually expressed as rms, we can measure noise directly with a true-RMS reading meter (such as an AC millivoltmeter) and use the reading directly.

If you use a DVM, you have to multiply the reading by 1.13. This is because the DVM is internally calibrated to read only sinewave rms voltages. If you doubt this, try measuring a square wave rms voltage using a DVM and an ACMVM. You will see different readings and the latter is the correct value.

Using a CRO, display the signal with a slow sweep (you can use AC or DC signal coupling), observe the p-p noise voltage and then estimate :

the noise rms voltage = (noise p-p voltage)/6

Note that it is incorrect to divide noise p-p voltage by 2√2. The factor 2√2 works only for sinewaves.

The conversion factors 1.13 and 6 mentioned above come from advanced mathematics beyond the scope of this module.

Methods to improve noise performance

To get acceptable SNR, we have to take steps to minimise noise, both external and internal.

To minimise *external noise,* good circuit construction in the form of shielding, good grounding, tight wiring and properly stabilized and filtered power supply is needed. Filtering to remove specific types of interference can also be useful.

A simple and powerful method to reduce external noise is to use a differential amplifier at the input stage, where the difference between the + and - input terminal voltages is amplified. Any externally picked up noise is likely to be present equally at both the inputs, and so will be subtracted and not appear at the output.

To minimise *internal noise,* the following are some helpful suggestions.

- Reduce bandwidth to minimum needed.
- Keep all resistor values as low as possible.
- Use low noise op amps.
- In a multistage amplifier, minimise the noise in the first stage. This is obviously because any noise generated in the first stage will be amplified and propagated in later stages.
- Keep the components cool by heat sinking or air flow.

Summary

- Noise in amplifiers can be due to external or internal sources. The internal noise sources are modelled by a lumped resistive voltage source and op amp voltage and current sources.
- Noise power is usually expressed by its density i.e. power per Hz. Noise voltage (or current) is expressed as $V/\sqrt{Hz}$ (or $A/\sqrt{Hz}$). In some data sheets, it is given as V^2/Hz (or A^2/Hz). You have to check the units carefully.
- Noise voltage is added using the rms addition formula.
- Noise voltage is approximately proportional to the square root of effective bandwidth. If filtering to limit noise is done, the effective bandwidth is obtained by multiplying the 3-dB bandwidth by a factor.
- The source noise resistance is the resistance between the input terminals of the op amp.
- The total equivalent input noise of the amp can be calculated by using the broadband noise graph in the data sheets, or by the rms noise addition formula.

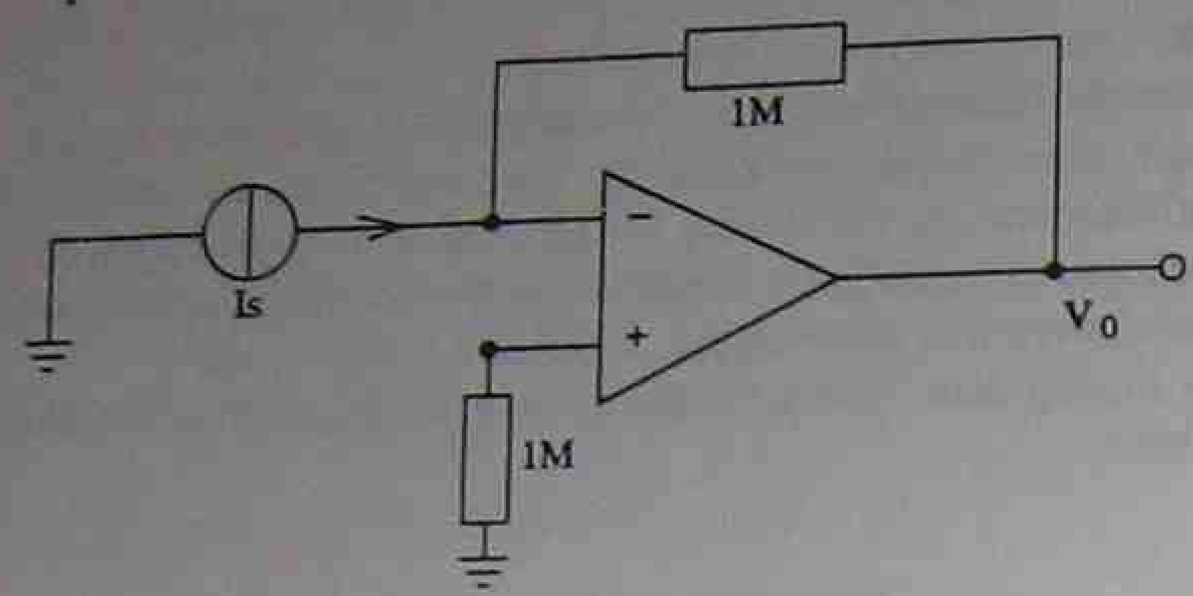
- The output noise is the total equivalent input noise multiplied by the non-inverting gain of the amplifier.
- The input SNR calculation uses only the resistive noise (and external noise) while the output SNR uses the amplifier's internal noise as well.
- Good circuit construction practice and careful selection of components can reduce noise.

Concluding remarks

We have built our discussion around intuitive concepts such as input resistance, bandwidth, noise voltage, current and power, voltage gain and SNR (even though the actual mathematical calculations are not always the same as with normal signals). In communications technology, some different terminology and specifications are often used such as noise temperature and noise figure, which are based on the fundamentals we have studied here. You may see them in a later module, but they will not be discussed here.

These questions will help you revise what you have learnt in Section 4.

1. The following circuit uses a 714 precision op amp, which has input noise voltage density = 11 nV/√Hz and input noise current density = 0.17 pA/√Hz. The circuit has a 3dB cutoff frequency of 30 kHz with a first order rolloff and the temperature is 27°C.



Determine:

(a) the source noise resistance.

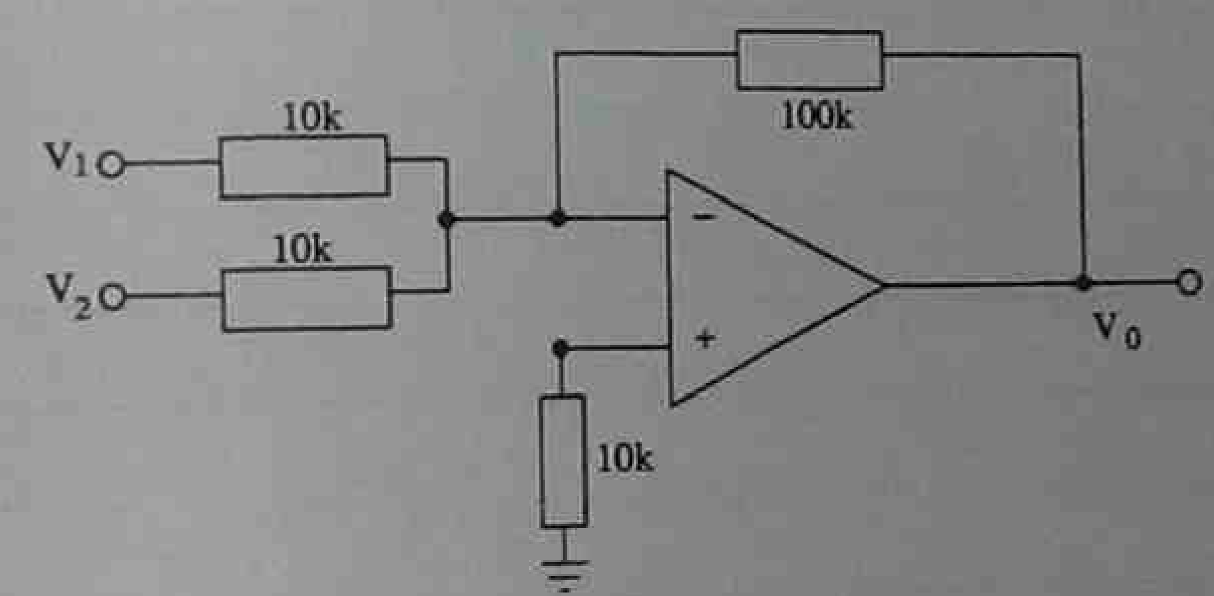
(b) the noise voltage gain.

(c) the total equivalent input noise.

(d) the output noise.

(e) the output SNR for an input current of 250 nA rms.

2. The following circuit uses a 741 op amp, which has input noise voltage density = 4E-16 V^2/Hz; input noise current density = 3E-25 A^2/Hz. The circuit has a 3dB cutoff frequency of 63.6 kHz with a first order rolloff and the temperature is 27°C.



(a) Determine:

(i) the source noise resistance.

(ii) the noise gain.

(iii) the total equivalent input noise.

(iv) the output noise.

(v) the output SNR for an input signal v_1 = 10 mV p-p and voltage v_2 = 0.

(vi) the input SNR.

(b) Check your answer for part (iii) using the broadband noise graph of the 741 op amp (Fig. 5).

3. State any two ways to reduce:

(i) external noise
(ii) internal noise in an amplifier.

Review questions

4. State how you can measure noise rms voltage using (i) an AC rms voltmeter; (ii) a Digital Multimeter; (iii) a CRO.

5. The noise at the output of an audio amplifier was 20 mV when the bandwidth was 20 kHz. Estimate the noise voltage if the bandwidth is changed to 30 kHz.

6. If two independent noise sources of -24 dBmV and -28 dBmV are added, what is the resulting noise voltage?

7. Describe the main characteristic of:
(i) white noise
(ii) pink noise.

Skill practice 5

Suggested duration

2 hours

Tasks

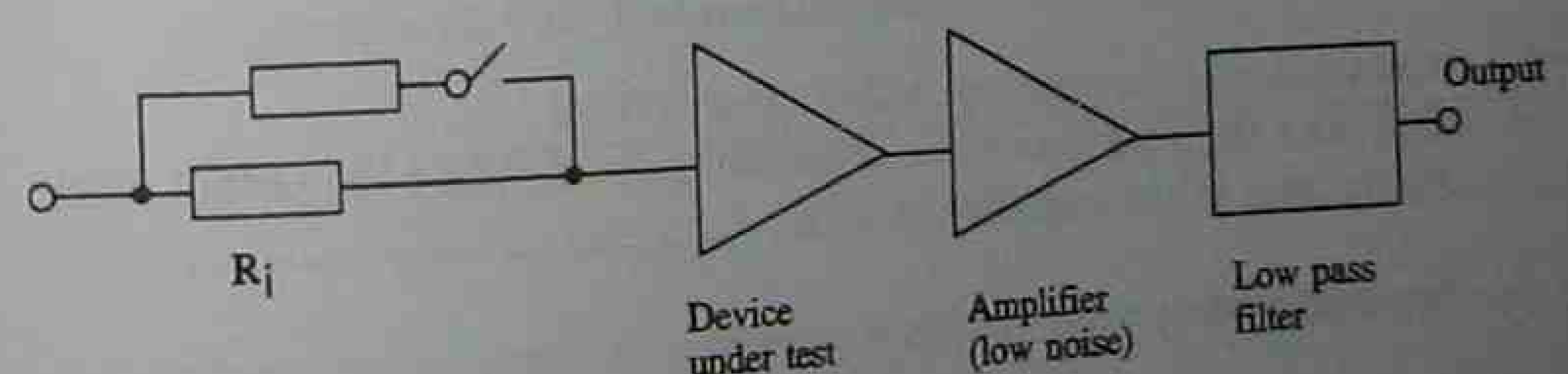
- To measure the input noise voltage of an op amp and observe its variation with source resistance and bandwidth
- To learn to measure noise rms voltage using a RMS voltmeter, a DVM and a CRO

Equipment

- 1 general purpose op amp (741 or 301 preferred).
- A high-gain, low-offset, low-noise amplifier (pre-wired in a shielded box - circuit diagram attached).
- DVM
- ACMVM
- CRO
- ±15V power supplies

Background Information

Since the equivalent input noise voltage of most op amps is in the order of microvolts, special precautions must be taken to measure such a small voltage. This experiment can be done only if a high-gain (A_v = 100,000), low noise, low offset (or AC-coupled) amplifier is available. The circuit must be shielded and only shielded cables must be used for external connections. The bandwidth of the circuit may be fixed or variable. The circuit diagram of a possible circuit is attached at the end. This circuit must be pre-wired in a shielded box. The voltage gain of the amplifier must be known. The block diagram of the setup is shown below.



- Overall gain of the amplifier = 100 dB
- The noise introduced by the low-noise amplifier is negligible.
- The cutoff frequency of the circuit can be switched between two values : 'high' and 'low'.
- The input resistance of the circuit can be switched between two values : 100Ω and 100kΩ.

Procedure

Step 1 *Setup*

- Power on the setup (without inserting your op amp) with ±15V supplies.
- Set the input resistance to 100Ω and cutoff frequency to 'low'.
- Observe the output on a CRO. You must see about 20 mV p-p of noise.
- Insert your op amp into the socket in the setup. (Switch off the power when inserting or removing op amps.)
- Connect a function generator to the input. Keep the signal voltage low because the circuit has high gain.

Step 2 *Measurement of the bandwidth of the circuit*

- Note the output signal voltage and measure the 3-dB cutoff frequency of the circuit using a CRO, ACMVM or DVM. (This is done by keeping the generator voltage constant and then increasing its frequency from a low value until the output signal voltage drops to 0.7 of, or 3 dB below, its low-frequency value. The frequency at which this happens is the 3-dB cutoff frequency.) For the suggested circuit, the cutoff frequency is about 2 kHz.
- Measured 3-dB Bandwidth = ...

Step 3 *Measurement of the output noise voltage using a RMS AC voltmeter*

- Disconnect the generator.
- Measure the output noise voltage using an ACMVM.
- Output noise voltage = ... rms

Step 4 *Calculation of the input equivalent noise voltage*

- Calculate the input equivalent noise voltage by dividing the output noise by the gain of the amplifier (= 10^5).
- Input equivalent noise voltage = ... rms

Step 5 *Noise measurement using a DVM*

- Connect a DVM to the amplifier output and measure the voltage.

  DVM reading = ...

- Multiply the reading by 1.13 to get the noise rms voltage.

  Noise rms voltage using a DVM = ...

Step 6 *Noise estimation using a CRO*

- Connect a CRO to the output of the amplifier.
- Select a slow sweep speed. You will see the noise voltage as a band with no internal details.
- Measure the noise p-p voltage.

  CRO p-p reading = ...

- Estimate the noise rms voltage by dividing the p-p voltage by 6.
- Estimated noise rms voltage using a CRO = ...

Step 7 *Observation of the effect of bandwidth and input resistance on output noise*

- Switch the input resistance to 'high'.
- Describe the output noise in terms of amplitude and frequency content.

  ____

  ____

  ____

  ____

- Switch the input resistance back to 'low' and switch the cutoff frequency to 'high'.
- Describe the output noise in terms of amplitude and frequency content compared with your first measurement.

  ____

  ____

  ____

  ____

Discussion questions

1. Take the result of measurement using the ACMVM as the correct one. Calculate the percentage error in measuring noise rms voltage using the DVM and CRO.

___

___

2. Calculate the effective noise bandwidth of the circuit (= 3dB cutoff frequency * $\pi/2$).

3. Use the broadband noise graph of your op amp to estimate the equivalent noise voltage for the chosen source resistance and effective bandwidth. Calculate the percentage error in your measurement.

___

___

4. Even though the measured output noise voltage includes the effects of source resistance noise and op amp noise current, we have ignored them in our calculations. Referring to equation 5 in the notes, explain why this is possible.

___

___

___

5. Suggest a method for measuring the op amp noise current. (Refer to equation 5 in the notes.)

___

___

___

___

Section 5: Frequency compensation

SUGGESTED DURATION	PREAMBLE
7 hrs 30 mins	To explain the need for, and select some of the common methods of, frequency compensation in amplifier circuits and calculate their effect on amplifier performance.

Objectives

At the end of this section you should be able to:

- ☐ sketch and give the reason for the general shape of the open loop gain and phase plots vs frequency of typical op amps
- ☐ given the open loop gain and phase response (Bode) plots of the op amp, determine if the negative feedback amplifier (inverting or non-inverting) is stable, and if so, its phase margin and closed loop bandwidth
- ☐ state/sketch how the phase margin changes with closed loop gain and the effect of phase margin on amplifier response for sine and square outputs
- ☐ state the need for frequency compensation. Compare the advantages and disadvantages of internal compensation and external compensation. State the main features of the large and small signal frequency response of internally compensated op amps
- ☐ state the three common methods of external compensation in amplifiers - single capacitor, two capacitor and feedforward. In each case calculate the required value of compensation capacitor
- ☐ for single capacitor compensation, estimate the gain-bandwidth product and slew rate as a function of the compensation capacitor
- ☐ state the general effects of under- or over-compensation on amplifier response
- ☐ compare the effects of the three compensation methods on gain-bandwidth product and slew rate
- ☐ for a given compensation, estimate the small-signal bandwidth as a function of the gain
- ☐ demonstrate the effect of different compensation methods and compensation capacitors on amplifier bandwidth and slew rate

- ☐ for a given amplifier, use square wave testing to determine the optimal value of compensation capacitor
- ☐ using square wave testing, measure the bandwidth and slew rate of the amplifier.

References

The following references deal with topics in this section.

1. Jacob (1993), pp 203-209
2. Rutkowski (1994), pp 96-97, 104-106
3. Gayakwad (1993), pp 210-225
4. Coughlin & Driscoll (1993), pp 266-269

Open loop frequency response of op amps

Until now we have not said much about the frequency response of op amps. We have assumed, for example, that if the op amp has an open loop voltage gain of 90 dB at DC, it also has 90 dB gain at 20 kHz or 20 MHz. In reality, the transistors in the op amp have some shunt capacitances. Though these capacitances may be small, they may appear in parallel with very large internal resistances and thus form low pass filters with low cutoff frequency. This means that the open loop gain of an op amp starts decreasing from a low frequency.

The low pass filters not only attenuate the output. They also introduce a phase shift. In an earlier circuit theory module, you have seen that the phase shift of an RC filter varies from 0° at DC to -90° at high frequencies. If there are 3 low pass filters in the various stages of an op amp, the output phase shift may vary from 0° to -270°.

Therefore, the *open loop* frequency response of a typical op amp might look as follows. Fig. 1 illustrates the gain plot (or magnitude plot) and the phase plot.

You are reminded again, we are talking about the open loop response, not the closed loop response with negative feedback.

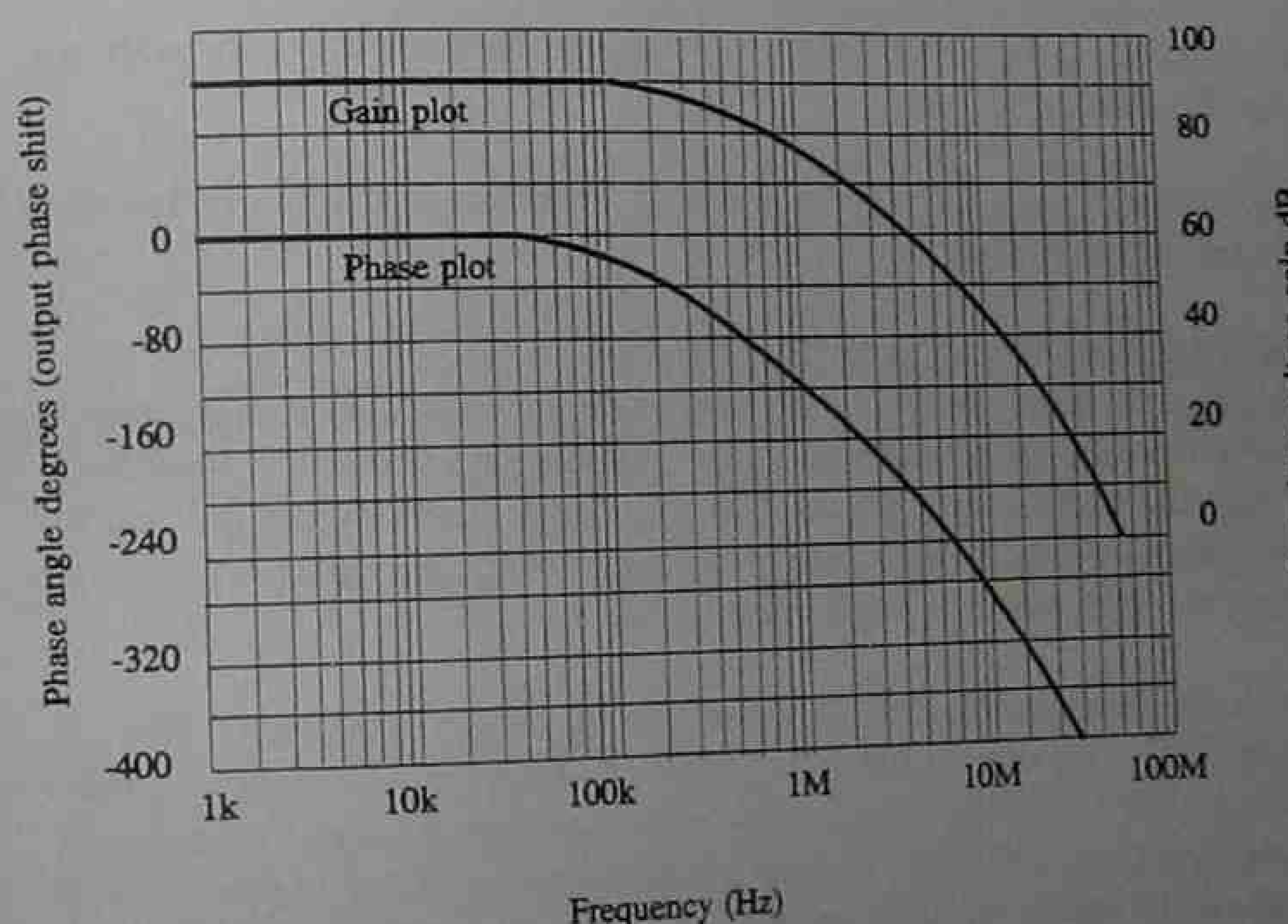


Fig. 1 Typical gain and phase response of an op amp

Stability of negative feedback circuits

We have seen in Section 1 that linear amplifier circuits do not use op amps in open loop, where the gain is very large and will clip the output. Negative feedback (NFB) is used to reduce the gain, by connecting a part of the output voltage back to the - input. Since the op amp actually amplifies ($V_+ - V_-$), the increased V_- reduces the effective input voltage to the op amp, reducing the output to usable levels.

The phase shift between the output and input in real op amps can change the picture. If the phase shift at some frequency is ≤ -180°, and the amount of feedback is large, then the -input can get a large inverted voltage from the output. Since this negative voltage is *subtracted* from the +input voltage, the overall effective input voltage is now bigger and the output gets bigger. This means that, even with no input signal, any tiny high frequency noise will appear at the output with large amplitude. The amplifier is said to be *unstable* in this situation.

Instability is unwelcome in amplifiers. We do not want to see a large, high frequency noise at the output mixed with our signal. However, instability is not always bad. Oscillators and clock circuits depend on the instability of NFB circuits.

Instability needs two conditions:

- the signal gain around the NFB loop ≥ 0 dB (which means that the open loop gain is large and/or the amount of feedback is large) ***and***

- the phase shift around the NFB loop ≤ -180°. These two conditions together are called the Barkhausen Criterion.

In this topic, we give a summary of the results as applicable to amplifiers and study methods to prevent instability.

Graphical determination of amplifier stability

You will learn the detailed theory on instability in another module on oscillators. To use the theory, we need the equations for the frequency response of the amplifier open loop gain and of the feedback circuit. With op amps, this information is not directly available — the data sheets only give a graph of the amplifier response. So in this section we study stability in graphical terms.

We also make another simplification — we study only resistive feedback. This means that we look only at amplifier circuits, and our results apply mainly to the basic circuits studied in Section 1, plus a few related circuits such as the summing amplifier and differential amplifier. If there are capacitors in series or parallel with R_F or R_1, our results cannot be used directly.

The graphical method for determining the stability of a NFB amplifier circuit is as follows.

- Calculate $A_{CL} = 1 + R_F/R_1$. This applies to both inverting and non-inverting amplifiers (as with offset voltage and noise calculations). As usual, R_F and R_1 are effective values after simplifying series and parallel connections.
  Change A_{CL} to dB : A_{CL} (dB) = 20 * log(A_{CL}).

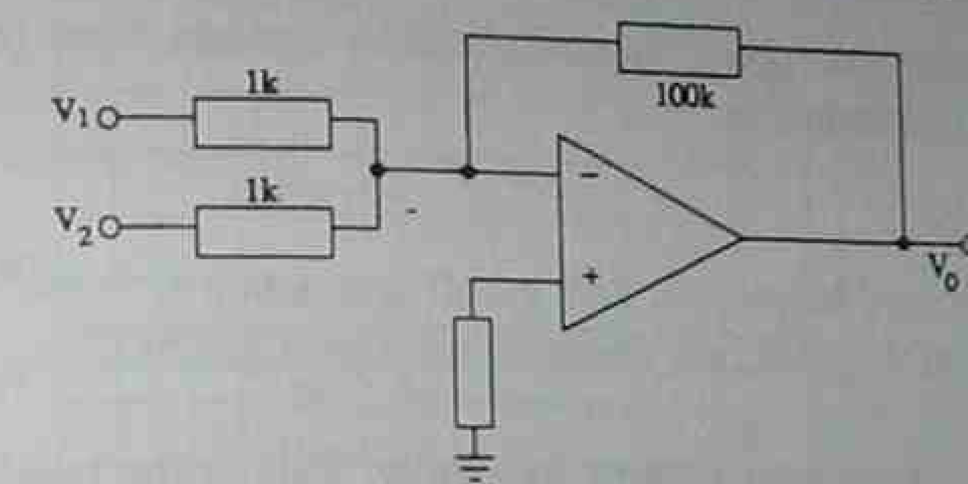
- On the open loop gain plot, draw a horizontal line at height A_{CL} (dB).
- Note the frequency at which the line cuts the open loop plot.
- Read the phase plot value at the relevant frequency.
- If the value is ≤ -180°, the amplifier is unstable. If it is >-180°, the amplifier is stable.

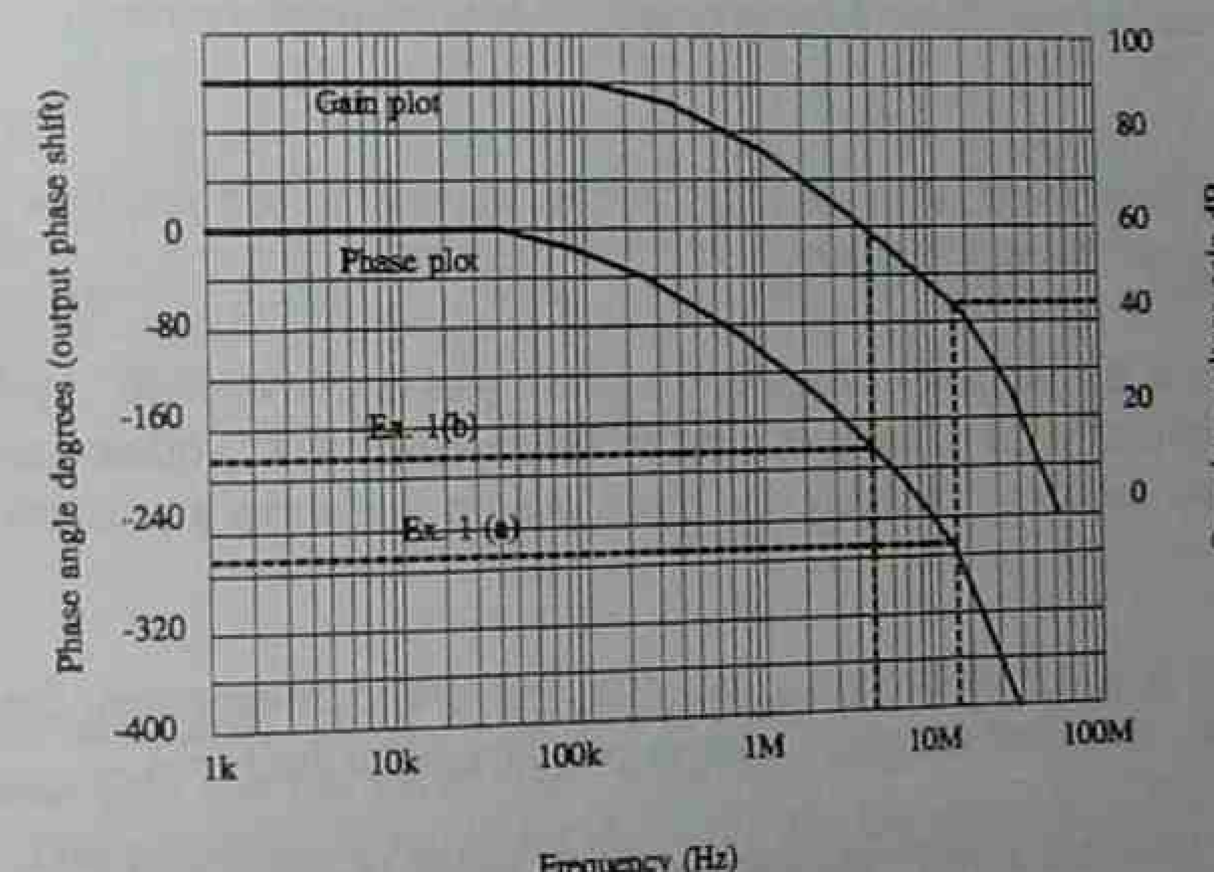
Example 1 : Stability condition

The op amp in the amplifier circuit shown below has the gain and phase plots shown on the subsequent graph.

(a) Is the amplifier stable ?

(b) What is the minimum A_{CL} needed to make the amplifier stable ?





Solutions

(a) R_1 = 1k ‖ 1k = 500Ω
∴ A_{CL} = 1 + 100k/500 = 200 = 46 dB
The horizontal line at 46 dB cuts the gain plot at 12 MHz.
At 12 Mhz, the phase plot value is -270°.
∴ The amplifier is unstable.

(b) Working backwards from the phase plot,
∠A_{OL} = -180° at 5 MHz.
At 5 MHz, the gain plot value is 60 dB.
∴ minimum A_{CL} for stability = 60 dB = 1000.

Sometimes the phase plot is not given in data sheets. There is a rule of thumb for determining stability using only the gain plot. The method follows.

- Calculate $A_{CL} = 1 + R_F/R_1$. This applies to both inverting and non-inverting amplifiers (as with offset voltage and noise calculations). As usual, R_F and R_1 are effective values after simplifying series and parallel connections. Change A_{CL} to dB : A_{CL} (dB) = 20 * log(A_{CL}).
- On the open loop gain plot, draw a horizontal line at height A_{CL} (dB).
- Note the frequency at which the line in the second cuts the open loop plot.
- Draw the slope of the gain plot at the frequency found in the third step.
- If the slope is ≤ 20 dB/dec, the NFB amplifier is sure to be stable. If the slope is between 20 dB/dec and 40 dB/dec, it may be (but not always) unstable. If the slope ≥ 40 dB/dec, the amplifier will be unstable.

This rule is easy to apply if the gain plot is approximated by straight lines (called Bode Plot). Such plots are quite common.

Closed loop bandwidth

If an amplifier is stable, its closed loop upper 3dB cutoff frequency $f_c \approx$ the frequency where the horizontal A_{CL} line cuts the open loop gain graph.

If A_{CL} gets larger, it cuts the gain plot at a lower frequency. This is the basis for the rule that the product of the closed loop gain and the bandwidth is constant.

Phase margin

Even if an amplifier is stable, we need a measure of stability to see just how stable it is. Phase margin provides this measure.

The phase margin is the number of degrees the phase plot is above -180° at the closed loop f_c. Obviously the phase plot must be above -180° at f_c, otherwise the amplifier would be unstable. Therefore, the phase margin is always a positive number.

We have seen that f_c gets smaller as A_{CL} increases. For lower frequencies, the phase plot value is larger, therefore *the phase margin gets larger for larger A_{CL}.*

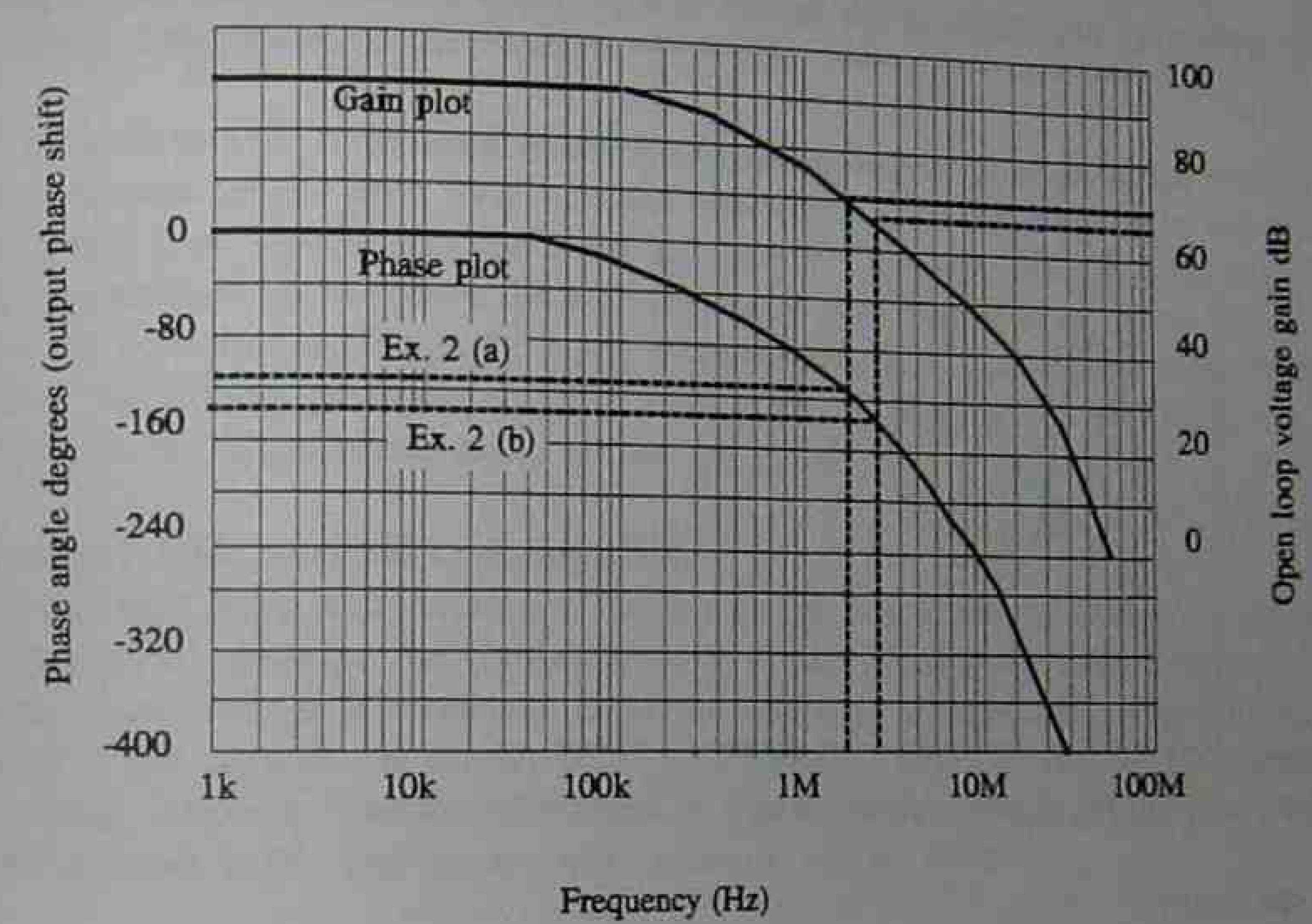
It is important to note that a NFB amplifier gets less stable if the closed loop gain is decreased, or if the open loop gain is increased. Since the smallest A_{CL} possible is 1, the amplifier has the greatest chance to be unstable as a voltage follower. If a NFB amplifier is stable as a voltage follower, it will be stable for any other closed loop gain.

Example 2 : Closed loop bandwidth and phase margin

The gain and phase plots of an amplifier are given below.

(a) estimate the closed loop bandwidth and the phase margin for a closed loop gain of 70 dB.

(b) What is the closed loop gain which gives a phase margin of 45° ?



Solutions

(a) The horizontal line at 70 dB cuts the gain plot at 2 MHz. At 2 MHz, the phase plot has the value -118° which is > -180°.
∴ The amplifier is stable for A_{CL} = 70 dB; the closed loop BW = 2 MHz; and the phase margin = -118 + 180 = 62°.

(b) Working backwards, Phase margin = 45° ==> phase plot value = 45° - 180° = -135°. This occurs at 3 MHz. At 3 MHz, the gain plot has the value 67 dB.
∴ A_{CL} = 67 dB = 2240 for a phase margin of 45°.

Effect of phase margin on closed loop response

The sketches below show the frequency response (Figure 2) and the square wave response (Figure 3) of the closed loop amplifier for different phase margins.

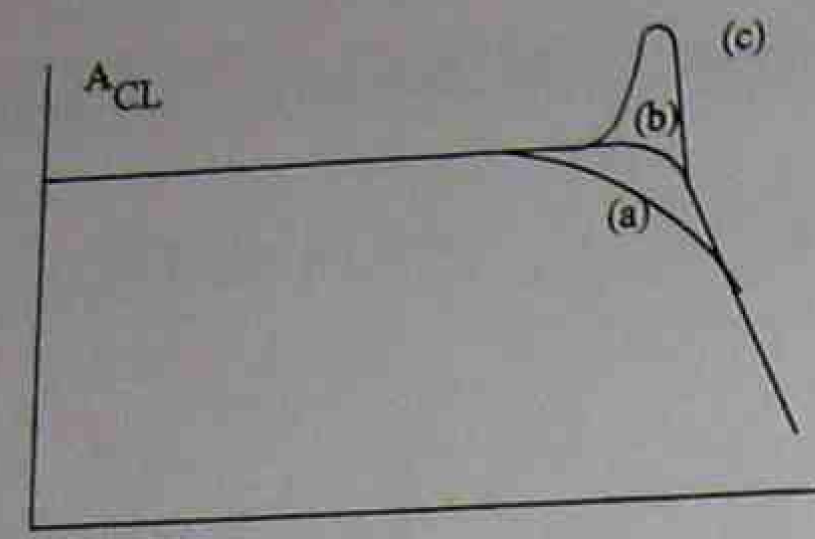


Fig. 2 Effect of phase margin on frequency response

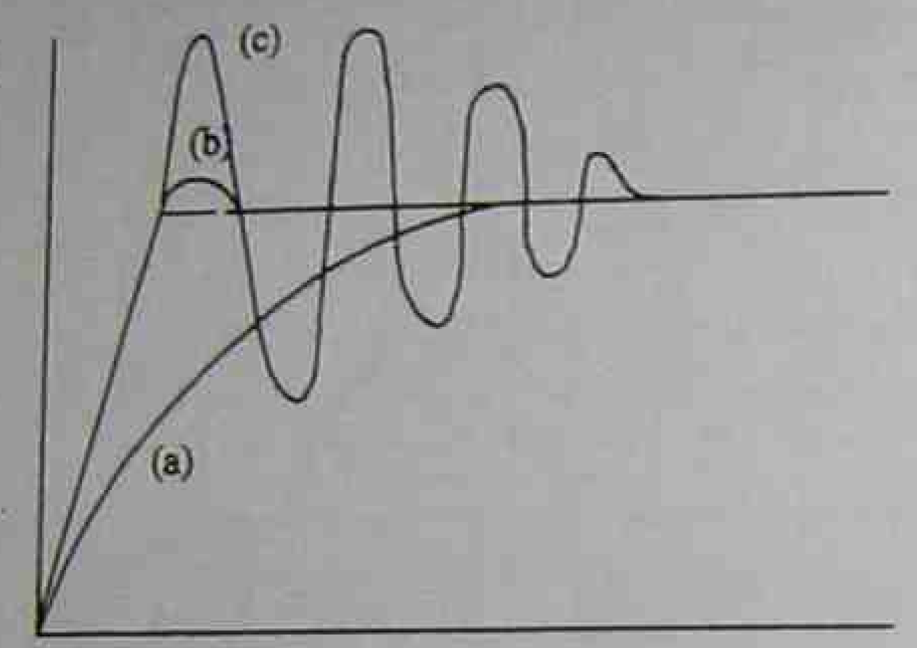


Fig. 3 Effect of phase margin on square wave response

In both the figures:

- curve (a) represents phase margin too high (close to 90°). In this case, the frequency response droops and the square wave response is slow (like an RC circuit).
- curve (b) represents the correct phase margin (close to 45°). In this case, the frequency response is nearly flat and the square wave response rises fast and settles down quickly (like a car with good suspension).
- curve (c) represents phase margin too low (close to 0°). In this case, the frequency response shows gain peaking near the cutoff frequency (like a resonant circuit). The square wave response shows 'ringing', i.e. many overshoots and undershoots before settling down.

The concept of phase margin is closely related to the concept of damping in second order circuits. In fact:

- phase margin of 45° <==> critical damping
- phase margin > 45° <==> overdamped
- phase margin < 45° <==> underdamped

Frequency compensation

In example 1, we have seen that the NFB op amp amplifier is stable only for high closed loop gain. You may ask how we can make a stable voltage follower with a 741 op amp. After all, a high A_{CL} is not always what we want since it can cause clipping of the output.

In Figures 2 and 3, we have seen that the phase margin affects the closed loop frequency response and square wave response. We may not get the desirable phase margin for our chosen A_{CL}. Usually the problem is that the phase margin is too little (or no margin at all i.e. instability).

Frequency compensation is a technique to change the phase margin by adding some external components to the op amp.

If we look closely at the section on phase margins, we see that the phase margin depends also on the open loop gain and phase plots of the op amp. If we can raise the phase plot, we can increase the phase margin. This is possible, but not easy. The other possibility is to reduce the high frequency open loop gain of the op amp, provided we do not change the phase plot too drastically. This can be done fairly easily, by putting a suitable low pass filter in the op amp open loop. This is the approach to compensation taken in most op amps.

Internal and external compensation

All op amps need compensation to work with reasonable values of A_{CL}. In internally compensated op amps, the compensation (low pass filtering the open loop gain) is done by an internally connected capacitor, called the compensating capacitor C_c. The value of the capacitor is usually set to give a phase margin of 45° when connected as a voltage follower. You cannot change C_c. For A_{CL} above about 10, the phase margin will be close to 90° and the amplifier response will be slow.

The benefit of internal compensation is that it is guaranteed to give a stable amplifier and reduces the component count. The disadvantages are that the speed as well as the slew rate are reduced, and that the user has no control over the frequency response.

Op amps are often used in open loop as comparators. In those applications, really no C_c is required, since there is no NFB. The fixed C_c in internally compensated op amps is a disadvantage in such cases, because the slew rate is unnecessarily reduced.

Many popular op amp types, such as the 741, LF351 and 714 are internally compensated. The 741 uses a C_c of 30 pF.

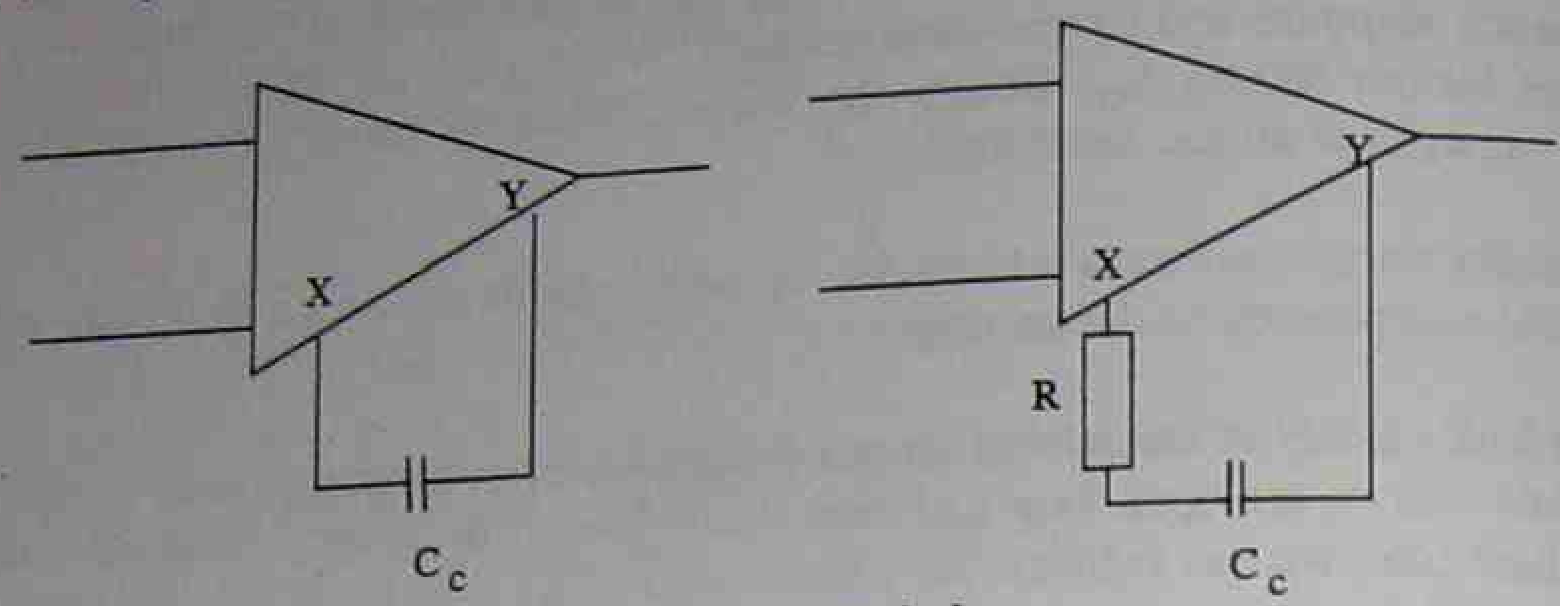
With externally compensated op amps, you have to connect your own filtering components, following the suggestions in the data sheets. For this purpose, the op amp provides compensation terminals in the package.

LM301 and NE5534 are examples of external compensation. LM301 is identical to the 741 except that the internal C_c is absent.

Common frequency compensation methods

Since the open loop response of the op amp can be filtered in many different ways, correspondingly there are several compensation methods. We study three methods commonly used.

Single capacitor compensation
The general schematic takes one of the two forms given below. The op amp's data sheets will say which form is recommended.



X , Y: compensation terminals

Fig. 4 *Fig. 5*
Single pole compensation configurations

Single capacitor compensation low-pass filters the open loop response by a RC filter. The effect is shown in Figures 8 and 9 on page 119.

For higher A_{CL}, we have seen that the phase margin is higher. The effect of low-pass filtering is to reduce the bandwidth,
and increase the phase margin. Therefore, to maintain a constant phase margin with larger A_{CL}, we have to filter less using a smaller C_c.

Required $C_c \propto 1/A_{CL}$ Equation 1

Note that we can use a value of C_c greater than or equal to the value given by equation 1. If we use a larger value, we get overcompensation, with a reduction in bandwidth and slew rate. This is usually what happens in internally compensated op amps.

The *'gain-bandwidth product'* (GBWP) of an amplifier is the product of the closed loop gain and closed loop bandwidth. For a closed loop gain of 1, the closed loop BW is clearly equal to the GBWP. Therefore, the GBWP is also called the *'Unity Gain Bandwidth'*.

From the previous discussion, we see that :
GBWP (nearly) $\propto$ 1/(actual C_c). Equation 2

We have learnt that slew rate is caused by the time taken to charge internal capacitances in the op amp. An extra compensation capacitor increases the charging time, so we can estimate that :

slew rate (nearly) $\propto$ 1/(actual C_c)Equation 3

The formulae in equations 2 and 3 are not exact, because of the effect of other stray capacitances in the circuit.

Example 3 : Calculation of compensation capacitor and its effect

The LM301 op amp requires a 30 pF compensation capacitor for use as voltage follower. Its slew rate is 0.5V/µs and gain bandwidth product is 1 MHz. This op amp is used to make a NFB amplifier with $A_{CL} = 10$.

(a) What is the optimal value of C_c ?

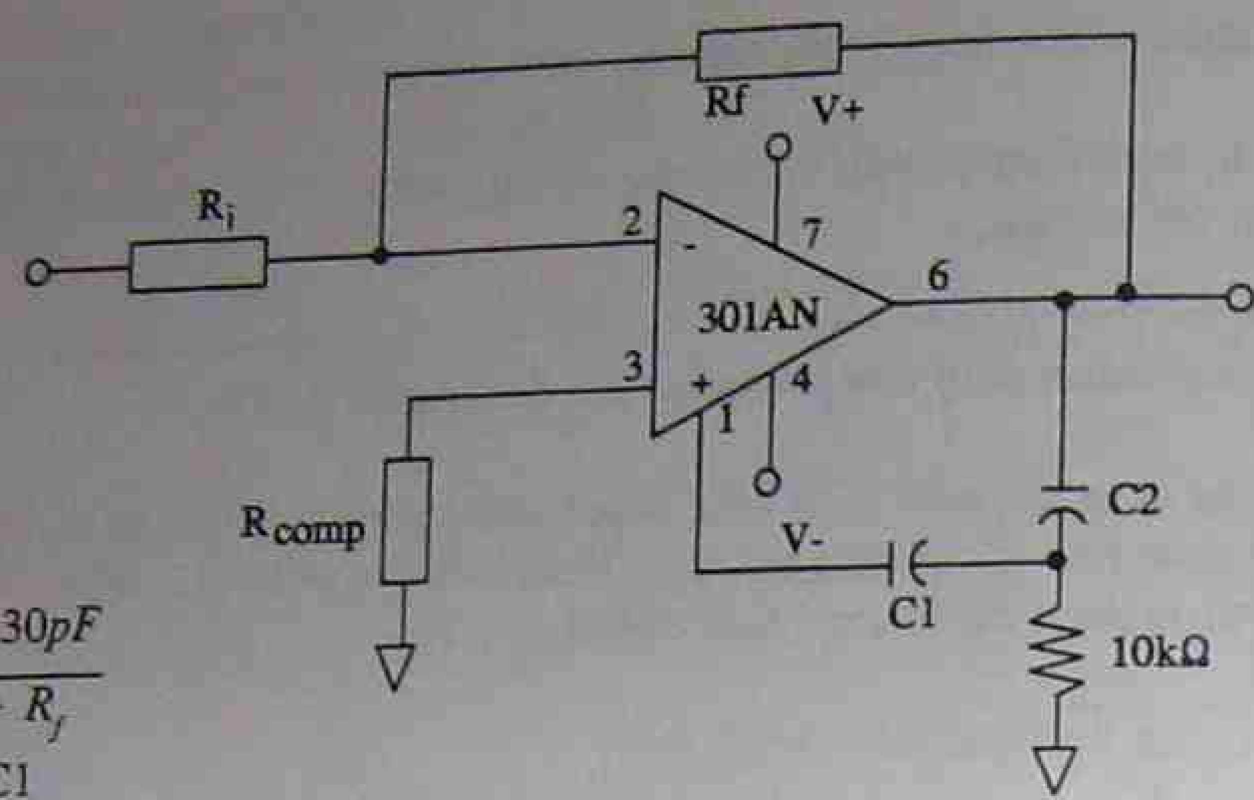
(b) If C_c of 10pF is actually used, what is the bandwidth of the circuit and slew rate ?

Solutions

(a) For $A_{CL} = 1$, $C_c = 30$ pF
$\therefore$ For $A_{CL} = 10$, $C_c \geq 30$ pF * 1/10 = 3 pF
The optimal value of $C_c = 3$ pF.

(b) With actual C_c of 10pF,
GBWP = 1MHz * 30pF/10pF = 3 MHz
$\therefore$ BW = 3MHz/10 = 300 kHz
Slew rate = 0.5V/µs * 30pF/10pF = 1.5V/µs

Two-capacitor compensation



$$C1 \geq \frac{R_i \times 30pF}{R_i + R_f}$$

C2 = 10 x C1

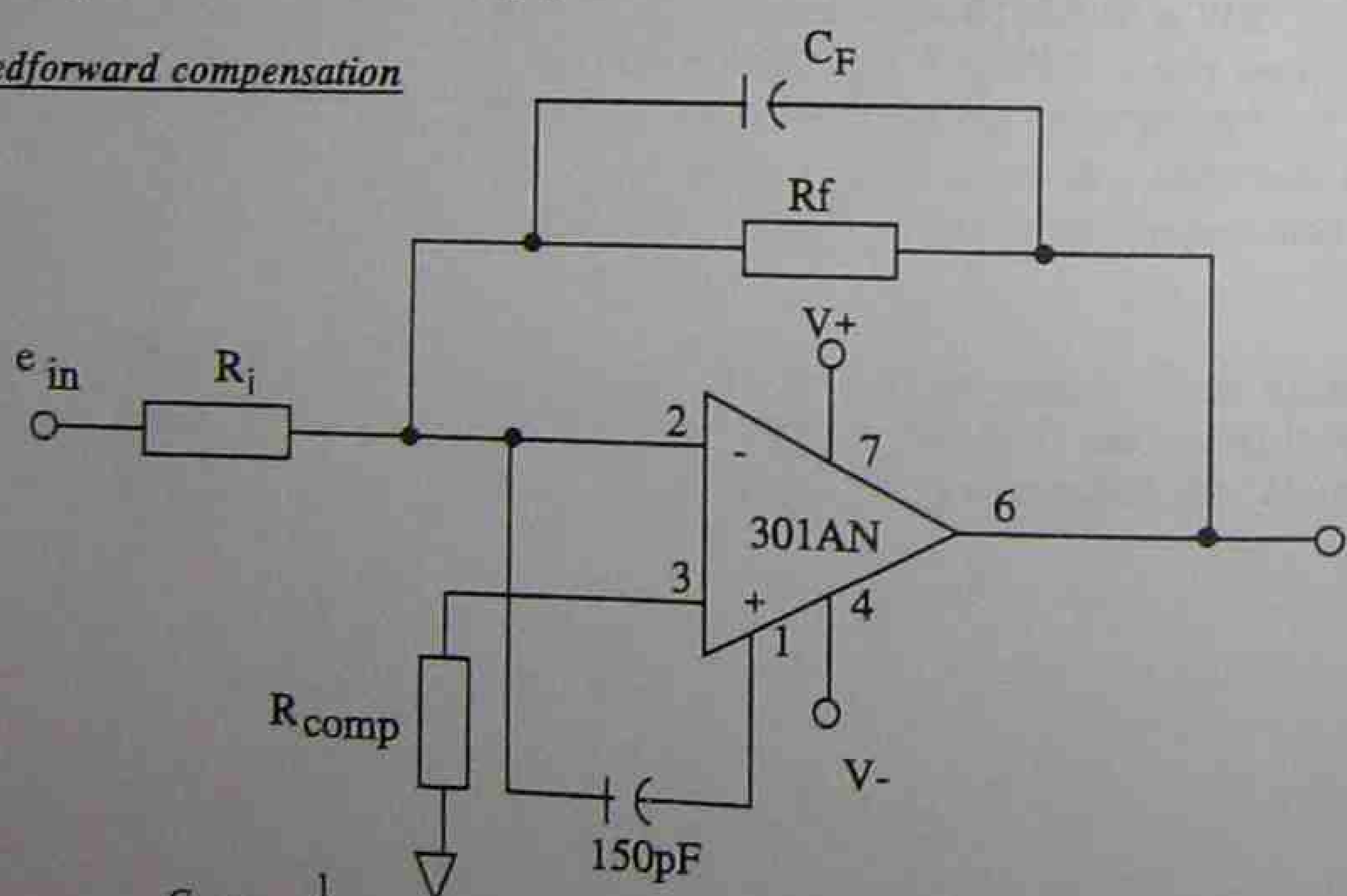
Fig.6 Two-pole compensation.

The reduction in high frequency gain, which is needed for stability, is achieved with a second order filter, which has a sharper rolloff. Therefore, smaller filtering capacitors can be used to get the same reduction in gain, improving the slew rate.

A two-capacitor filter gives a higher slew rate (and full power bandwidth) for a given gain-bandwidth product, compared to a single capacitor.

As with single capacitor compensation, the required (optimal) capacitor values are inversely related to the closed loop gain.

Feedforward compensation



$$C_F = \frac{1}{2\pi f_H R_F}$$

Fig. 7 Feedforward compensation

In many op amps, the signal (usually the inverting one) has to pass through some slow biasing transistors, which introduce a large negative phase shift, and so reduce stability. In feedforward compensation, the phase shifting transistors are bypassed by a capacitor. This means the NFB signal does not have to go through these transistors, and is not phase-shifted as much.

The high frequency gain is still large, which also causes stability problems. To reduce the gain, R_F is shunted by a suitable small capacitor C_F. The value of C_F must be at least 3 pF, and can be larger if a reduction in bandwidth can be tolerated.

The use of C_F works best only for inverting amplifiers, so feedforward compensation is used only in inverting amps.

Feedforward compensation gives extremely high slew rate and GBWP - typically about 10 times better than can be obtained by two-capacitor compensation.

The following figures 8 and 9 compare the open loop frequency response and full power bandwidths for different compensation methods.

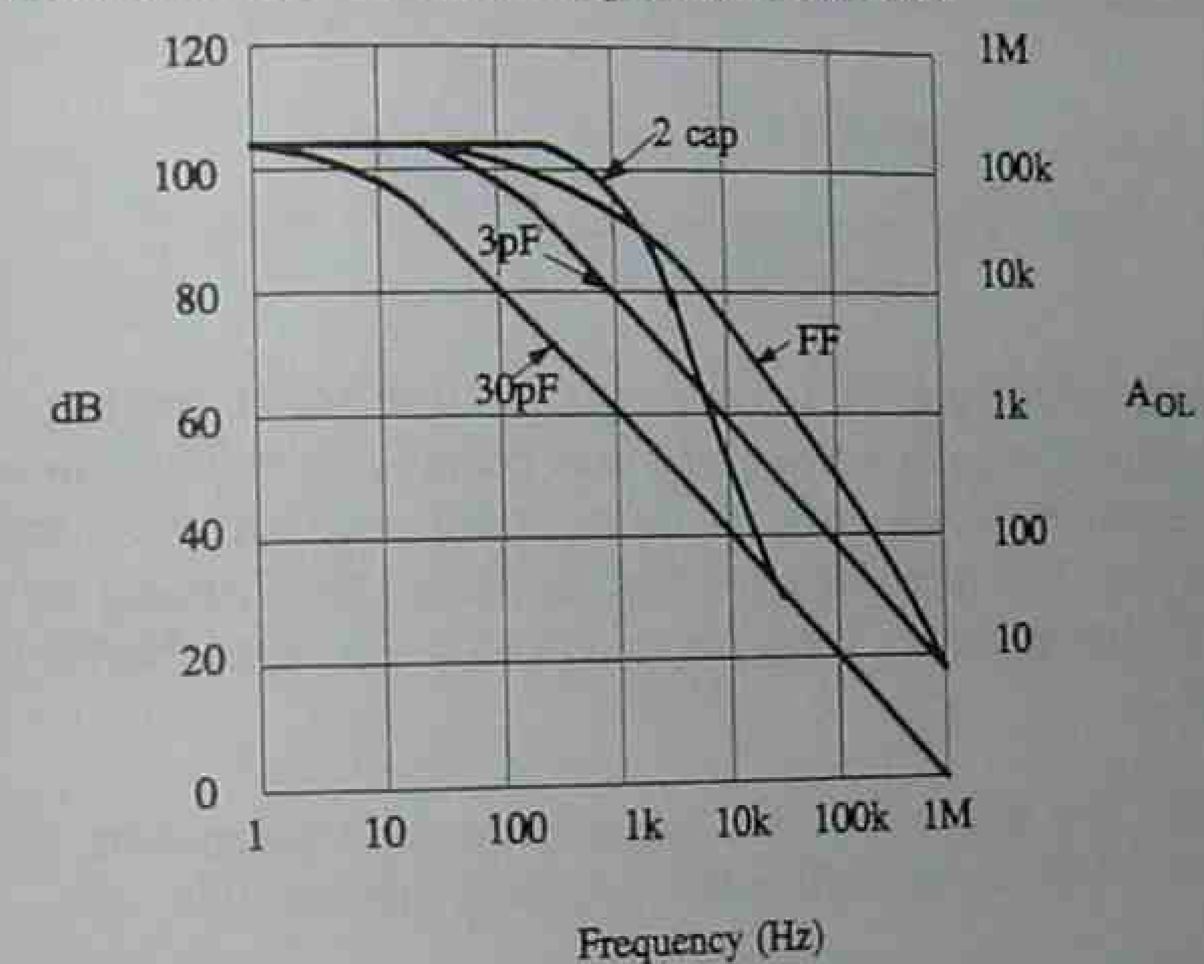


Fig. 8 Effect of external compensation on gain bandwidth.

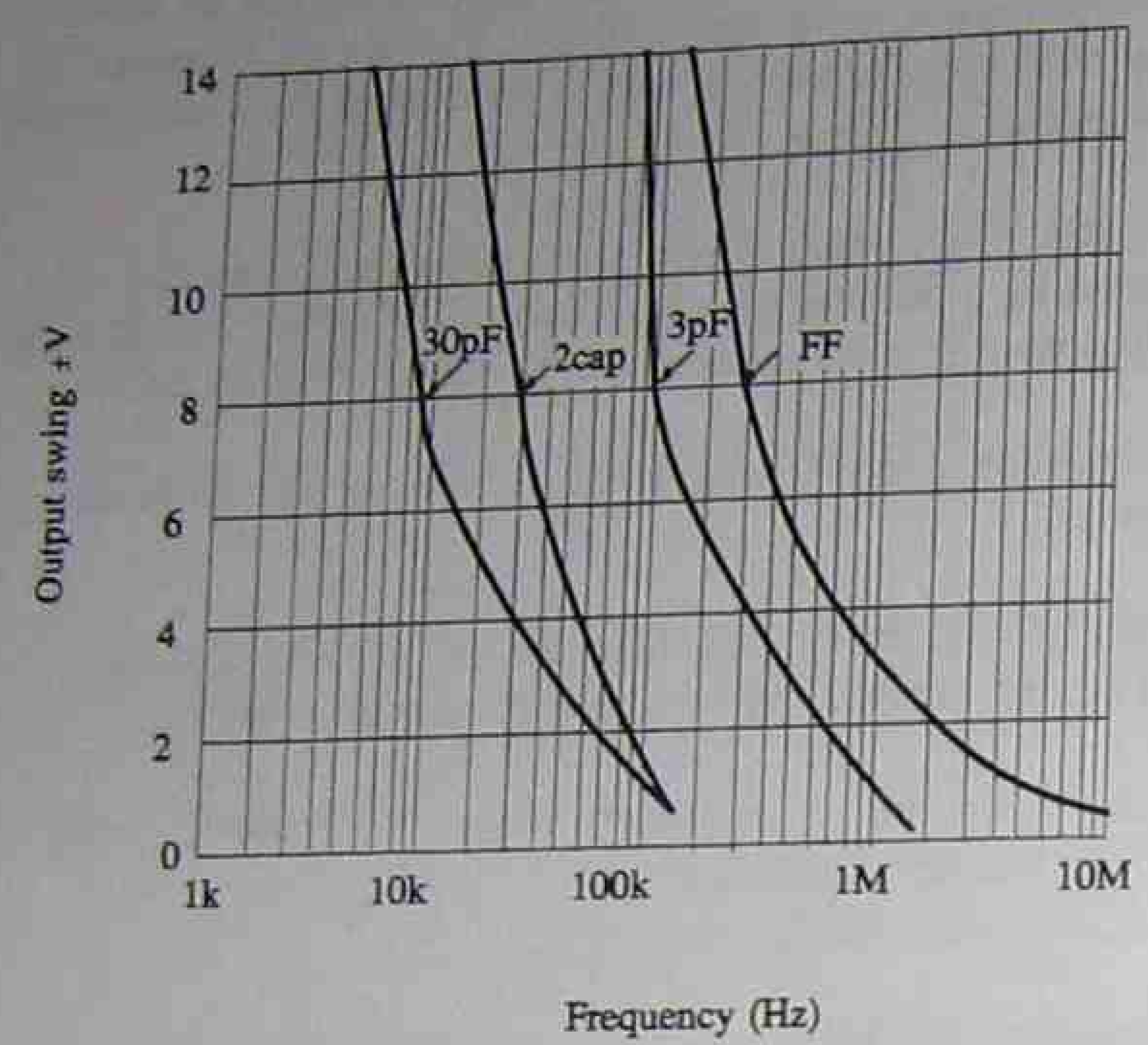


Fig. 9 Effect of external compensation on full power bandwidth

Measurement of bandwidth and slew rate using source wave testing

The aim of compensation is to provide a proper phase margin to the NFB circuit, and that with a proper margin, the square wave output has a fast rise time and a small overshoot without ringing. This means that the best practical way of setting up compensation is to observe the square wave output and adjust the capacitor(s) until the output looks good.

The square wave output also allows us to calculate the two main parameters limiting the frequency response. These are the 3dB cutoff frequency and the slew rate.

The 3dB f_c can be measured only with *small* output signals. 'Small' means not slew rate limited.

The 3dB $f_c = 0.35/t_r$ where t_r is the 10%-90% rise time of the output signal, as shown below in Figure 10.

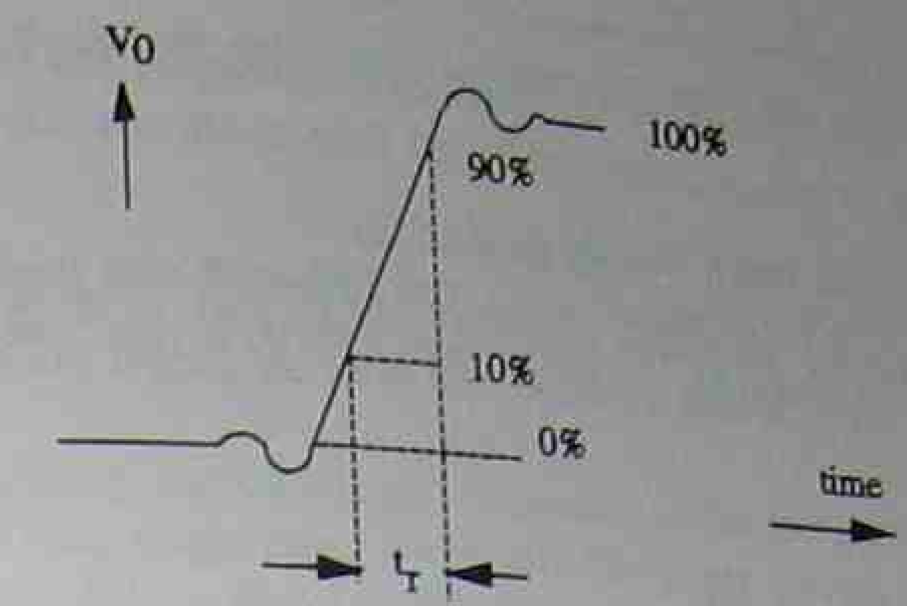


Fig. 10 Bandwidth measurement using square wave testing

There is no need to change the frequency of the square wave, or even know it. The 10% and 90% points are taken so as to avoid any glitches or ringing at the corners of the square wave.

Slew rate is measured with large amplitude signals and is calculated as rise in voltage/run in time (as discussed in Section 3).

The output signal is considered 'large' if the rise time due to slew rate (which increases with output voltage) is larger than the rise time due to bandwidth (which is independent of the output voltage).

Example 4 : Bandwidth and slew rate calculations using the results of square wave testing

A non-inverting amplifier has a voltage gain of 20.
When the input is a 10mV p-p square wave, the 10%-90% rise time was 3.5 µs.
When the input was increased to 1V, the output 10%-90% rise time increased to 12.8 µs.
Calculate :

(a) the small signal bandwidth.

(b) the slew rate.

(c) the gain bandwidth product of the amplifier.

(d) the p-p square wave input voltage when the 10%-90% rise time due to slew rate is equal to the 10%-90% rise time due to the bandwidth limitation.

(e) the new small signal rise time if the gain is adjusted to 5.5 .

Solutions

Since the rise time for 1V input has increased from that for 10mV input, we expect that the second output is limited by the slew rate and the first by the bandwidth. (The rise time due to bandwidth is independent of voltage.)

(a) BW = 0.35/3.5µs = 100 kHz

(b) Output voltage = 10 * 1V p-p = 10V p-p
10%-90% of output voltage = (90%-10%) * 10V =
8V = rise in voltage
rise time = 12.8 µs
∴ slew rate = 8V/12.8µs = 0.625 V/µs

(c) GBWP = 10 * 100 kHz = 1 MHz

(d) 10%-90% rise time due to BW = 3.5µs
Suppose v_i is the input voltage p-p,
then output voltage = $10v_i$ p-p
10%-90% of v_o = 80% * $10v_i$ = $8v_i$
∴ rise time due to S.R. = $8v_i$/0.625V/µs = $12.8v_i$µs
For the two rise times to be equal,
$12.8v_i = 3.5$ ∴ v_i = 273 mV p-p
(Input voltages below 273 mV p-p are small signal, and above 273mV p-p are large signal.)

(e) For A_v = 5.5, BW = 1 MHz/5.5 = 182 kHz
∴ rise time = 0.35/182kHz = 1.9 µs

∴ rise time = 0.35/182kHz = 1.9 µs

Summary

- The magnitude of the frequency response of an op amp in open loop decreases with frequency, and the phase shift gets more negative, due to stray capacitances in the device.

- If the signal phase is ≤ -180° at the frequency where the open loop and closed loop magnitudes are equal, the amplifier becomes unstable (i.e. oscillates).

- For a stable amplifier, the 3dB cutoff frequency is the frequency where the open loop and closed loop magnitudes are equal. The phase margin is the number of degrees the phase shift is above -180° at the cutoff frequency. For higher closed loop gain, the cutoff frequency decreases and phase margin increases.

- If the phase margin is about 45°, the closed loop frequency response is nearly flat and the square wave response shows fast rise time, no ringing and little overshoot. If the phase margin is well above 45° (≈90°), the bandwidth decreases and the square wave rise time increases. If the phase margin is well below 45° (≈0°), the frequency response shows gain peaking and the square wave response has ringing.

- Frequency compensation is used to stabilize an NFB amplifier or to adjust its phase margin. The idea is to reduce the high frequency gain, negative phase shift, or both, of the op amp.

- Common methods of compensation are single capacitor, two capacitor or feedforward.

- The optimal value of the compensation capacitor is inversely related to the closed loop gain.

- The gain bandwidth product and slew rate are approximately inversely related to the actual value of the compensation capacitor.

- Internally compensated amplifiers use a fixed single compensation capacitor. This is convenient, but since the circuit is usually overcompensated, the GBWP and slew rate are poor.

- Compared to single capacitor compensation, two-capacitor compensation gives a higher slew rate for a given GBWP. Feedforward gives very high GBWP and slew rate and is usually used only with inverting configurations.

- Square wave testing is convenient to set up optimal compensation, and to measure the 3dB cutoff frequency (with small signals) and slew rate (with large signals).

Review questions

These questions will help you revise what you have learnt in Section 5.

1. The open loop gain and phase plots of an amplifier are given in the following graph.

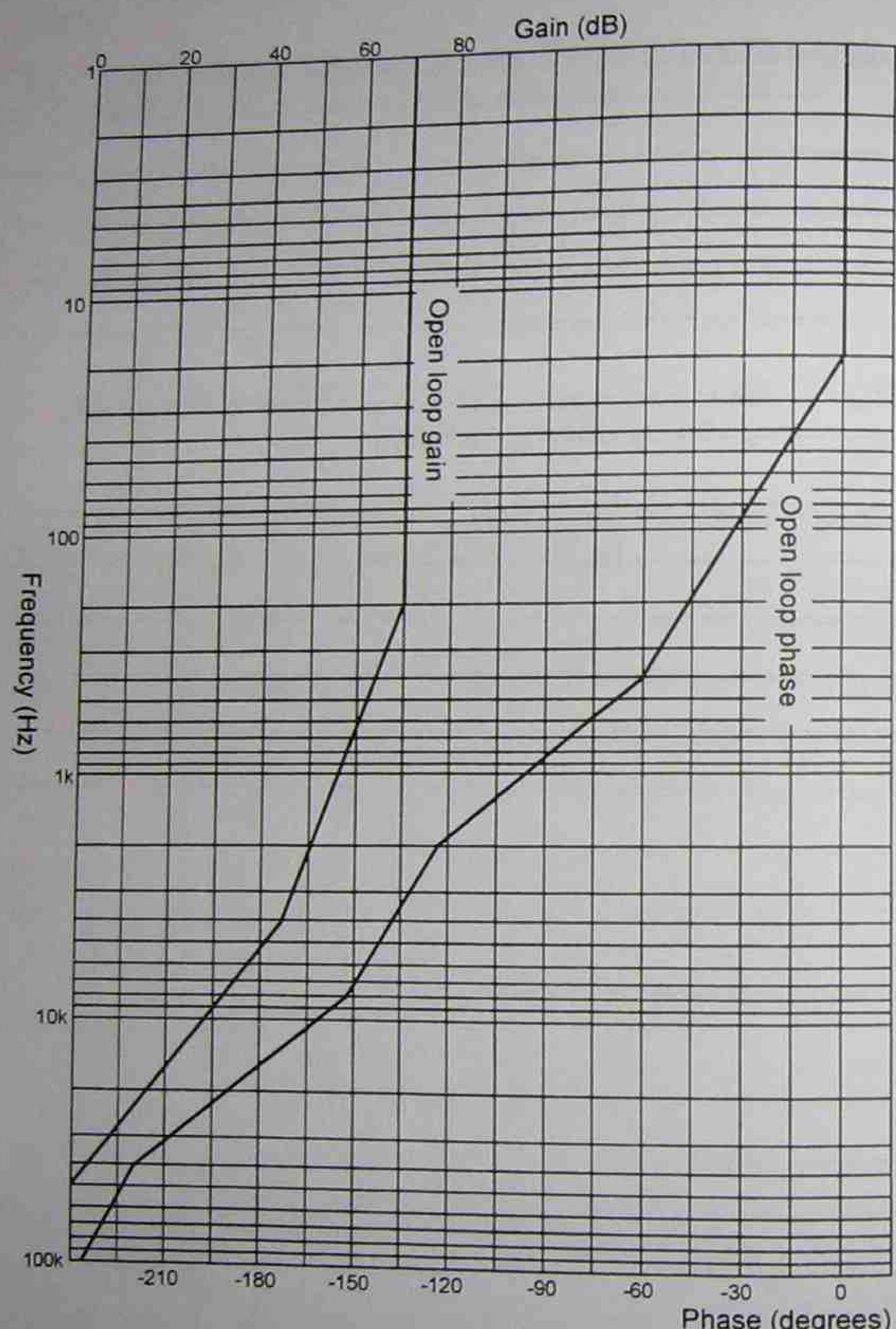
   (a) State whether the amplifier with NFB is stable for a closed loop gain of 0 dB. Outline the reasons for your conclusion.

   (b) State whether the amplifier with NFB is stable for a closed loop gain of 30 dB. Outline the reasons for your conclusion.

   (c) What is the phase margin if the amplifier is connected for a closed loop gain of 50 dB ?

   (d) If a phase margin of 45° is needed, what is the corresponding closed loop gain?

   (e) What is the minimum closed loop gain for stable NFB operation (regardless of good phase margin)?



2. (a) What is the effect on the step response of a NFB amplifier if the phase margin is
   (i) very small?
   (ii) too large?

____

____

   (b) What is the effect on the frequency response of a NFB amplifier if the phase margin is
   (i) very small?
   (ii) too large?

____

____

____

3. (a) What are the advantages of using internally compensated op amps?

____

____

____

   (b) What are the advantages of using externally compensated op amps?

____

____

____

____

4. For a given compensation, if the closed loop gain is increased, what is the effect on:
(i) closed loop bandwidth?
(ii) phase margin?

5. (a) For what closed loop gain do internally compensated op amps give best performance?

(b) Why is this particular value of A_{CL} chosen?

6. (a) For single capacitor compensation, if C_c is increased and the gain is unchanged, how will the closed loop bandwidth, gain bandwidth product and slew rate change (increase, decrease or unchanged)?

(b) For single capacitor compensation, if C_c is unchanged but the gain is increased, how will the bandwidth, gain bandwidth product and slew rate change (increase, decrease or unchanged)?

(c) What is the advantage of two capacitor compensation over single capacitor compensation?

(d) What is the operating principle of feedforward compensation?

(e) What is the general effect of feedforward compensation on gain bandwidth product and slew rate?

(f) Why is feedforward compensation usually used only in inverting configurations?

(g) What is a possible advantage in overcompensation?

7. An audio power amplifier had a compensating capacitor (single) of 10 nF, a closed loop bandwidth of 25kHz, slew rate of 4V/μs, closed loop gain of 26dB and SNR of 58dB. Later, C_c was changed to 22nF. Estimate :

(a) the closed loop bandwidth.

(b) the slew rate.

(c) the gain bandwidth product.

(d) the small signal rise time

(e) the 10% - 90% rise time for an output step of 80V.

(f) the SNR (assume that the signal frequency components all fall within the bandwidth of the amplifier and the noise is white).

8. Use the data in Question 7 above but suppose that the compensation capacitor is unchanged (remains at 10nF) and the gain is increased to 32dB.

Estimate:

(a) the closed loop bandwidth.

(b) the slew rate.

(c) the gain bandwidth product.

(d) the small signal rise time.

(e) the 10% - 90% rise time for an output step of 80V.

(f) the SNR (assume that the signal frequency components all fall within the bandwidth of the amplifier and the noise is white).

Review questions

9. Use the compensation characteristic graphs for the 301 op amp (Figures 8 and 9 in notes). Determine the open loop gain at 10 kHz and full power bandwidth for the following external compensation techniques.

(a) single 30pF capacitor

____

(b) single 3pF capacitor

____

(c) single 3000pF capacitor

____

(d) two capacitor

____

(e) feedforward

____

Skill practice 6
Frequency compensation

Suggested duration

3 hours

Tasks

- To observe the variation in closed loop bandwidth and slew rate when the compensation is changed, using square wave testing
- To optimally compensate an op amp by observing its square wave response

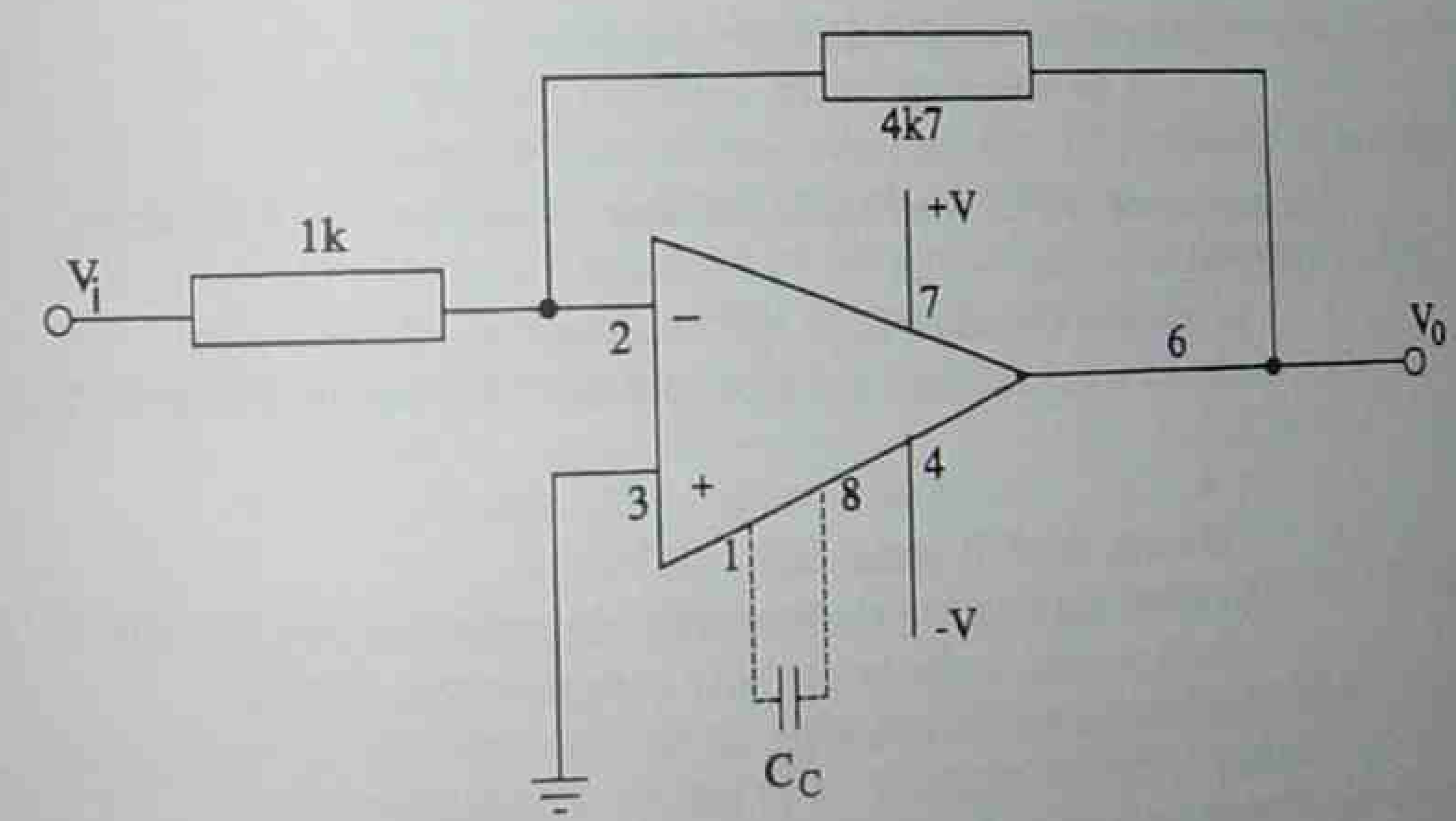
Equipment

- one externally compensated op amp — type LM301 recommended. (Internally compensated types such as 741 or LF351 are *not* suitable for this work.)
- selection of capacitors in the range ≈ 5p to 100p (values not critical)
- selection of resistors (values not critical)
- CRO (preferably with x5 or x10 time base)
- DC power supplies ±15V
- function generator
- variable capacitor range ≈ 3p to 30p

Procedure

Step 1 *Circuit connection for single-pole compensation*

- Connect Circuit A using your closest available value components.



Circuit A Single capacitor compensation

Step 2 *Measurements of 3dB bandwidth and slew rate for C_c = 30 pF using square wave testing*

- Connect and power up the circuit A, using C_c = 30pF.
- Select square wave output in function generator.
- Observe the output on the CRO and adjust the input voltage to get an output of about 1V p-p. This makes it 'small signal' for measuring the bandwidth.
- Adjust signal frequency so that you can easily see the rise and fall of the square wave. Use the x5 or x10 time base switch on the CRO, if available, to get an expanded picture of the rise and fall. A suitable input frequency is probably in the range 5kHz - 50kHz.
  (The exact input voltage and frequency are not important).
- Measure the output 10%-90% rise or fall times, whichever is greater. (Note : if you use the x10 time base switch, you have to divide the timebase 'μs/div' dial reading by 10).
  t_r =
- Calculate the 3dB bandwidth = $0.35/t_r$ = ...
- Observe the output waveform and, in particular, whether the circuit is over, under or properly compensated.
- To measure slew rate, increase the input voltage to get a large (>20 V p-p) or saturated output. Measure the rise or fall in voltage, whichever is greater, (ΔV) and the corresponding run in time (Δt).
  ΔV = ... Δt = ...
- Calculate slew rate = $\Delta V/\Delta t$ = ...

Step 3 *Measurements of 3dB bandwidth and slew rate for C_c = 3 pF using* square wave testing

- In circuit A, change C_c to 3pF or closest available value.
- Measure the rise time and calculate the 3dB bandwidth as in Step 2.
  t_r =
- Calculate the 3dB Bandwidth = $0.35/t_r$ = ...
- Observe the output waveform and, in particular, whether the circuit is over, under or properly compensated.
- Measure slew rate as in Step 2.
  ΔV = ... Δt = ...
- Calculate slew rate = $\Delta V/\Delta t$ = ...

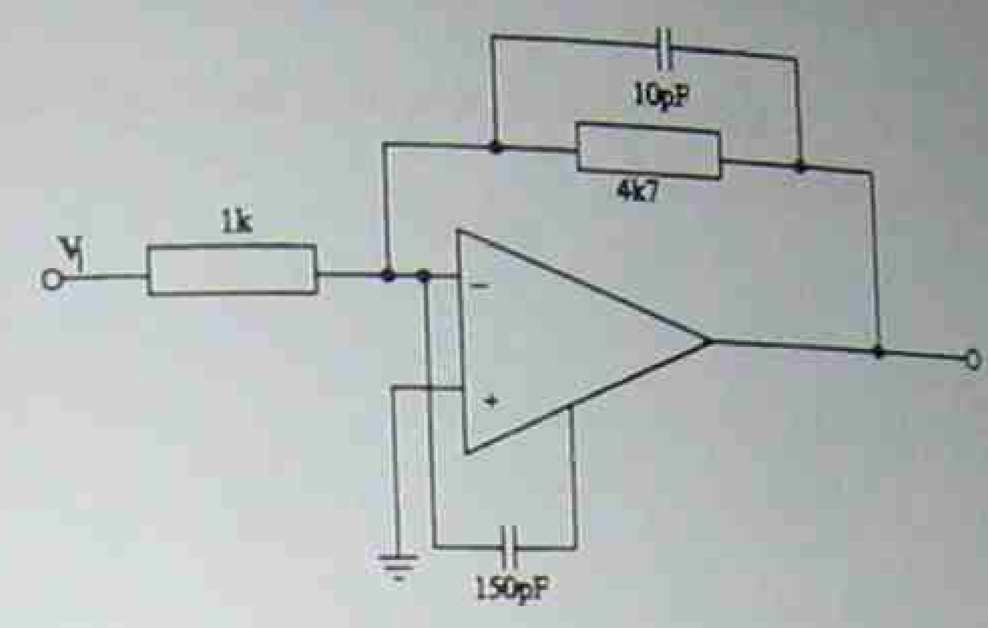
Step 4 *Measurements of 3dB bandwidth and slew rate for C_c = 100 pF using square wave testing*

- In circuit A, change C_c to 100pF or closest available value.
- Measure the rise time as in Step 2.
  t_r =
- Calculate the 3dB bandwidth = $0.35/t_r$ = ...

...and, in particular, whether the circuit is over, under or properly compensated.

- Measure slew rate as in Step 2.
  ΔV = ... Δt = ...
- Calculate slew rate = $\Delta V/\Delta t$ = ...

Step 5 *Measurements of 3dB bandwidth and slew rate for feedforward compensation using square wave testing*



Circuit B Feedforward compensation

- Connect circuit B, using C_c = 150pF or closest available value. Repeat the procedure in Step 2 above.
- Note that, in this method, the bandwidth and slew rate may be so large that you have difficulty measuring it.
- Measure the rise time and calculate the bandwidth as in Step 2.
- Calculate the 3dB bandwidth = $0.35/t_r$ = ...
- Observe the output waveform and, in particular whether the circuit is over, under or properly compensated.
- Measure slew rate
  ΔV = ... Δt = ...
- Calculate slew rate = $\Delta V/\Delta t$ = ...

Step 6 *Determination of optimal single-pole compensation capacitor by square wave testing*

- Connect circuit A, replacing the fixed C_c by a variable capacitance in the range of approximately 3 - 30 pF.
- Observe the output and adjust C_c until the output looks closest to a square wave (fastest rise and fall times, no ringing, maybe a slight overshoot before flattening out). This represents optimal single capacitor compensation.
- Measure the bandwidth as in Step 2.
  t_r =
- Calculate the 3dB bandwidth = $0.35/t_r$ = ...
- Measure slew rate as in step 2.
  ΔV = ... Δt = ...
- Calculate slew rate = $\Delta V/\Delta t$ = ...

Discussion questions

1. Make a table showing the different compensations you studied, and the corresponding bandwidths and slew rates.

2. How does the value of C_c affect the BW and slew rate ?

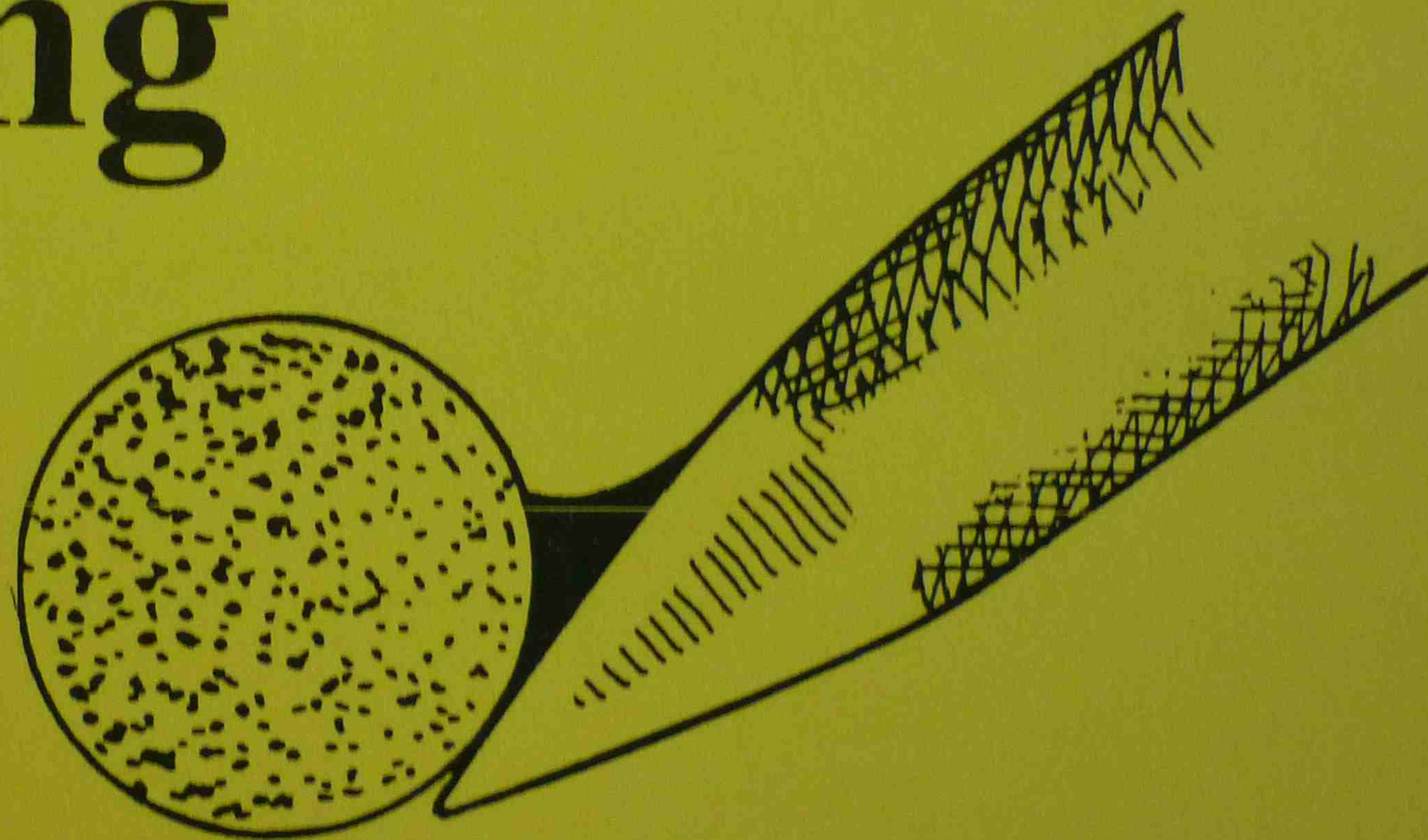
3. Which method gave the highest BW ?

4. Which method gave the highest slew rate ?

5. For single capacitor compensation, check whether the bandwidth and slew rate are both inversely proportional to C_c.

Electronic Hand Soldering

COURSE 7794J



Contents

Aim

The aim of this course is to acquaint the student with basic soldering techniques, and to teach the student how to:

- Prepare a safe work area.
- Prepare and maintain equipment.
- Identify a correctly formed solder joint.
- Load a printed circuit board.
- Solder a wide range of components and hardware.
- Handle sensitive components.

Introduction

As electronics becomes more and more high tech, so does the art of soldering. It is now looked upon as a technology in its self, requiring highly skilled operators. The aim of this course is to increase your hand skills, and to give you an understanding of the standards required to meet the changing needs of industry.

The main cause of equipment failure today is the solder joint, electronic soldering is far more important today, than it has ever been.

The introduction of the microchip has seen the greatest changes in the area of hand soldering. These devices combine hundreds of components in a single package, and as the packaging gets smaller so increases the problems associated with soldering these devices.

Although automation has taken over from hand soldering in the majority of applications, the need for highly skilled operators still exists. Hand soldering is still required for such applications as terminal and connector soldering, as well as retouch and repair areas.

In a computer the failure of a single solder joint could cause it to be totally inoperable. This means the ability to solder well and to recognise unsatisfactory solder joints is critical, if we are to maintain reliable operation of today's high tech equipment.

Safety

Safety

Safety in the work place is of major concern for everyone. It is important that you are able to identify possible safety hazards and know the procedures for reporting and correcting any problems that might occur.

This applies to both large jobs such as wiring up a power distribution board, where contact with high voltages is a potenital danger, to small jobs such as forming a simple solder joint.

Something as simple as soldering two wires together does not seem as if it could present a safety hazard, but the damage a hot soldering iron can cause to skin tissue is horrendous. Every time a soldering iron is left sitting on a bench, instead of being placed back in its holder, the opportunity for such an accident exists.

Remember a large number of potential accidents can be avoided if you, as an individual, follow correct work practices.

Personal Safety

Personal safety is not just a matter of watching out for safety hazards around the work place, but, in the case of an accident, minimising the chances of injury.

This can be achieved by ensuring that you wear clothing and foot wear which are appropriate to the conditions under which you work.

Setting up a Work-station

The setting up of a solder work station, so that everything is laid out in a logical and neat manner is one of the best ways of preventing accidents. Not only are accidents prevented, but it enables you to work in a fast and efficient manner.

If tools are placed so that they are easy to reach and are returned to the same position after use, the soldering process will become a natural action. You won't have to be looking for that pair of cutters, every time you have to cut a component lead. You will just naturally reach for the tool you need, knowing it is in the place it should be.

Having the tools in the right place is only part of the process, having the soldering iron to suit you, is just as important. Forever reaching across the other side of the work bench to get the soldering iron, when you are right handed, is an unsatisfactory situation.

Always have the soldering iron positioned so that it is readily accessible and leads are not laying across the work area.

By spending the time to correctly set up a solder work station, will in the end save time and effort, and will help to make it a safe working environment.

Soldering Process

Why Solder?

Electronic equipment requires good electrical and mechanical connections for reliable operation, so when choosing connection methods these two requirements must be considered.

Another consideration is the ability of the joint to be reworked, and the cost of the equipment required to carry out the reworking of the connection.

There are several methods by which we can join two metal components. We can use epoxy, welding, mechanical fastening or soft soldering, these are only some of the available methods.

Epoxy type glues will give the mechanical strength required, but will not provide good electrical connection. Epoxies also prove extremely difficult to remove.

Welding gives good mechanical and electrical connections, but the equipment required to make the connection is expensive and the components are difficult to rework.

Mechanical fastenings in the long term can prove unsatisfactory, as they are affected by vibration and other mechanical shock.

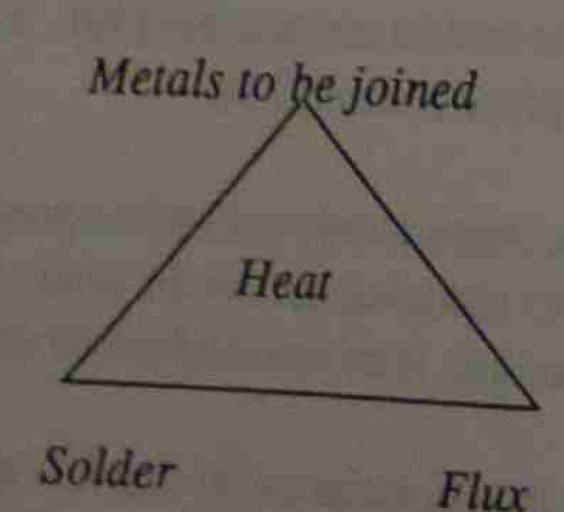
Soft soldering on the other hand provides good mechanical and electrical connection, is easy to rework and the equipment is affordable. There is an added advantage in that soldering also provides protection from corrosion for the connection.

Soldering

Soldering is the process by which two metals can be joined together to form a solid connection. It is a simple process in which flux, solder alloy and heat are applied to the metals to be joined and are then allowed to cool.

The result is a connection in which the properties of the joint are different from those of the original materials, but it has the mechanical and electrical characteristics we require.

Figure 1 shows the constituents of a solder joint.



Solder

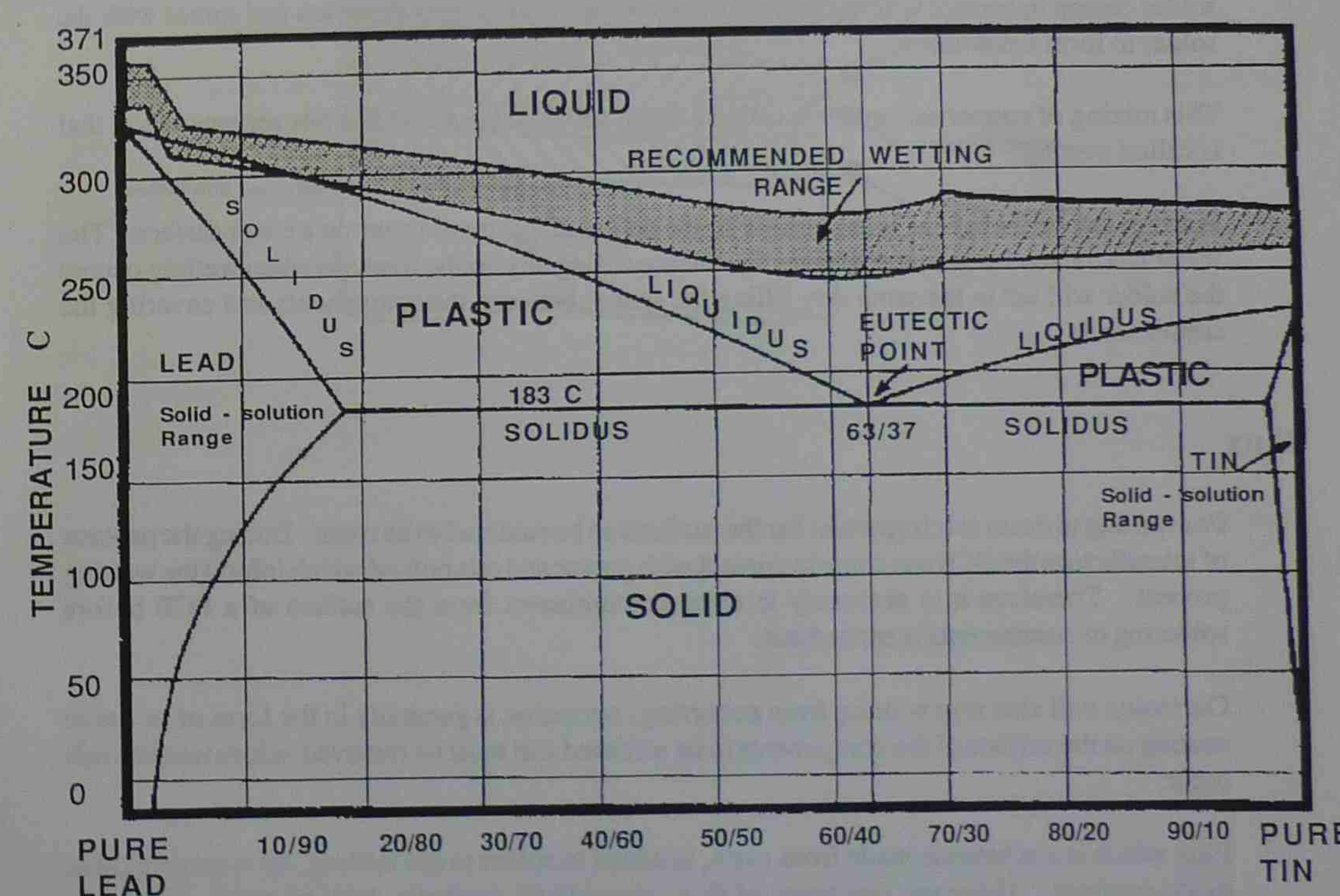
As was mentioned earlier solder is an alloy and an alloy is a mixture of two or more metals. The solder we use for electronics hand soldering is an alloy of tin and lead mixed in a ratio of 60% tin and 40% lead. Although solder is available in other ratios this combination of 60/40 has the characteristics most suitable for electronic soldering.

Characteristics of Solder

The characteristics of solder are different to that of either lead or tin. To start with pure lead has a melting point of 327 degrees celcius and pure tin melts at 232 degrees celcius, yet 60/40 solder melts at 191 degrees celcius. This is lower than lead and tin.

Figre 2 shows the characteristics for lead and tin as weil as the various combinations of the two.

TIN - LEAD FUSION



The *solid region* shows the temperature at which the various alloys are solid.

The *plastic region* is the range over which the various alloys are workable before they turn into a solid. For instance plumbers' solder is a 30/70 solder and has a large plastic range which allows the plumber to position his pipes correctly before the solder sets.

The *liquid region* is the temperatures at which the various alloys are in a liquid state and will flow freely.

The *eutectic composition,* which is 63% tin - 37% lead, is the combination where the plastic region is minimal or non-existent. Eutectic solder is ideally suited to electronic hand soldering, because it changes from solid to liquid almost instantaneously. This reduces the possibility of joint defects due to movement of the components or mechanical stock.

Wetting

Wetting is the process that occurs when components are soldered together. When the molten solder comes in contact with the copper tracks some of the copper dissolves and mixes with the solder to form a new alloy.

This mixing of copper and solder is called a metal solvent action and it is this solvent action that is called *wetting*.

Wetting can be thought as being similar to the action of a drop of water on a clean surface. The water has the tendency to spread over the surface filling any voids, likewise when wetting occurs the solder will act in the same way filling the spaces between the components and covering the copper track.

Flux

For wetting to occur it is important for the surfaces to be soldered to be clean. During the process of manufacture the PCB can come in contact with grease and oils both of which inhibit the wetting process. Therefore it is necessary to clean contaminates from the surface of a PCB before soldering of components is carried out.

Corrosion will also stop wetting from occurring, corrosion is generally in the form of an oxide coating on the surface of the components to be soldered and must be removed before wetting will occur.

Flux which is a substance made from rosin, is added to solder to aid wetting, by removing these oxide coatings. There are two types of flux, natural and synthetic, both of which are highly corrosive at the temperature at which solder melts.

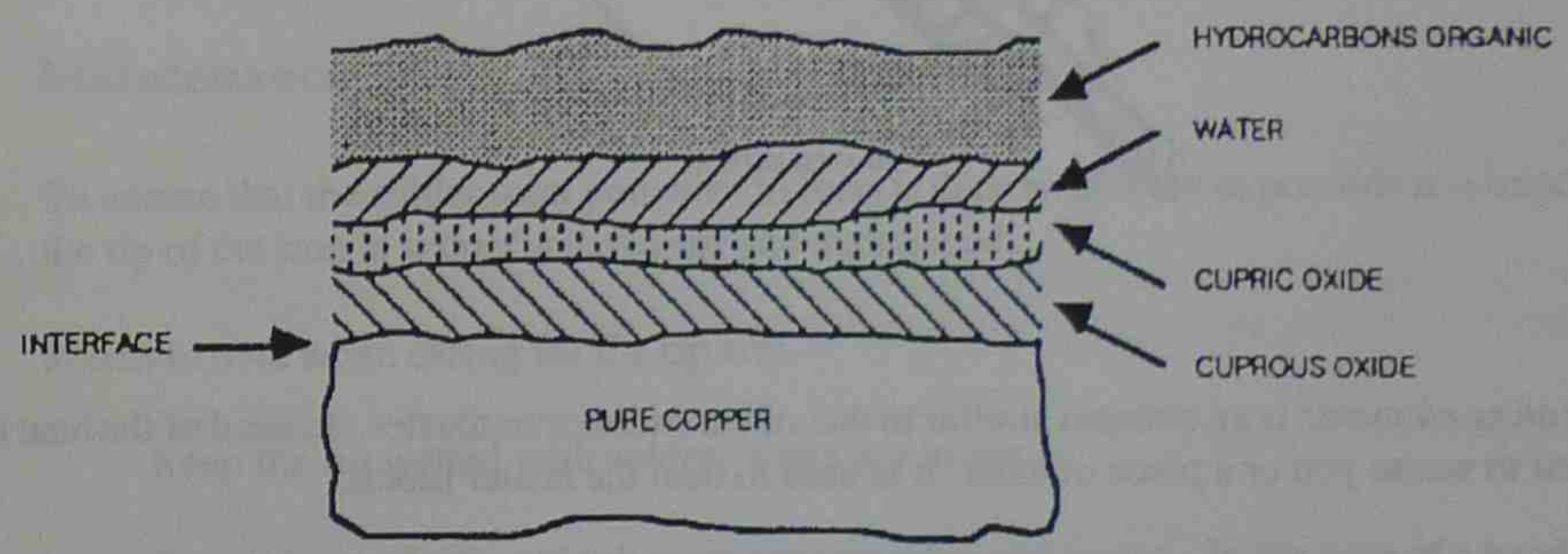
The good thing about flux is that at room temperature it is non-corrosive and therefore it will not damage the components or the PCB.

Flux can be applied to the joint to be soldered from a container or as in most cases it is a part of the solder itself. This is done by placing a core of flux in the middle of the solder, this type of solder is called cored solder.

The result is, as soldering is performed the correct amount of flux is dispensed automatically to the joint. Once the flux is activated by the heat from the soldering iron it removes the oxide coating and prevents a new one from forming.

It should be noted that the temperature at which the flux is activated is lower than the melting point of solder.

Figure 3 shows the layers of oxides and films on an apparently clean copper track.

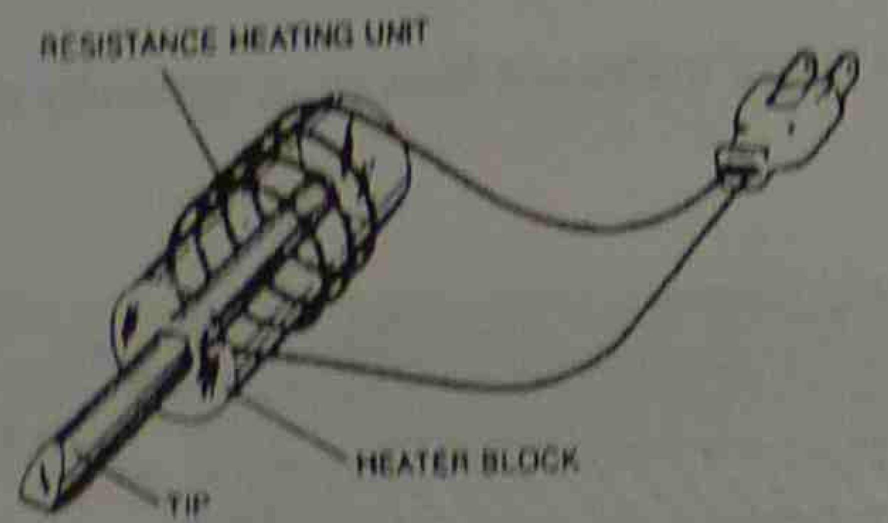


Soldering Iron

Construction

We said earlier that in order to form a solder joint one of the requirements was heat, to melt the solder. The soldering iron is the tool used to generate the heat required and the variety of types of soldering irons is endless.

While there may be a wide variety of soldering irons they basically have three things in common. Each has a heating element, a heater block and a tip.



Heating element: is an element similar to that of bar radiator or toaster. Instead of the heat being used to warm you or a piece of toast, it is used to heat the heater block.

Heater Block: is the storage part of the system, heat from the element is stored in the block, like water in a dam, for use when required. The amount of heat required will depend mainly on the size of the solder joint to be soldered. The larger the joint the more heat required to form that joint.

Tip: is the point of transfer, the heat is transferred from the tip to the solder connection. The shape and size of the tip is also determined by the solder joint to be formed.

Choosing a Soldering Iron

The choice of soldering iron can be a difficult thing. Do I get a hand held iron or one that is part of a work station? What size iron should I get? Do I need a small or large wattage iron? Does it have to be temperature controlled?

Basically the answer to all the above is Yes, but unfortunately the cost of purchasing soldering irons to cover every possible situation would send you broke. The only solution is to purchase a general purpose iron that has a power rating of about 16-25 watts, an iron this size will fill 90% of all your soldering needs.

If you can afford it, a temperature controlled soldering iron, is a valuable tool because it allows you to set the operating temperature of the tip to suit the application. These temperature controlled irons will allow you to solder sensitive semiconductor components, and then turn the temperature up to handle terminations and connections. This allows you to hande a wide variety of situations with a single soldering iron.

The temperature control can be achieved a number of ways, the two most common are:

> a temperature sensor in the tip which controls the power applied to the heating element.
>
> a system that makes use of the curie effect of some magnetic materials. These materials loose the magnetic properties, once they reach temperature, this loss of magnetism is used to cut the power to the heating element.

Temperature control of the soldering iron tip is kept within a few degrees of the set temperature, ensuring that the components being soldered will not be unduly over heated.

Maintenance

To ensure that the solder joint you wish to form is done as quickly as possible it is important that the tip of the iron is properly cared for and maintained.

Points to note when caring for the tip are:

> Keep the tip wetted with solder, when not in use.
>
> Do not leave the tip at high temperatures when not in use. In the case of a hand held iron turn it off, for the temperature controlled irons reduce the operating temperature.
>
> Periodically remove the tip from the iron to make sure it is not seized in the barrel of the iron.

Tip maintenance is a simple process and should be carried out on a regular basis, to ensure maximum life of the tip.

The Solder Joint

Characteristics

The climax of all we have looked at is the solder joint, soldering is not just a matter of melting the solder and applying it to the metals to be joined. It is also knowing what a correctly formed solder joint looks like, its characteristics and the possible defects that might occur when soldering.

The physical characteristics that identify a correctly formed solder joint are:

- wetting of the joint has occurred, on all the surfaces to be joined.
- the sides or fillets of the joint are concave in shape.
- the joint is smooth and shiny.
- there are no voids or cracks in the joint.

Making the Joint

Not only are there certain characteristics to be looked for once the joint has been made, but there is also a correct procedure to be followed in order to make the joint.

Steps in making a solder joint.

- Preparation of the materials to be joined.
- Application of the flux.
- Heat and solder are applied.
- The joint is allowed to cool.
- Residue flux is removed.

Before we start to make the joint it is important that we prepare both equipment and the materials to be joined. Both must be free from contaminants and damage. Contaminates on the iron may get into the joint during the soldering process and cause a defective joint. Likewise contaminates on the components can cause the same sort of problems.

When making the joint, the soldering iron is tinned with a small amount of solder and is placed on one side of the joint. The solder already on the iron not only makes sure the tip of the iron is clean, but also increases the heat linkage area ie. the amount of surface contact between the iron and the joint.

The cored solder is now applied to the joint, but on the opposite side to the soldering iron, the heat from the iron activates the flux and the solder starts to melt. It is important that the solder is not applied directly to the tip of the iron as this will cause the flux in the solder to vaporise and make the joint harder to achieve.

As the solder is applied to the joint we start to look for the first signs to the formation of the joint, wetting. Once it is obvious that the solder has wetted the joint and in the case of a PCB the land area is covered by solder the solder and heat should be removed and the joint allowed to cool.

Inspection

The next step in the soldering process is the inspection of the solder joint. It is important that the joint be free from defects and to check that wetting has taken place. If a faulty joint is discovered steps should be taken to correct the fault. Some of the possible defects that can occur are:

Dry Joint

Disturbed Joint

Overheated Joint

Cold Joint

Insufficient Solder

Excess Solder

The Stripping and Tinning of Wire

With all electronic equipment we need to be able to connect to the outside world. A PCB with all its high tech IC's and components is useless until we connect the power and the necessary signal lines that are needed to bring the circuitry to life.

Connections inside the equipment are usually made by the use of a harness or loom where the wires are soldered to the PCB. Connections to the equipment itself is by cables and specialised connectors.

The weak link in all these connections is the termination of the wire, if the wire is not prepared correctly or the joint correctly formed, the reliability of the equipment will be affected.

In order for the wire to be soldered to either a connector or a solder terminal it must be prepared correctly. Preparation usually takes the form of *stripping and tinning* of the wire.

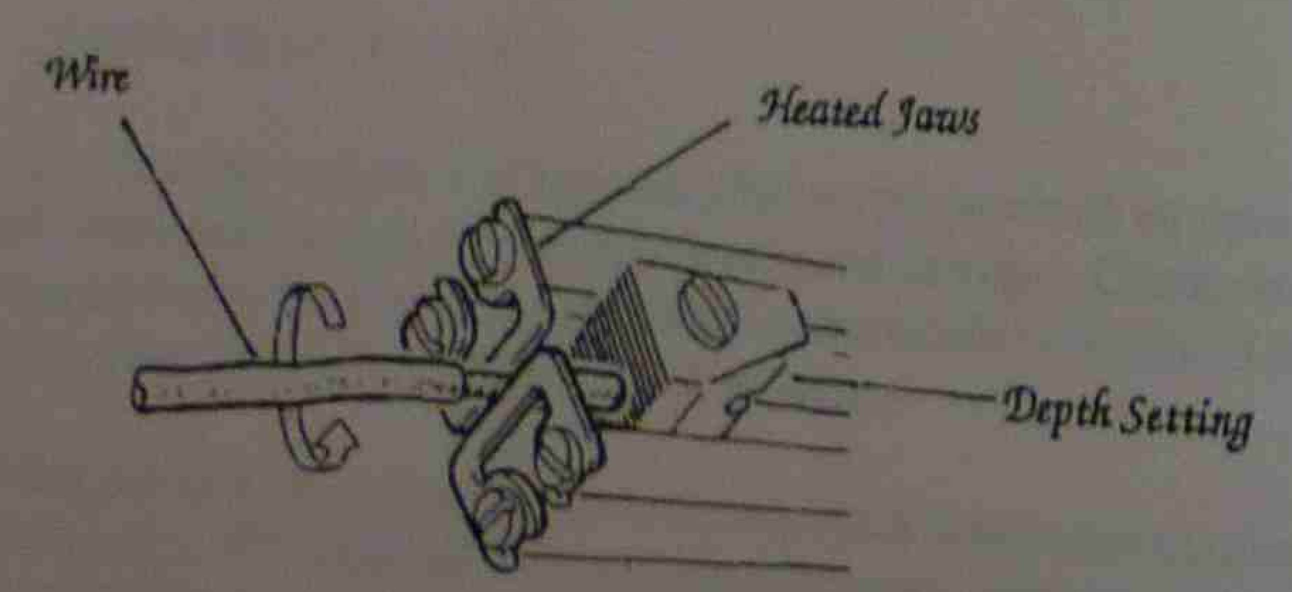
Stripping of the wire involves the removal of the insulation protecting the wire. Tinning is the coating of the wire with a thin layer of solder.

Stripping is usually achieved by cutting through and removing the insulation with wire strippers or cutters. The cutting is either a hot cut in which heated jaws melt through the insulation or a cold cut which the jaws of the cutters cut through the insulation.

The advantage of hot cutting, over cold cutting is the wire is not damaged as much during the stripping process. With cold cutting the possibility exists for the wire to be scored and any damage can reduce the operating life of the wire.

The disadvantage of hot stripping is that a thin layer of insulation material can be left on the bare wire and when soldered can contaminate the solder joint. Cold stripping on the other hand removes all of the insulation leaving a clean working surface.

Figure 3 shows the hot strip process.

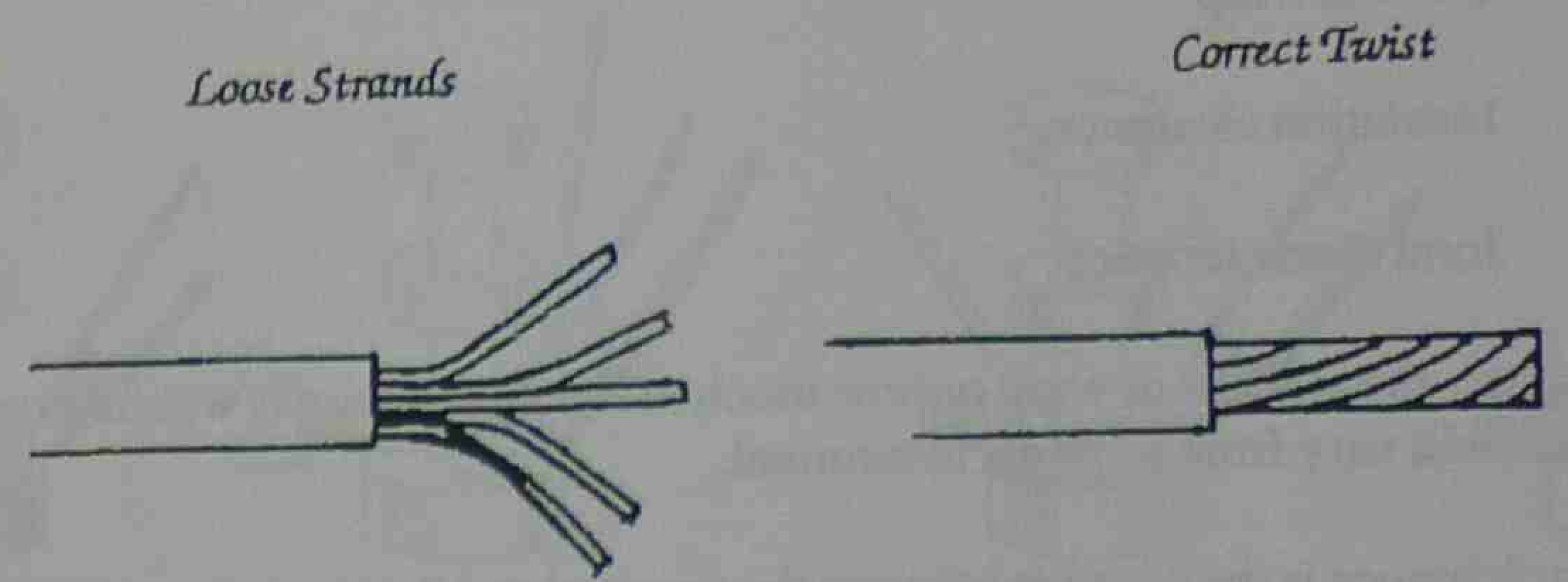


Once the wire has been stripped the individual strands will tend to separate, so before soldering they must be twisted to form a neat bundle again.

When twisting the strands there is two important things to remember:

1. The twist should follow the natural twist of the wire.

2. The twist is not too tight (2 to 3 turns).



Soldering the wire is not simply a matter of putting a bit of solder on the bare wire. Firstly, the flux and solder must not be allowed to wick up under the insulation.

When we do solder, the profile of the individual wire strands must be visible through the wire and there should not be any obvious insulation damage due to heat.

Terminals

Soldering to Terminals

There are a number of characteristics associated with the soldering of wire or components to solder terminals.

These are:

1. Correct wrap
2. Insulation clearance
3. Joint characteristics

Correct wrap - the amount of wrap or how much of the tinned wire is wound around the body of the terminal will vary from terminal to terminal.

Insulation clearance is the distance between the terminal and the insulation of the tinned wire. As a general rule the clearance should be 1-2 wire diameters.

Joint characteristics - the characteristics of the solder joint are basically the same as for the standard solder joint that is

- good wetting
- correct amount of solder
- correct profile

There is one other requirement which is the ability to still see the profile of the wire in the joint, that is, the wire should not be completely hidden by the solder.

There are a number of different types of solder terminals available for use in electronics. These include turret terminals, bifurcated cup, hook, etc.

We will only be looking at the first three of these and discussing the characteristics of each.

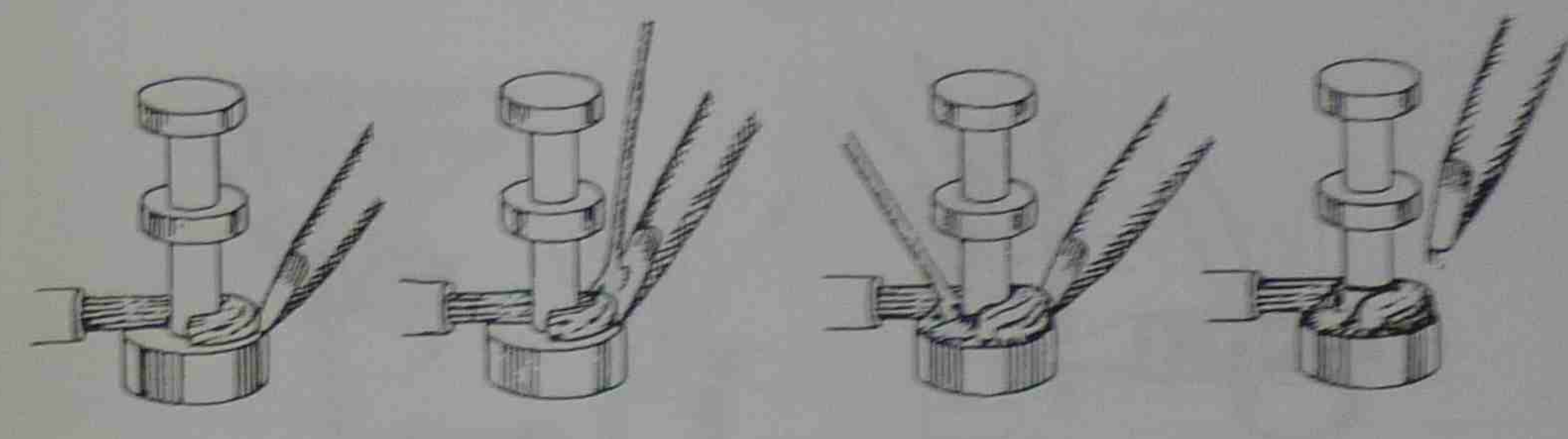
Turret Terminals

Characteristics

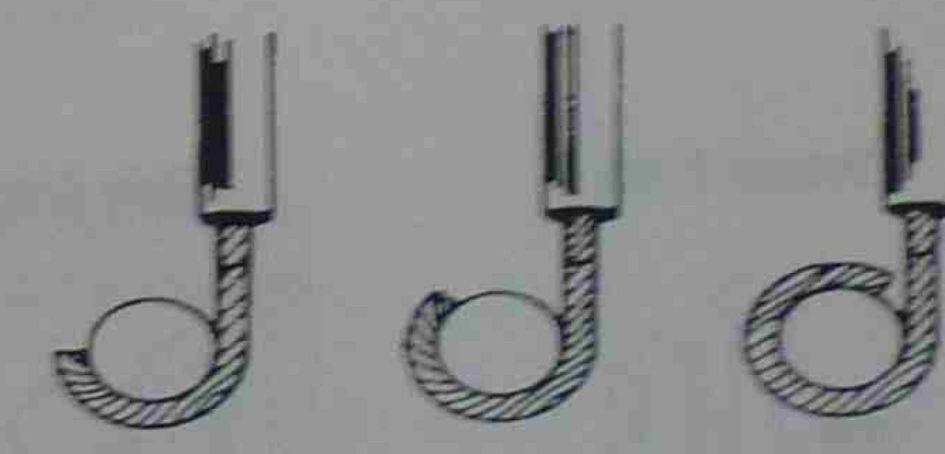
Termination: the wire enters the terminal at 90 degrees, side entry.

Wrap: a wrapping angle anywhere between 180 and 360 degrees is acceptable, with the preferred angle being 270 degrees.

Termination of a Turret Terminal



Soldering iron in contact with both wire and terminal.	Apply solder to form heat linkage.	Form the Joint apply solder to opposite side.	Remove iron and allow the joint to cool.



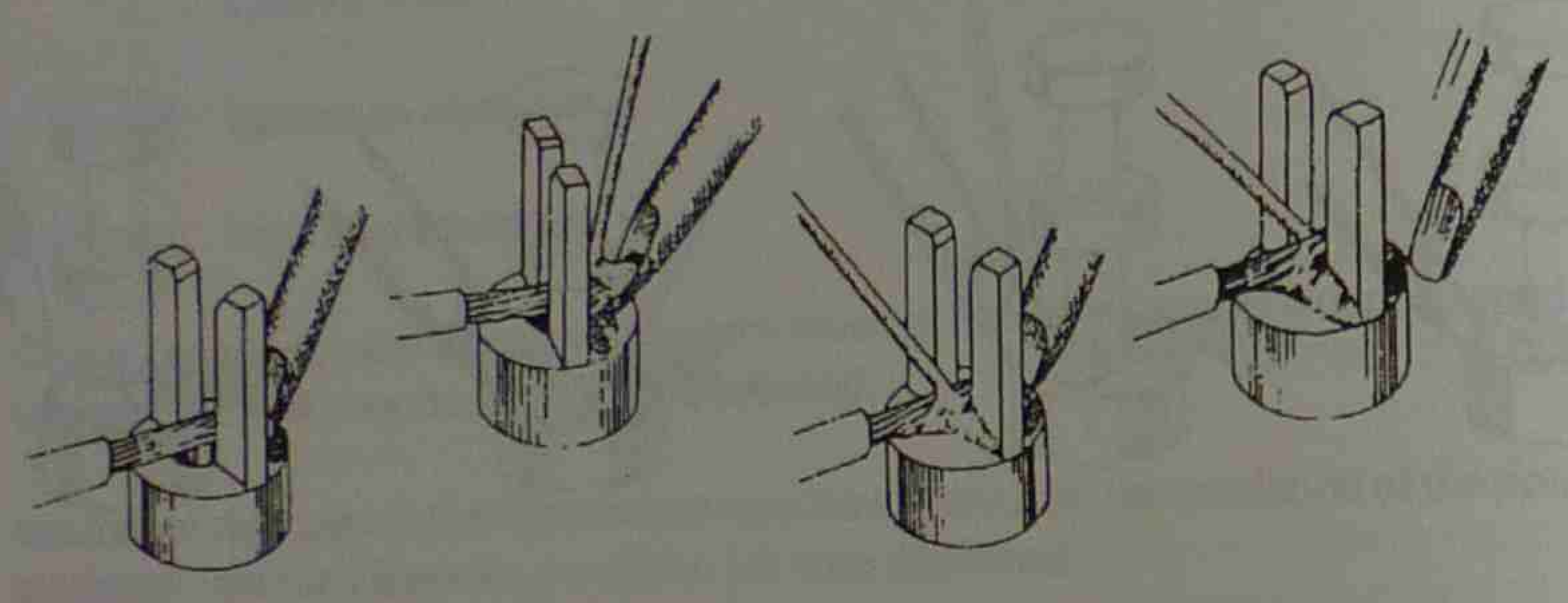
Bifurcated

Characteristics

Termination: the wire can enter the terminal as either a top entry, bottom entry or side entry.

Wrap: the wrapping angle for both the side and bottom entry is 90 degrees and at no time should the wire protrude pass the body of the terminal.

Termination of a Bifurcated Terminal



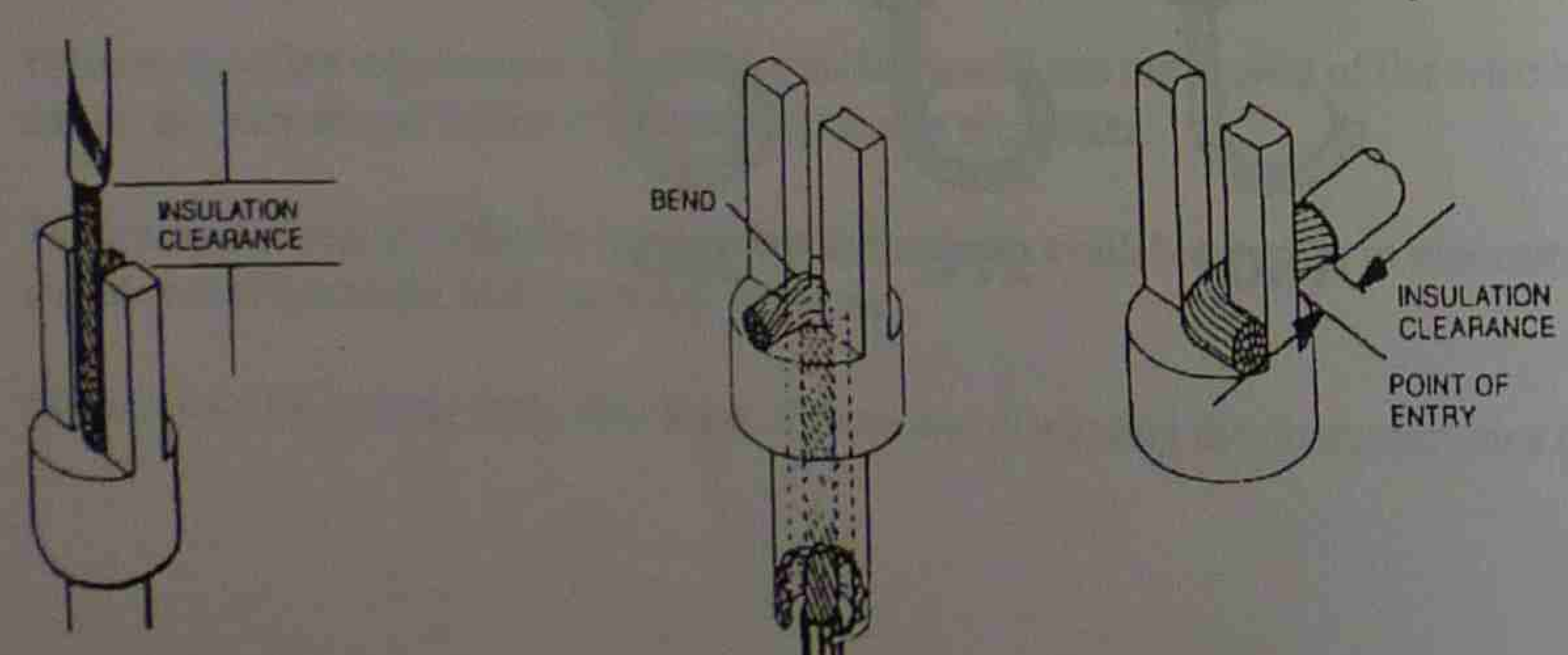
Soldering iron in contact with both wire and terminal.

Apply solder to form heat linkage.

Form the Joint apply solder to opposite side.

Remove iron and allow the joint to cool.

Top Entry | Bottom Entry | Side Entry



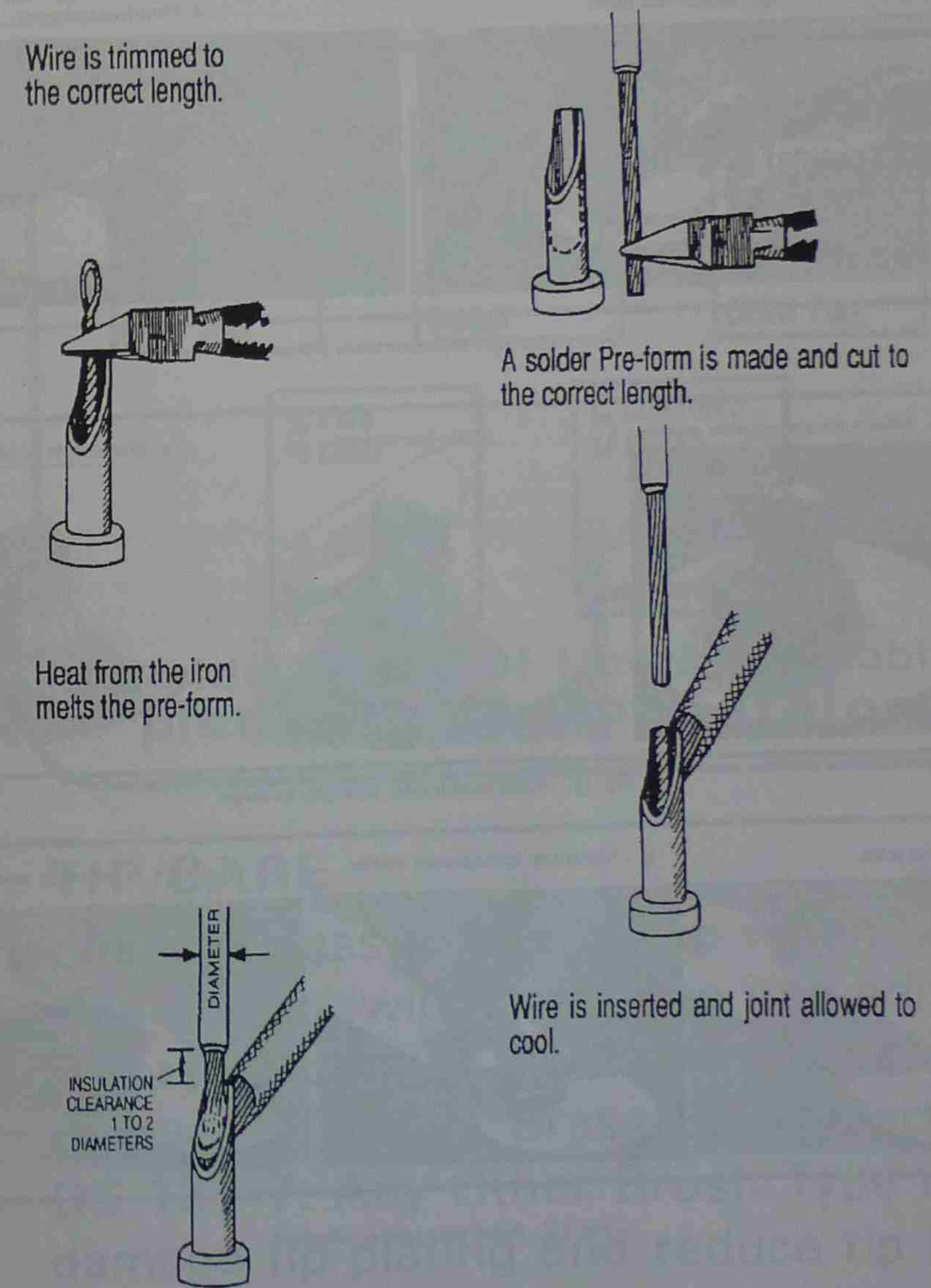
Cup Terminal

Characteristics

Termination: entry is from the top.

Wrap: Nil.

Termination of a Cup Terminal



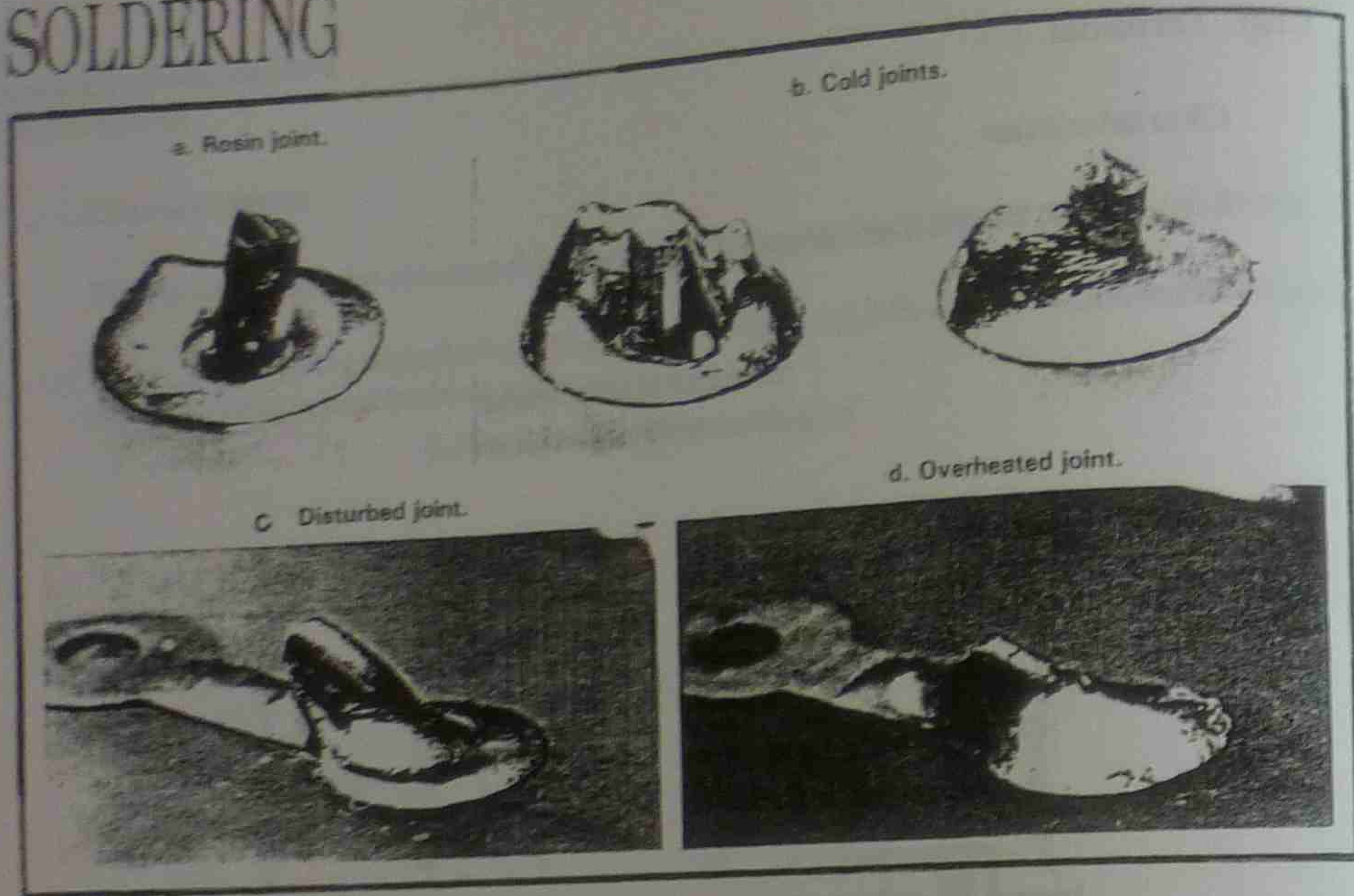


Fig. 10 Unacceptable joints.

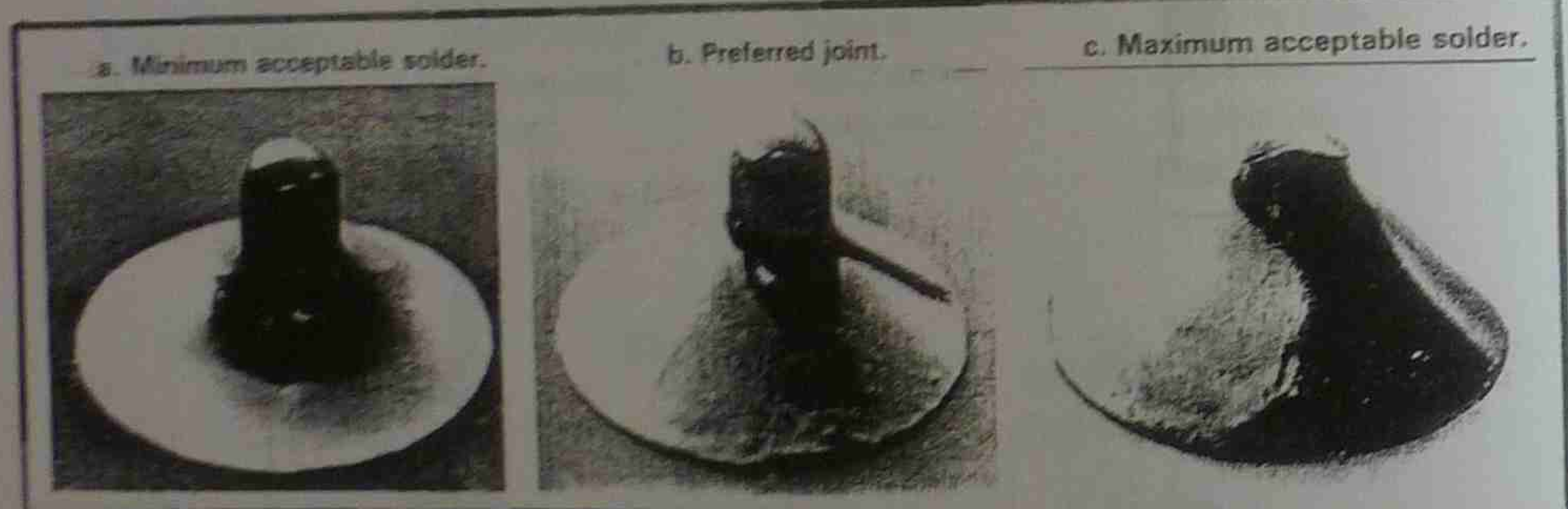


Fig. 11. Axial lead joints, straight-through.

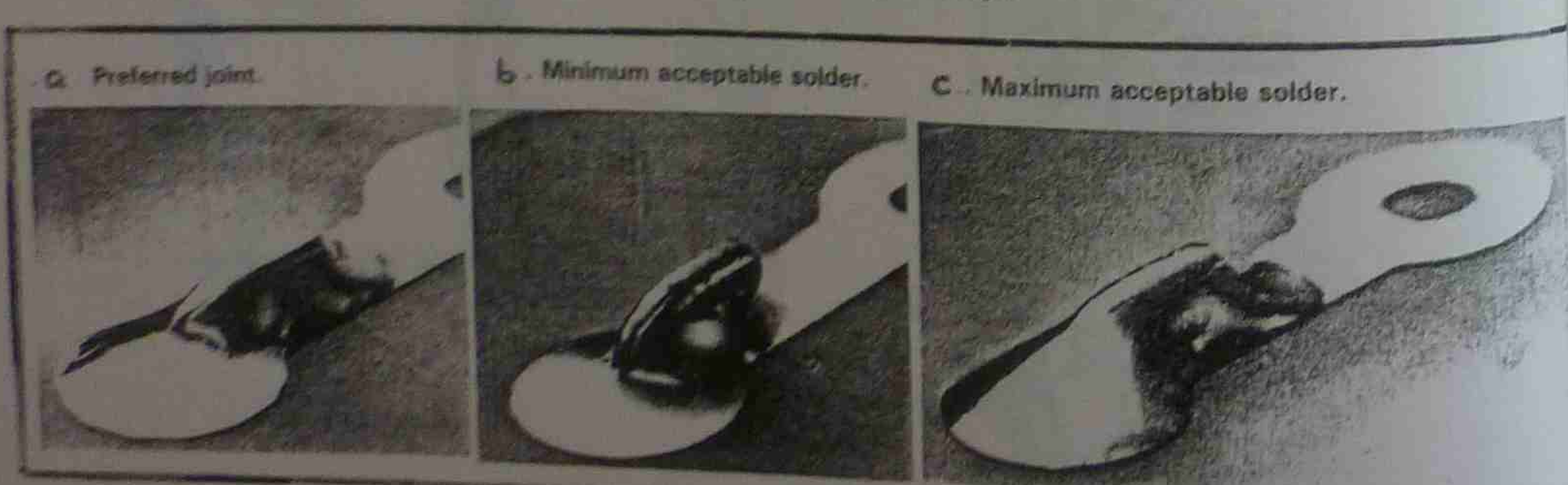
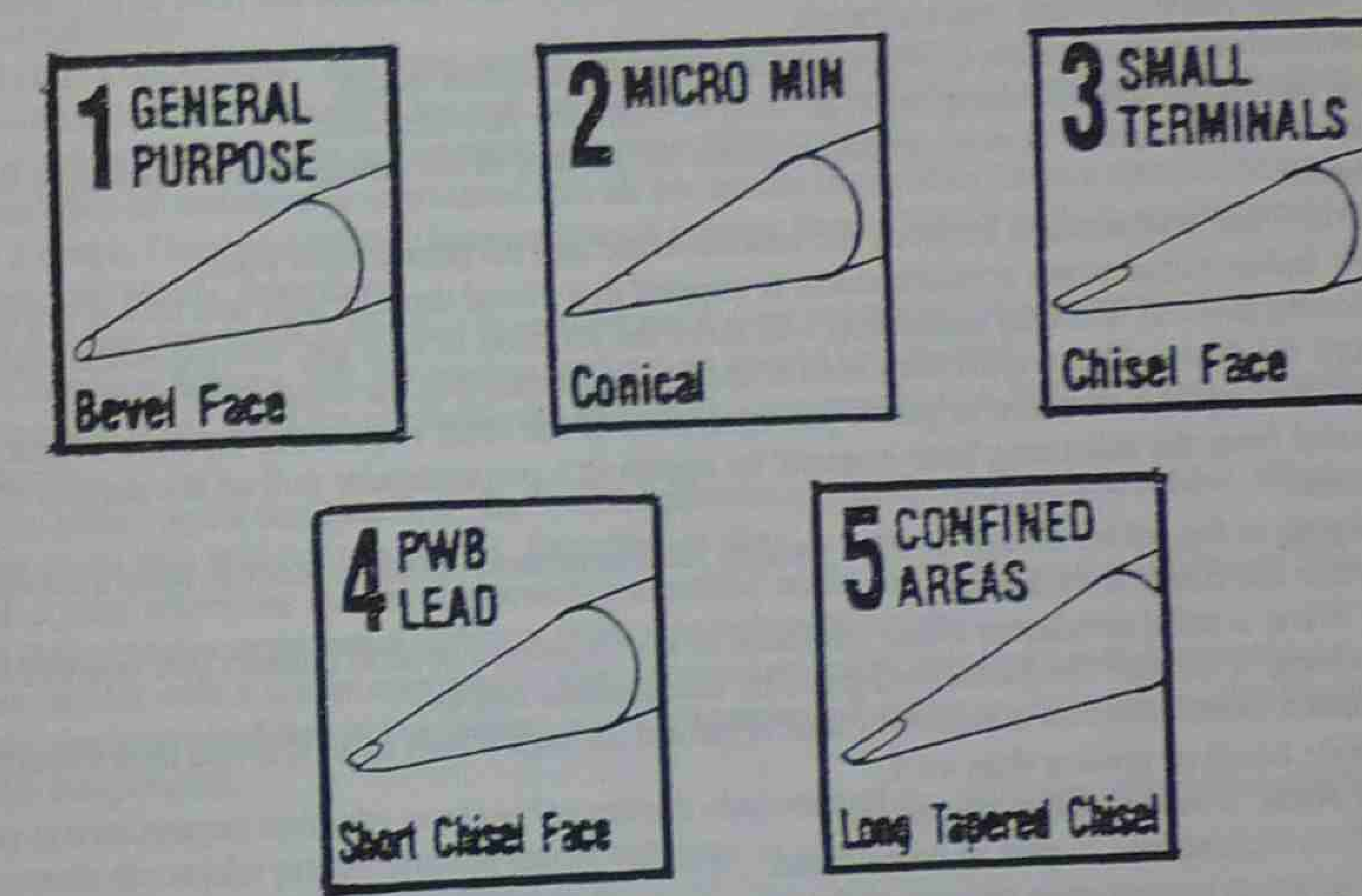


Fig. 12. Axial lead joints, clinched.

ROYEL ARMCLAD

LONG LIFE SOLDERING TIPS



A wide range of tips is available.
please Refer to Catalog

TIP CARE

Leave excess solder on tip when 'idling'. Use wet wiping sponge immediately before soldering. If 'TINNING' is lost, recover by use of Brass Brush Part No. DS 137-7. Any Other Brush Type may damage tip plating and reduce tip life.

Ambient temperature: the temp...

Artwork: refers to the accurately scale configuration used to produce the pri...
i.e. solder resist artwork, track artwork.

Baselaminate: the insulating support holding the print circuit tracks.

Base material: the insulating material used to make the baselaminate.

Blister-Interlaminor: a void produced under the surface of the baselaminate that makes the layers to separate.

Blister-Foil: a separation between the baselaminate and the print circuit track.

Blow holes: occur when contamination is trapped in a plated through hole and heat is applied during the soldering process, the heat expansion will force the junction to vent gas, leaving small holes in the joint (s).

Board thickness: the overall thickness of the base material plus the conductive material.

Bond Strength: a measure in grams per square cm of the force required to separate a layer of conductive material from the adjoining base material by applying a perpendicular pull on the surface of the conductive material.

Bridging is the solder bridge or fillet used to joint to or more joints in order to form a conductive path between the conductive areas.

Bus Wire: a solid conductor either insulated or non-insulated used to connect two or more terminals.

Checking: a surface condition evidenced by fine hairline cracks.

Clinched Lead: where a component lead is passed through a hole in a PCB and bent towards the track at an angle equal or greater than 45°C.

Cold Joint: a solder defect caused by

- Insufficient heat to part to be soldered
- Incorrect temperature control of the iron used
- Insufficient flux
- Surfaces to be soldered are not properly cleaned and free from all oxide coating. In a cold joint, the required metallic lustre is missing and has the appearance of roughly piled up solder with a granulated surface that looks chalky and has a frosty look.

Cold solder: very poor wetting due to insufficient heat. The fillet does not blend into the connecting point and the solder has a lumpy appearance.

Component: any electrical/electronic device that has distinct electrical characteristics and terminals or lead-outs, which allow them to be connected to form a circuit.

Component side (Side 1): the side of the PCB upon which components are mounted.

Component leads: the solid uninsulated wires protruding from the body of the component giving conductive connection to the outside world.

Component part: electrical/electronic devices, e.g. an integrating circuit (IC), transistor etc.

Conductive foil: a form of conductive material that covers both sides of the base material, also known as the etch.

Conductive pattern: a pattern of conductors in a conductor layer.

Conductor side: (Side 2) the side of the PCB that contains the conductors.

Connection: the means by which a connection is made to a conductive pattern.

Corrosive: a chemically reactive area that has the ability to wear away gradually.

Crazing: a condition that exists in the form of connected white spots or crosses on or below the surface of the baselaminate reflecting the separation of fibres in the glass cloth and connecting via interconnections.

Crystallized solder joint: caused by movement between the conductor and its connecting point during the soldering activity while the molten solder is solidifying or the joint is reheated. The joint has a dull, heavily pitched surface.

Cross Latching is the breaking up of large conductive areas by use of a pattern of voids in the conductive material.

Delamination: the separation of layers of PCB material within the laminations.

Dents: depressions in the copper foil, which do not significantly decrease the thickness of the copper foil and provided they are not widespread, they are acceptable.

De-wetting: the solder initially covers the surface of the base metal before pulling back, leaving a thin solder film with lumps and ridges of solder at random, on the surface. De-wetting looks like patches of water in a greasy surface. The angle at the edge of the ridges is characteristically high. De-wetting is usually caused by contamination in the materials being soldered. In de-wetting, solder appears as droplets or balls withdrawn from previously wet adjacent areas or never wetting them at all.

Double sided: a PCB with conductive material on both sides that may not be connected together at various points by plated trough holes.

Dross: oxide and other contaminants that form on the surface of molten solder.

Dry joint: (non-wetting): where the solder has not wetted the metal surface due to severe contamination of the solder area e.g. infringement of the solder mask into the soldering areas.

Eutectic Alloy: an alloy of metals so proportioned that all the metals in the alloy have a specific melting point rather than a range. They possess the characteristic of passing from a liquid cycle to a solid cycle to without an intermediate plastic formation.

Excessive solder profile: too much solder has been used but it does not necessarily mean the joint is defective. It does however indicate that the process parameters should be adjusted.

Fixing Hole (mounting hole): used for mechanically mounting a component to a PCB or fixing the PCB assembly to a chassis.

Flux: a chemically active agent that speeds up the wetting process of metals with molten solder. When heated it removes oxide (but will not remove oil, fingertips or dirt).

Fractured joint: a joint where the lead or component has moved while the solder was solidifying. The joint will have a dull granular appearance and may show signs of spiral cracks.

Granular solder: solder with a coarse appearance like a grained structure that lacks metallic lustre. Usually due to the unclean condition of the joining areas, contaminated solder or the molten solder being of an excessive high temperature.

Hand soldering is a technique normally confined to rework stations and the insertion of large components. The operator controls the solder profile and the solder volume.

Icicle: an undesirable protrusion from a solidified joint.

Land: the portion of a PCB used for making electrical connections between the conductor pattern and the components.

Legend: the lettering or symbols shown on the PCB indicating part numbers, component locations etc.

Machine soldering: a mechanical method of mass soldering PCB assemblies.

Maximum acceptable solder profile: maximum amount of solder that would normally be accepted.

point rather than a range. They possess the characteristic of passing from a liquid cycle to a solid cycle to without an intermediate plastic formation.

Excessive solder profile: too much solder has been used but it does not necessarily mean the joint is defective. It does however indicate that the process parameters should be adjusted.

Fixing Hole (mounting hole): used for mechanically mounting a component to a PCB or fixing the PCB assembly to a chassis.

Flux: a chemically active agent that speeds up the wetting process of metals with molten solder. When heated it removes oxide (but will not remove oil, fingertips or dirt).

Fractured joint: a joint where the lead or component has moved while the solder was solidifying. The joint will have a dull granular appearance and may show signs of spiral cracks.

Granular solder: solder with a coarse appearance like a grained structure that lacks metallic lustre. Usually due to the unclean condition of the joining areas, contaminated solder or the molten solder being of an excessive high temperature.

Hand soldering is a technique normally confined to rework stations and the insertion of large components. The operator controls the solder profile and the solder volume.

Icicle: an undesirable protrusion from a solidified joint.

Land: the portion of a PCB used for making electrical connections between the conductor pattern and the components.

Legend: the lettering or symbols shown on the PCB indicating part numbers, component locations etc.

Machine soldering: a mechanical method of mass soldering PCB assemblies.

Maximum acceptable solder profile: maximum amount of solder that would normally be accepted.

Measling: a condition that exists in the form of discrete white spots or crosses on or below the surface of the baselaminate reflecting the separation of fibres in the glass cloth at weave.

Minimum acceptable solder profile: minimum amount of solder present to make a successful joint.

Mounting Hole (fixing hole): used for mechanically mounting a component to a PCB or fixing the PCB assembly to a chassis.

Non-wetting (dry joint): where the solder has not wet the metal surface due to severe contamination of the solder area e.g. infringement of the solder mask into the soldering areas.

Alternatively poor wetting: where the solder makes a large contact angle with the conductor, the surface is incontinuous with irregular with irregular non wet areas exposed.

PCA: a printed circuit assembly.

PCB: a printed circuit board.

PTH: a plated through hole.

Plated-through-hole (PTH): an interfacial or interlayer connection formed by depositing a conductive material on the side of a hole through the laminate layers.

Plating is a process consisting of non-chemical or electrochemical depositing of metal in all or part of the conductive pattern.

Polarizing slot: the slot in the edge of the PCB used to ensure proper insertion and location in a mating connector.
Pinholes: small holes occurring as imperfections which penetrate into the solder. Providing they are not widespread in a particular PCB they are acceptable.
Pre-heating: an immediate heating operation used to raise the temperature of a PCB assembly above room temperature to reduce the thermal shock of an elevated solder temperature.
Printed Circuit Assembly (PCA): is a PCB on which separately manufacturing components have been added.
Printed Circuit Board (PCB): the board formed by the baselaminate and the print circuit track.
Printed edge contacts: patterns extending the edge of the PCB intended for mating with external contacts mostly gold plated either on copper or on nickel plated copper to reduce the cost of gold.
Reflow: refers to all the techniques where solder is preplaced on a substrate and heat applied subsequently. The heating methods used are vapour phase, infra red and hot belt. Reflow methods have a tendency to make the joints dull in appearance which is caused by the growth of lead dendrites and is a cosmetic effect which does not influence the integrity of the joint. The profiles tend to be of minimum acceptable solder.
Residue: excessive substance(s) remaining after soldering such as flux or oil.
Resin joint: where the joint has a layer of solidified flux or dirt between the solder and the surfaces to be soldered which prevents a sound electrical and physical contact. See rosin connection
Rosin connection: a layer of solidified flux between the conductors that has no metallic or electrical continuity. This gives a high resistance joint with no mechanical strength.
Side 1: the side of the PCB that contains the components.
Side 2: the side of the PCB that contains the conductors.
Single sided board: a PCB with conductors on one side only. Side 1 is the component side. Side 2 is the soldered side.
Solder: a fusible alloy consisting essentially of tin and lead used for the purpose of joining two or more metals at temperatures below their respective melting points. Solders that melt readily are called soft solders.
Soldered joint: a joint where sufficient solder has been applied to form a concave fillet between the terminal pad and each side of the component wire. The contour of the wire should be visible. A concave fillet is unacceptable.
Solderability: the property of a metal surface that allows it to be wetted by solder.
Solder Impurities: trace metals appearing in a solder bath during usage.
Solder splashes: drops of solder in areas that should be free of solder: a most undesired condition.
Tinning is a solder coating of the conductive paths and terminals primarily to minimize in-process oxidation and enhance solderability.
Twist: in a PCB, is the bending or curving distortion from a true plane surface, in a direction parallel to the diagonal between two opposite corners of the laminate sheet/PCB assembly.
Unacceptable solder profile: none or too little solder
Warp: in a PCB, is the deviation from flatness characterized by a roughly cylindrical or spherical curvature such that if the board is rectangular, its corners or edges are not in the same plane as the major surfaces of the board.
Wave soldering is a process where the conductor pads of a PCB are brought into contact with the surface of continuously flowing circulated solder. The joints tend to be bright and with a fuller profile than those produced with reflow. Device orientation with respect to the wave direction of travel can affect the joint profiles. Normally in the leading edge joint the solder is concave and in the trailing edge joint is convex. To prevent voids and bridges in the joints, dual or vibrating wave-soldering equipment is used.
Wetting is the coating of the surfaces being soldered by a concave layer of solder correctly adhered at the ideal angle of 40°. Between 40° to 70° the angle is average. Below 40° the angle is good. More than 70° is not acceptable

PREPARING AND MAINTAINING THE SOLDERING IRON

MATERIALS NEEDED:

Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, ⅛″ diameter (3mm)
Solder, rosin-cored, .028″ (22 ga) diameter (.8mm)
Wire stand and cleaning unit
Small screwdriver

PROCEDURE:

1. Before plugging in the iron, insert the tip fully into the heating element and tighten the set screw that holds it in place. Next, saturate the sponge of the cleaning unit with water.
2. Tinning the tip: This must be done before the tip reaches operating temperature. Therefore, while it is heating up, hold a piece of cored solder against each face of the tip. In this way the surfaces will be tinned when they reach the lowest temperature at which solder will melt.
3. Cleaning: Before each use, the tip needs to be cleaned of all residues and oxides. Remove the iron from its stand and wipe the tip lightly on the wire brush to remove any charred flux and solder dross that may be on it. Next, touch it lightly and quickly on the wet sponge to shock off the remaining oxides.
4. After use, and before returning the iron to its stand, put a small amount of solder on the surface of the tip.

Procedure Sheet 2

SOLDERING TO TURRET TERMINALS

MATERIALS NEEDED:

Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, ⅛" diameter (3mm)
Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
Turret terminals
Terminal holding block
Insulated wire, #22, blue
Insulated wire, #20, red
Diagonal cutters
Pliers, round nose
Solvent
Cleaning brush

PROCEDURE:

1. Tinning the wires: Position iron, solder and wire as illustrated in the Handbook. Move the wire across the iron tip, first toward the insulation and then away from it, and off the end. Have the wire end tilting downward so that any excess solder will drain off the wire and onto the tip.
2. With the round nose pliers, form a basic 180° loop on the tinned end of the wire. For this exercise use the 270° wrap and a minimum insulation gap equal to the diameter of the insulated wire when attaching to the terminal.
3. Place the terminal in the terminal holding block and then attach the wire to the terminal.
   A. Place the wire in the lower guide slot and in full contact with the terminal base and post and wrap the loop to 270°.
   B. Cut off excess tinned wire.
   C. Set the loop tightly around the post with long nose pliers for stability.
4. Solder the wire to the terminal using the technique illustrated in the Handbook.
   A. Apply the clean iron tip to the connection at the point of maximum thermal mass.
   B. Make a solder bridge between the iron tip and the work.
   C. Apply solder to a point opposite at which the iron tip contacts; "paint" it on with a light touch and then withdraw the solder.
   D. Withdraw the iron, using a forward wiping motion.

(continued)

Procedure Sheet (continued)

5. The wire must not move while cooling, or a disturbed, faulty joint will result. Allow the joint to cool and solidify before moving it, then clean it thoroughly with solvent and brush and wipe dry.
6. Inspect the joint and compare it with those shown in the reference photographs, using the following Criteria of Acceptance:
   A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, pinholes or porosities.
   B. The surface is free of any lumps, remaining pockets of flux, or granular appearance.
   C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.
   D. The solder fillets on the terminal are concave.
   E. The outline of the individual strands of wire can be seen in the joint. The wire is in contact with both the base and post of the terminal.
   F. There is no copper showing at the end of the wire, and no solder off the edge of the terminal base.

Procedure Sheet

SOLDERING TO CUP TERMINALS

MATERIALS NEEDED:

Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, ⅛″ diameter (3mm)
Solder, rosin-cored, .028″ (22 ga) diameter (.8mm)
Cup terminals
Terminal holding block
Insulated wire, #20, red
Diagonal cutters
Solvent
Cleaning brush

PROCEDURE:

1. Pre-tin the wire; insert it in the cup and measure for the correct lead length; then trim the end to maintain a proper insulation gap.
2. Twist together a short length of solder to make a preform, insert it in the cup and cut it off flush with the top of the terminal.
3. Place the soldering iron at the back of the cup for heating and hold it on until the solder has melted and flux has bubbled to the surface.
4. Insert the pre-cut wire into the molten solder, being sure that the wire bottoms in the cup.
5. Move the wire forward and backward once or twice to allow the solder to flow around the wire and coat the inside walls of the cup, and heat until all flux flows to the surface. Entrapped flux is not permitted.
6. Hold the wire upright and against the back of the cup; then remove heat, allowing the joint to cool. Do not move the wire while the solder is cooling.
7. Clean the connection with solvent and brush, and inspect the joint for quality. Compare it with the joints shown in the reference photographs, using the following *Criteria of Acceptance:*
   A. All the strands of the wire are in the cup.
   B. The surface of the solder is smooth, bright and shiny. There is no evidence of pits, pinholes or porosities. (If the cup has been gold-plated, the solder may have a frosty surface from traces of gold.)
   C. The surface is free of any lumps, remaining pockets of flux, or granular appearance.
   D. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.

(continued)

   E. The solder has flowed out to the edges of the milled faces. (In the case of gold terminals, it is permissible to show a little gold on the outside edges of the face.)
   F. No solder has spilled over and down the sides of the cup, and there are no cracks or voids between the wire and the terminal.
   G. The outline of the wire need not be seen below the point where it enters the terminal.
   H. Proper insulation gap.

Note: An anti-wicking tool may be required by some specifications.

SOLDERING TO BIFURCATED TERMINALS

MATERIALS NEEDED:

Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, 1/8" diameter (3mm)
Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
Bifurcated terminals
Terminal holding block
Insulated wire, #20, red
Diagonal cutters
Pliers, long nose
Solvent
Cleaning brush

PROCEDURE:

1. Pre-tin several wires, leaving no excess solder on the outside of the wire.
2. For a side entry put the tinned wire across the terminal to determine the proper insulation gap and bend area—unless otherwise specified the insulation gap should be the diameter of the insulated wire.
3. Using the terminal post as a gage, use the long nose pliers to bend the wire firmly into a 90° bend.
4. Be sure the wire is placed onto the terminal so that it contacts the base and one post of the bifurcated terminal.
5. Cut the waste end of the wire flush with the base of the terminal; there must be no protrusion beyond the terminal base.
6. Center the formed wire so that it lies over the hole in the base of the terminal and is hooked flat to the base and post. Attach a wire-holding device, such as a piece of tape to hold the wire in place during soldering.
7. Place the soldering iron onto the terminal base and slide gently up to the wire. Melt solder at the junction of the iron tip and wire to form the solder bridge and establish good thermal linkage.
8. Move the solder over to the opposite side of the terminal (entry point of wire) and form a fillet of solder on each side of the wire at the base of the terminal.
9. Bridge the hole between the posts with solder.
10. Remove the iron while the solder is still being applied and flowing; this type of terminal has a large enough thermal mass to continue melting the solder after the iron is removed.

(continued)

11. For the double-side entry, preform 2 wires exactly the same as steps 2 and 3.
12. Place the wires into the terminal, one on top of the other with each wire end pointing in opposite directions. (One wire will be attached to the left post and the other will be attached to the right post.)
13. Attach the wire-holding device to the wires to hold them in place while soldering, and solder as above.
14. Allow the joint to cool, then clean thoroughly with brush and solvent before inspecting.
15. Compare your joints with the reference photographs. They should meet the following *Criteria of Acceptance:*
    A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, porosities or pinholes.
    B. The surface is free of any lumps, remaining pockets of flux, or granular appearance.
    C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.
    D. The solder fillets are concave; the wire strands are distinguishable within the solder, and the wire end is within the limits of the terminal.
    E. The hole in the terminal has been bridged over with solder to prevent collection of any contaminants.
    F. No evidence of bare copper.
    G. The end of the wire does not protrude beyond the diameter of the base, except in special cases with small diameter wire.

Procedure Sheet 5

SOLDERING TO HOOK TERMINALS

MATERIALS NEEDED:

Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, ⅛" diameter (3mm)
Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
Hook terminals
Terminal holding block
Insulated wire, 22 ga, blue
Diagonal cutters
Pliers, long-nose
Solvent
Cleaning brush

PROCEDURE:

1. Pre-tin several 22 gage wires.
2. Preform one wire to a standard 180° loop, place it around the hook, and then cut off the end of the wire with the diagonal cutters so that the wire protrudes above the hook.
3. Set the wrap tightly with the long nose pliers.
4. If necessary attach a wire-holding device to the insulated part of the wire. Do not use excessive tension or the wire will unbend and fly loose from the terminal.
5. Once the wire is properly positioned in the terminal, place the soldering iron tip at the base of the hook and gently slide the tip up to meet the wire.
6. Apply solder at the junction of the iron tip and the wire to create thermal linkage.
7. Move the solder to the other side of the wire and tin the cut-off end to cover bare copper. This terminal uses the least amount of solder. If controlling the solder quantity is difficult, use a solder of a smaller gage.
8. Pre-form 2 wires to a 180° loop and cut off to the proper length to fit the terminal. Insert one wire from the left side of the terminal and the other wire from the right side of the terminal.
9. Position the wires and hold in place with the wire-holding device; use minimum tension on wires.
10. Slide the soldering iron tip from the base of the terminal upward to lightly contact the wire. Establish thermal linkage and feed a slight amount of solder between the iron tip and the wire.

(continued)

11. Move the solder to the opposite side of the terminal and tin the bare copper end of the wire. Allow sufficient cooling time to prevent disturbed connections.
12. Clean and inspect the joints. They should meet the following *Criteria of Acceptance:*
    A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, pinholes or porosities.
    B. The surface is free of any lumps, remaining pockets of flux, or granular appearance.
    C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.
    D. The solder fillets are concave.
    E. A proper insulation gap exists on the wire, and the individual strands of wire are visible beneath the solder.
    F. There is no copper showing at the end of the wire, and the end does not protrude beyond the diameter of the terminal.

6 Procedure Sheet

SOLDERING AXIAL LEAD COMPONENTS TO PRINTED CIRCUIT BOARDS

MATERIALS NEEDED:

Printed circuit board, single sided
Resistors, ½-watt
Resistors, ¼-watt
Soldering iron, 30-watt, pencil-type
Soldering iron tips, plated, chiseled, ⅛" (3mm) and 1/16" (1.6mm) diameters
Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
Diagonal cutters
Pliers, long nose
Lead forming tool
Solvent
Cleaning brush
Abrasive stick

PROCEDURE:

1. *Clean all resistor leads and circuit pads before attempting to solder.* Use the abrasive stick to remove all heavy oxides from the surfaces; then clean with solvent after the abrasive.
2. Using the ½ watt resistors first, form the leads with the lead forming tool as illustrated in the Handbook.
   A. Adjust the tool to span the distance between the holes on the circuit board on the large circuit.
   B. Place the component body between the forming posts and bend first one lead and then the other to 90°
   C. Wipe the leads with solvent to remove oil or grease
3. Carefully insert the component leads into the holes of the *large* circuit until the component body is flush to the board surface.
4. Bend the leads in the direction of the circuit run to an angle of approximately 45° using the long nose pliers. Do not pull up on the lead as this will pull the stress relief of the component lead into the hole on the component side of the board, causing tautness in the lead. This condition is a discrepancy and is known as "rainbowing."
5. Using the diagonal cutters, cut the lead by entering from the side (parallel to the board). *DO NOT* cut leads with the diagonal cutters pointing down toward the

(continued)

circuit. Finish clinching the lead with a non-metallic tool until the end of the lead is in contact with the circuit. The lead length of a full-clinched lead should be approximately the same as the diameter of the circuit pad. Circuit pads with no circuitry attached should have a lead length approximately equal to the radius of the pad and may be straight-through or semi-clinched.

6. Clean, form and assemble all components to the board to complete a row before soldering. This will provide better continuity in learning these new techniques and will reduce the soldering dwell time to the required 2 seconds or less per connection.

   There are 2 rows of circuitry for the components, and assembly will be as follows:

   Row 1—Large circuit: Place a ½ w resistor in every space and full clinch the leads to the pads that have circuitry leading to them. Isolated pads will be semi-clinched or straight-through termination.

   Row 2—Small circuit: All leads will be straight-through termination (¼w resistor).
7. Solder the straight-through leads first. Bring the iron onto the joint from one side and then come in from the other with solder, as illustrated in the Handbook. The tip should always be in contact with *both* the lead and the pad. Remember, the lead is the principal thermal mass in the joint.

   After the solder is "painted" on, remove first the solder and then the iron, sweeping the iron tip over the end of the lead to cover it with solder.

   It is most important to develop a "light touch" with the iron, barely touching the pads with no more pressure than would come from the weight of a pencil balanced on its point.
8. Now solder the clinched leads, observing the correct motion of the tip as shown in the illustration. With a clinched lead joint your *workpiece indicator* is the length of the solder flow along the conductor. An excessively long flow indicates too long a heat dwell time, and the possibility of heat damage at the joint or component.
9. Clean all joints thoroughly with solvent and brush, wipe them dry, and then inspect them. A good technique is to "roll" the board around under an overhead light. This directs the light on every spot and quickly reveals any pits or discontinuities on the surface, or flux residue left on the board.

   Compare your joints with the reference photographs. They should meet the following *Criteria of Acceptance*.

FOR ALL JOINTS

A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, pinholes or porosities.

(continued)

B. The surface is free of any lumps, remaining pockets of flux, or granular appearance.

C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.

D. The solder fillets are concave.

E. The profile of the lead is clearly visible under the solder.

FOR STRAIGHT-THROUGH LEADS

F. The lead should protrude above the board a distance at least equal to the radius of the pad, but not exceeding the pad diameter.

G. The end of the wire should be visible above the solder, and tinned.

FOR CLINCHED LEADS

H. With single leads, clinching is done in the direction of the circuitry, with the end of the lead in contact with the pad. The lead should protrude a distance equal to the pad diameter.

I. When two leads are clinched on the same pad, they should be bent toward each other so that both can be soldered in one operation.

J. The outline of the lead is visible within the solder.

K. Solder has flowed along the conductor far enough to cover the end of the wire and leave a concave fillet at the end.

SOLDERING DUAL INLINE PACKS (DIP'S) TO PRINTED CIRCUIT BOARDS

MATERIALS NEEDED:

- Printed circuit board, double sided
- IC's—dual inline packs
- Soldering iron, 30-watt, pencil-type
- Soldering iron tip, plated, chiseled, 1/16" diameter (1.6mm)
- Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
- Pliers, long-nose
- Solvent
- Cleaning brush

PROCEDURE:

1. Prepare the DIP by first bending each row of leads so that they fit the holes in the board. To make the bend, the body of the component should be carefully held with the fingers, or pliers, and then each row of leads pressed down onto a flat surface, as illustrated in the Handbook. Use no more pressure than necessary to create the correct angle.
2. Mount the DIP on the component side of the board, hold it in with a finger and turn the board over. Semi-clinch the two leads on opposite corners to hold the DIP in place.
3. Solder the leads using a 1/16" iron tip, and progress down the row skipping around so that no two leads next to each other are soldered consecutively. This prevents heat buildup.
4. Apply the iron tip to one side of the joint, being sure it contacts both the lead and pad surfaces, then apply solder to the opposite side.
5. Observe the joint carefully as the solder flows. A mound will build up during the first second or second and a half, and the hole will appear to be full. It is not.
6. Hold the solder and the iron on the joint. The solder pool will drop, indicating that solder has flowed through the hole and wet the pad and lead on the opposite side. (See the illustration in the Handbook.) The solder will then mound up again on top.
7. Remove the iron with an upward wiping motion to cover the end of the lead. Again, practice using the light touch; pads are easily lifted by too much pressure. You should be off the joint within three seconds.

(continued)

8. Clean the joints and board thoroughly with solvent and brush or spray unit, and inspect them, comparing your results with the joints shown in the reference photographs. They should meet the following *Criteria of Acceptance:*
   A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, pinholes or porosities.
   B. The surface is free of any lumps, remaining pockets of flux, or granular appearance. (With gold-plated leads, the appearance of the solder may be slightly frosty due to gold amalgamation.)
   C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.
   D. Concave fillets exist on both sides of the board. On the component side, the filet should extend only up to the wider part of the lead.
   E. On the circuit side of the board the end of the lead is tinned and plainly visible under the solder.

SOLDERING TO-5 TYPE PACKAGES TO PRINTED CIRCUIT BOARDS

MATERIALS NEEDED:

Printed circuit board, double sided
IC's—TO-5
Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, ⅛" diameter (3mm)
Solder, rosin-cored, .028" (22 ga) diameter (.8mm)
Soldering flux
Pliers, long nose
Diagonal cutters
Solvent
Cleaning brush

PROCEDURE:

1. Preform the leads: Use a quarter-inch rod (6.5mm), the abrasive stick, or a pen or pencil and force this down between the leads toward the base of the component. This will cause the leads to spread evenly to a ¾ inch diameter (19.05 mm) at their ends. (See illustrations in the Handbook.)
2. Using the diagonal cutters, cut each lead in the circle so that it is slightly shorter than the one next to it. Then clean the leads with solvent.
3. With the long nose pliers bend the ends of the leads parallel to the body, and insert the component into the holes in the board. Put the longest lead in first and rotate the TO-5 to align and insert each lead in turn. Then cut to the proper length for soldering, a length not greater than the radius of the pad. See Handbook for illustrations. (In actual practice, the component leads may need to be cut much shorter than is shown, allowing the component to be mounted closer to the board.)
4. Solder the leads into place using the same techniques as those in steps (3) and (4) for DIP's (Procedure Sheet No. 7). When the iron is removed, remember to move it *upward* and off, to cover the end of the lead with molten solder.
5. Clean each joint with solvent and brush or spray unit. Using the reference photographs, inspect the joint to insure that it meets the following *Criteria of Acceptance:*
   A. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, porosities or pinholes.
   B. The surface is free of any lumps, remaining pockets of flux, or granular appearance.

(continued)

Procedure Sheet *(continued)*

C. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.
D. Solder fillets are concave, and the end of the wire is tinned.
E. There is complete flow-thru of solder in all plated-thru holes.
F. The lead should be visible beneath the solder.

9 Procedure Sheet

SOLDERING PLANAR-MOUNTED COMPONENTS (FLAT PACKS) TO PRINTED CIRCUIT BOARDS

MATERIALS NEEDED

Printed circuit board, single sided
IC's—flatpacks ¼″ x ¼″
Flux, liquid
Soldering iron, 30-watt, pencil-type
Soldering iron tip, plated, chiseled, 1/16″ diameter (1.6mm)
Solder, rosin-cored, .010″ diameter (.25mm)
Pliers, long nose
Diagonal cutters
Solvent
Cleaning brush
Abrasive stick
Scissors
Tweezers

PROCEDURE:

1. Prepare the PC board by cleaning the pad areas with the abrasive stick and washing the board with solvent and disposable tissue.
2. Pre-tin the pads using the soldering iron with 1/16″ tip and .010 (.254 mm) diameter solder. This is most easily accomplished by laying the solder down along the total length of the pad, and then melting the solder. After tinning, again clean the pads with solvent to remove used flux.
3. Pre-form the leads as shown in the illustrations, so that the part will rest flat on the board and a 60° angle exists at the heel of the lead where it meets the pad.
4. Cut the leads to the proper length for the pads with scissors, then clean them with solvent. (Note: Under some circumstances, the specifications may require that gold-plated leads be pre-tinned before soldering. This pre-tinning minimizes the gold-solder contamination and establishes the solderability of the surfaces.)
5. Put new flux on the top and bottom of the leads. This is necessary because no additional solder is to be used on the pre-tinned pads.
6. Position the flatpack so that each lead is centered over the circuit; it can be temporarily held in place during soldering with a piece of tape. Insure that each lead is precisely positioned on its pad and does not overhang along the edges. The minimum length that the lead should extend onto the pad is a distance equal to twice the width of the lead.

(continued)

7. Reflow solder the two leads in opposite corners first for proper positioning. When solder melt is observed, wipe the iron tip along the lead, from the component to the end of the lead. Your workpiece indicator is the flow of solder from the pad up onto the surface of the lead. When complete flow is observed, remove the iron, using a wiping motion toward the end of the lead.

8. Reflow solder the remaining leads, skipping around so that no two leads side by side are soldered consecutively. Heat build-up can be minimized by using a piece of wet tissue put over the component body to act as a heat sink, or by placing the tips of the tweezers on the component lead.

9. Carefully clean the joints, using solvent and brush or spray unit, and then inspect them. Compare your results with the Handbook photographs. They should meet the following *Criteria of Acceptance:*

   A. At the minimum, the leads extend onto the pads for a distance equal to twice the width of the lead. They meet the pad at a 60° angle.

   B. On the pad, the leads are straight, parallel to the pad, and centered over the circuit. They may lie along the edge of the pad, but they must not overhang the edge.

   C. The surface of the solder is smooth, bright and shiny; there is no evidence of pits, pinholes or porosities. (With gold-plated leads that were not pre-tinned, there may be a slight frosty appearance.)

   D. The surface of the solder is free from any lumps, remaining pockets of flux, or granular appearance.

   E. Good wetting action is evidenced by the feathering out of the solder to all surfaces of the joint.

   F. Concave solder fillets exist along the side of the lead, at the heel and at the end.

   G. As shown in Fig. 38c, the fillet existing at the heel of the lead should extend upward for a minimum of one-third the total distance between the pad and the horizontal part of the lead. Solder should not extend above the upper level of the component lead. (See Handbook illustration.)

   H. The shape of the lead is visible beneath the solder.

PANEL METERS

Common Specifications for QP-5570, QP-5580 & QP-5585	
Polarity	Automatic (-) for negative value
Decimal point	User selectable
Input	Analogue
Full scale range	200mV
Configuration	True differential input and reference
Impedance	100MΩ
Conversion rate	2.5 per second
Zeroing	Automatic
Operating temp.	Standard 0 to 50°C

3.5 Digit LCD Panel Meter

- Attractive modern bezel
- Snap retaining into panel
- Shallow depth
- 0.1" launcher connections
- See above for additional specifications

NEED TO MEASURE CURRENT? SEE OUR CURRENT SHUNTS ON PAGE 29

Specifications	
Type of display	LCD
Digits	3 1/2 digits (1999) 12.7mm figure height
Power	5V powered
Over range	Display flashes "1"
Accuracy	±(0.1%rdg + 1dgt.)
Connection	13 pins
Size	62(W) x 25(H) x 15(D)mm
Panel cutout	50.5(W) x 24(H)mm

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5570	$27.95	$24.95	$22.35

3.5 Digit LED Panel Meter

- Surface mount construction for reliability
- 0.1" launcher connections
- Brightness matched superbright LEDs
- Attractive bezel with mounting clips
- 5V powered, TTL compatible common ground
- See above for additional specifications

Specifications	
Type of display	LED, red super bright
Digits	3 1/2 digits (1999) 14.2mm figure height
Power	5V powered, common ground
Over range	Display blinks
Accuracy	±(0.1%rdg + 2dgt.)
Connection	10 pins
Size	68.5(W) x 33(H) x 16(D)mm
Panel cutout	64(W) x 29(H)mm

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5580	$29.95	$26.90	$23.70

Add On Board For QP-5570/80

This add on board makes it easy to set the range and decimal point on our QP-5570 or 80 panel meters. The ranges can be permanently set with a solder bridge or wired to a switch for selectable ranges.
- No 200mV range on QP-5580
- Cannot be used in applications where voltage to be measured also powers DPM.

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5575	$9.95	$8.75	$7.75

3.5 Digit Jumbo LED Panel Meter

This is not just a digital display, but a fully functional high impedance digital panel meter that only requires power to operate. It will measure voltages from 200mV to 200VDC (jumper selectable) and has a maximum count of 1999. The module is suitable for a host of applications and can be used as a voltmeter or adapted for use as a current meter, thermometer etc. The decimal point location can be selected with a jumper and the module uses an easy "plug-in" fixing method. The meter shares a common ground with the voltage being measured.

Specifications	
Type of display	LED
Digits	3 Digits (1999) 25.4mm figure height
Power	5VDC, common ground, 80mA typical
Over range	[illegible]
Accuracy	[illegible]
Size	[illegible]
Panel cutout size	[illegible]

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5585	[illegible]	[illegible]	[illegible]

LCD MODULES

LCD Panel 2 Line 16 Character - Wide Viewing Angle

QP-5517 Without BACKLIGHT

QP-5518 With BACKLIGHT

Wide viewing angle of standard 16 character, 2 line LCD.

Actual stock will vary from photograph

Specifications	QP-5517, QP-5518
Type	2 line LCD
Characters	16 x 2 lines
Character size	4.89 x 2.78mm
Viewing angle - vertical	Sym 0 condition: CR>=2°, Min -10° Max 40°
Viewing angle - horizontal	Sym 0 condition: CR>=2°, Min -30° Max 30°
Supply voltage for logic	SYN: VDD - VSS Min 4.5V Typ 5V Max 5.5V
Other mechanical data	STN, yellow green, 1/16 duty, 6 o'clock
Module dimensions	85(W) x 30(H) x 13.2(D)mm
Viewing display area	65(W) x 16(H)

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5517	$27.95	$24.95	$22.35
Backlit	QP-5518	$34.95	$31.45	$27.90

LCD Panel 2 Line 16 Character Compact with Backlight

Compact sized 16 character, 2 line LCD. Handy for use in smaller sized project boxes where standard sized LCD's are a little too large. LED backlit means there is no need for special inverters as in models which are EL backlit.

Specifications	
Type	2 line LCD
Characters	16 x 2 lines
Character size	3.15 x 1.85mm
Viewing angle - vertical	Sym 0 condition: CR>=2°, Min -10° Max 40°
Viewing angle - horizontal	Sym 0 condition: CR>=2°, Min -30° Max 30°
Supply voltage for logic	SYN: VDD - VSS Min 4.5V Typ 5V Max 5.5V
Other mechanical data	STN, yellow green, 1/16 duty, 6 o'clock
Module dimensions	53(W) x 20(H) x 8(D)mm
Viewing display area	36(W) x 10(H)

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5519	$29.95	$26.90	$23.70

Dot Matrix Liquid Crystal Display Modules Alphanumeric - Large Display

Large character size LCD. Can be viewed from further distances and also handy when the panel needs to be mounted behind Perspex for protection.

Actual stock will vary from photograph

Specifications	
Type	Alphanumeric, 2 line LCD
Characters	16 x 2 lines
Character size	9.7 x 4.9mm
Viewing angle - vertical	Sym 0 condition: CR>=2°, Min -10° Max 40°
Viewing angle - horizontal	Sym 0 condition: CR>=2°, Min -30° Max 30°
Supply voltage for logic	SYN: VDD - VSS Min 4.5V Typ 5V Max 5.5V
Other mechanical data	STN, yellow green, 1/16 duty, 6 o'clock
Module dimensions	122(W) x 44(H) x 13.2(D)mm
Viewing display area	99(W) x 24(H)

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5520	$44.95	$40.45	$35.90

Dot Matrix Liquid Crystal Display Modules Alphanumeric

Compact Liquid Crystal Diode module with ASCII (96 characters), Special Letters (92 characters) and Custom Letters (8 characters). A two line screen displays up to 16 characters at a time and the module will hold the current display in its own inbuilt memory.

Specifications	QP-5515 & QP-5516
Type	Alphanumeric, 2 line LCD
Characters	16 x 2 lines
Character size	4.89 x 2.78mm
Viewing angle - vertical	Sym 0 condition: CR>=2°, Min -10° Max 40°
Viewing angle - horizontal	Sym 0 condition: CR>=2°, Min -30° Max 30°
Supply voltage for logic	SYN: VDD - VSS Min 4.5V Typ 5V Max 5.5V
Other mechanical data	STN, yellow green, 1/16 duty, 6 o'clock
Module dimensions	85(W) x 30(H) x 13.2(D)mm
Viewing display area	65(W) x 16(H)

↗	Cat.	Qty 1+	Qty 5+	Qty 10+
	QP-5515	$17.95	$15.95	$14.35
Backlit	QP-5516	$19.95	$17.70	[illegible]

SOLDERING EQUIPMENT

Soldering equipment is another area where we have developed an excellent range of quality products at an affordable price. We cater for a broad spectrum of user needs and have products suitable for almost any applications from the entry level hobbyists to apprentices and serious professionals and industrial users etc. Our soldering iron range includes gas and electric irons, temperature controlled soldering stations, SMD rework stations as well as iron stands, solder and a variety of soldering accessories. We are also monitoring the impact of the new European RoHS guidelines and are adapting our range as the demand for RoHS compliance products starts to grow. We already have lead-free solder an RoHS compliant soldering station in our range and you can be sure that as the demand for these items grows, Jaycar will have the products to support you.

*RoHS means Reduction of Hazardous Substances.

40W Temperature Controlled Soldering Station

An ideal entry-level soldering station for the hobby user. This station comes with a lightweight iron with anti-slip grip and tip cleaning sponge, with temperature adjustment up to 450°C.

It also has a 4mm banana socket connected to mains earth for soldering static-sensitive components.

PRICE BREAKTHROUGH

Specifications	
Recommended use	Hobby
Power	40W
Temperature range	150 - 450°C
Analogue/digital	Analogue
Operating voltage	240VAC
ESD rated	No
Lead-free rated	No
Dimensions	135(L) x 82(W) x 70(H)mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Station	TS-1620	$59.95	$53.95	$47.90
Spare Tip	TS-1622	$14.95	$13.35	$11.70

50W Temperature Controlled Soldering Station

Features a high quality ceramic heating element for accurate temperature control, which is adjustable between 200 and 480°C. The pre-tinned factory tip is hollow, and fits right over the heating element itself. This provides a much more efficient and accurate heating system than cheaper alternatives, which transfer heat to the tip via a narrow collar. It is supplied with a separate iron stand with sponge tray.

THE SOLDERING PENCIL:
The soldering pencil (iron) weighs just 45g (excluding cable), making it comfortable to use for extended periods of time. The lead is attached to the control unit with a multi-pole plug, which has a threaded collar to ensure it doesn't pull loose. The lead is a 1m silicon rubber cable for safety and comfortable use. It also features excellent cable strain relief at both ends to ensure long cable life.

THE CONTROL UNIT / POWER STATION:
The case is strong, made from quality ABS plastic. The main temperature control is centrally located, and features large printed numbers for easy temperature identification. There is a temperature status LED, which illuminates when it is heating, and flashes when the iron is up to temp. The temperature dial is factory calibrated, and is likely to never need adjustment, but there is a calibration dial just in case.

Specifications	
Recommended use	Hobby / Trade
Power	50W
Temperature range	200 - 480°C
Analogue/digital	Analogue
Operating voltage	24VAC
ESD rated	No
Lead-free rated	No
Dimensions	140(L) x 115(W) x 93(H)mm

50 WATTS

DURATECH

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Station	TS-1560	$109.00	$97.95	$86.95
Spare Pencil	TS-1561	$59.95	$53.95	$47.90

80W ESD Safe Lead Free Soldering Station with LCD Panel

Differing types of lead-free solder have a variety of melting points which can create problems for older soldering stations or those not designed for lead free solder. This excellent soldering station is particularly suited to lead-free soldering and is just as capable with ordinary leaded solder. The iron has a wide temperature range which is microprocessor controlled and maintained to within a few degrees of the selected setting.

The lightweight soldering pencil is fitted with a soft insulated rubber grip for comfortable long term use and has a silicon rubber sheathed power cable that won't be damaged by contact with the hot soldering tip. It is connected to the main unit via a 1.1m low voltage cable which is terminated with a secure 4 pin connector.

The control unit houses the temperature controls and simultaneously displays the set operating temperature and actual tip temperature, as well as system status on its digital LCD panel.

Specifications	
Recommended use	Trade / Industrial
Power	80W
Temperature range	160 - 480°C
Analogue/digital	Digital
Operating voltage	24VAC
ESD rated	Yes
Lead-free rated	Yes
Dimensions	240(L) x 120(W) x 180(H)mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
Solder Station	TS-1390	$179.00	$160.95	$142.95
Spare tips to suit:		Qty 1+	Qty 3+	Qty 6+
0.4mm Conical	TS-1391	$12.95	$11.35	$10.35
1mm Conical	TS-1392	$12.95	$11.35	$10.35
2mm Chisel	TS-1393	$12.95	$11.35	$10.35
3mm Bevel	TS-1394	$12.95	$11.35	$10.35

240V SOLDERING IRONS - STANDARD

20/130W Soldering Iron Starter Kit

All the soldering essentials for the hobbyist. This kit represents excellent value, the best in soldering we've seen, anywhere. The sum of the individual parts amount to more than double the price we are selling this kit for.

new

Kit contains:
- 240V 20/130W Turbo soldering iron
- Spare tip
- Basic stand
- 1mm solder in dispenser tube
- Metal solder sucker with spare tip and O-ring

	Cat.	Qty 1+	Qty 3+	Qty 6+
Starter kit	TS-1651	$24.95	$22.35	$19.70

20/130W Turbo Soldering Iron

1 year warranty

Sometimes you need that extra bit of heat in your soldering. Push the little red button on this iron and you get a lot more power - from 20 watts to 130 watts to be precise. Weller-style removable barrel, plated tip and ceramic element.

- Spare tips available:
  TS-1555 - 0.5mm conical tip
  TS-1556 - 1mm conical tip

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1554	$19.95	$17.70	$15.70
Spare Tips		Qty 1+	Qty 3+	Qty 6+
0.5mm conical tip	TS-1555	$4.95	$4.45	$3.90
1mm conical tip	TS-1556	$4.95	$4.45	$3.90

25 Watt Soldering Iron

1 year warranty

Ideal for the hobbyist and handy person. Has a stainless steel barrel and orange cool grip, impact resistant handle.

Spare tip available - TS1466

DURATECH

OUR MOST POPULAR IRON

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1465	$13.95	$12.35	$10.95
		Qty 1+	Qty 3+	Qty 6+
Spare tip	TS-1466	$3.95	$3.45	$3.15

Super Soldering Starter Kit

After Short Circuits 1, the next two series require you to start soldering components to the printed circuit board. So you'll need a good soldering iron. This Soldering Starter kit is ideal.

This kit contains:
- 1 x KC-5178 Clifford the Cricket Kit
- 1 x TH-1886 Precision Cutters
- 1 x TH-1502 Soldering Stand
- 1 x TS-1465 240V 25W Soldering Iron
- 1 x 15g pack 60/40 0.71mm solder

	Cat.	Qty 1+	Qty 3+	Qty 6+
Starter Kit	TS-1655	$49.95	$44.90	$39.90

50W Temperature Controlled Soldering Iron

1 year warranty

If your budget doesn't extend to a soldering station, you can still have temperature control. This iron is adjustable with a dial on the handle. It has a temperature range of 200 - 450°C, high enough for silver soldering if needed. Plated long-life tip.

- Spare tip available
  TS-1542 - 1mm conical tip

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1540	$29.95	$26.90	$23.70
		Qty 1+	Qty 3+	Qty 6+
Spare tip - 1mm conical	TS-1442	$19.95	$17.70	$15.70

40W Soldering Iron

1 year warranty

Ideal for the hobbyist and handy person. Has a stainless steel barrel and orange cool grip impact-resistant handle. Fully electrically safety approved. Spare tip to suit use Cat. TS-1476

DURATECH

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1475	$18.95	$16.95	$14.95
		Qty 1+	Qty 3+	Qty 6+
Spare tip	TS-1476	$4.95	$4.45	$3.90

60W Soldering Iron-12VDC

This is a quality 12V soldering iron. It plugs into your car cigarette lighter and with 60 watts it's hot enough to solder terminals, heavy cable etc. 2.5 metre lead length. Supplied with solder as well. Before use, check your car's wiring is enough to carry the current, or else connect direct to battery.

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1530	$17.95	$15.95	$14.35
		Qty 1+	Qty 3+	Qty 6+
Spare Tip	TS-1531	$6.95	$6.25	$5.50

60W SMD Soldering Iron

1 year warranty

Using a conventional soldering iron for surface mount components can be slow and clumsy. These surface mount soldering tweezers will allow easy soldering and desoldering of all surface mount components from small resistors to large 'flat pack' IC's. With tips available from 2mm to 20mm, it is truly a versatile tool. The iron itself is mains powered, and rated at 60 watts. It heats up in just minutes and features rapid heat recovery. It has a 1.8m mains lead and is supplied with 2 x 2mm tips and a metal stand with sponge. Fully electrically safety approved.

Cat. TS-1701 Cat. TS-1702 Cat. TS-1703 Cat. TS-1704

Supplied with handy stand

	Cat.	Qty 1+
SMD Soldering Iron	TS-1700	$79.00
Spare Tips		Qty 1+
Spare Tip 2.0mm (Pkt 2)	TS-1701	$6.40
Spare Tip 5.0mm (Pkt 2)	TS-1702	$8.80
Spare Tip 10mm (Pkt 2)	TS-1703	$8.80
Spare Tip 20mm (Pkt 2)	TS-1704	$8.80

80 Watt

Ideal for the hobbyist and handy person. Has a stainless steel barrel and orange cool grip impact resistant handle. Fully electrically safety approved. Spare tip to suit use Cat. TS-1486

1 year warranty

DURATECH

	Cat.	Qty 1+	Qty 3+	Qty 6+
Soldering Iron	TS-1485	$22.95	$20.35	$18.35
		Qty 1+	Qty 3+	Qty 6+
Spare tip	TS-1486	$8.95	$7.95	$6.95

PORTASOL GAS SOLDERING

A leading brand recognised worldwide. Portasol products are designed and manufactured in Ireland, to the highest quality standards. Since commencing operation in 1983, they have forged their way into the market with some of the most rugged and reliable gas powered soldering and heating tools in the world. They utilise catalytic combustion (flame tips excluded), which converts the gas flame into infrared radiation. It is a safe, flameless way of heating the tips evenly and continuously. They all use standard butane gas, and have a variety of tips available to support a wide range of applications.

portasol

Portasol Super Pro Gas Soldering Tool Kit

1 year warranty

Features 90 minute run time, 10 second fill, maximum 1300°C temperature and 40 second heat up.
This kit contains a Portasol Super Pro Gas Soldering Iron and the following.
- Quality storage case.
- 2.4mm double flat tip (TS-1322).
- Hot air blow (TS-1324).
- Hot air deflector.
- Cleaning sponge and tray.
- 4.8mm double flat tip (TS-1323).
- Hot knife tip (TS-1325).

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1328	$159.00	$142.95	$126.95

Portasol Pro Piezo Gas Soldering Tool Kit

1 year warranty

This kit contains a Portasol Pro Piezo Gas Soldering Iron and the following.
- Quality storage case.
- 2.4mm double flat tip (TS-1312).
- Hot knife tip (TS-1315).
- Flame tip (TS-1316).
- Cleaning sponge and tray.
- Hot air blow (TS-1314).
- Hot air deflector.

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1318	$129.00	$115.95	$102.95

Portasol Super Pro Gas Soldering Iron

1 year warranty

The Portasol Super Pro is the big brother of all irons in the range. It features adjustable tip temperature up to 580°C, with equivalent electrical power of between 25 and 125W! 125W may seem somewhat excessive to some. You don't need a V8 to drive around town, but it's nice to know that the power is there when it's needed isn't it? Ignition is achieved with the internal Piezo crystal mechanism, and run time is around 2 hours on a 30 second refill. The stainless steel finished gas tank features a viewing window to keep an eye on fuel level, & the protective end cap also acts as a safety cut off for the gas valve control when replaced. With the various tips available & huge power at hand, no job is a match for the Portasol Super Pro.
- 2.4mm tip supplied.

Specifications:
Operating Time: 120 min (approx).
Refill Time: 30 sec (approx).
Ignition: Internal Piezo crystal ignitor.
Dimensions: 234(L) x 25(Dia)mm.
Weight: 135g without gas.

TIPS TO SUIT

1.0mm Chisel	TS-1321	$24.95
2.4mm Chisel	TS-1322	$24.95
4.8mm Chisel	TS-1323	$24.95
Hot Air Blow	TS-1324	$24.95
Hot Knife	TS-1325	$24.95

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1320	$115.00	$103.95	$91.95

Portasol Pro Piezo Gas Soldering Iron

1 year warranty

The Portasol Pro Piezo is a powerful gas soldering iron featuring an adjustable temperature control allowing soldering tip temperatures up to 580°C. With equivalent electrical power of between 15 and 75 Watts, there is little that's not possible with this beauty. It incorporates a viewing window at the end of the gas tank, to keep an eye on the fuel level. The fuel tank fills in around 20 seconds, and will provide approximately 45 minutes of run time. Ignition is via the built in Piezo crystal mechanism, and the end cap provides safety gas shut off when replaced. The Portasol Pro Piezo is a dream to use, and will not let you down.
- 2.4mm double flat tip supplied.

Specifications:
Operating Time: 45 min (approx).
Refill Time: 20 sec (approx).
Ignition: Internal Piezo crystal ignitor.
Dimensions: 178(L) x 22(Dia)mm.
Weight: 75g without gas.

TIPS TO SUIT

1.0mm Chisel	TS-1311	$23.95
2.4mm Chisel	TS-1312	$23.95
4.8mm Chisel	TS-1313	$23.95
Hot Air Blow	TS-1314	$23.95
Hot Knife	TS-1315	$23.95
Flame	TS-1316	$23.95

related products

SEE NEXT PAGE FOR BUTANE GAS

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1310	$99.00	$88.95	$78.95

Portasol Technic Gas Soldering Iron

1 year warranty

The Portasol Technic combines compact power, and convenient reliability, making it one of the most versatile gas soldering irons available. It offers a run time of around 60 minutes, and tip temperature is adjustable up to 450°C. With equivalent electrical power of between 10 and 60 Watts, it is ready to tackle just about anything. The protective end cap features a built in flint type igniter, and if the gas is left on, it will turn it off when replaced. If a compact, versatile and reliable gas iron is what you're after, then the Portasol Technic is for you.

Specifications:
Operating Time: 60 min (approx).
Refill Time: 10 sec (approx).
Ignition: Flint ignitor in end cap.
Dimensions: 170(L) x 19(Dia)mm.
Weight: 60g without gas.

TIPS TO SUIT

1.0mm Conical	TS-1306	$23.95
2.4mm Chisel	TS-1307	$23.95
4.8mm Chisel	TS-1308	$23.95

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1305	$59.95	$53.95	$47.90

Portasol 50 Gas Soldering Iron

1 year warranty

The Portasol 50 is designed specifically with the dedicated DIY enthusiasts in mind. Powered with standard butane gas, it has a run time of around 30 minutes. Ignition is via the flint ignitor in the cap and tip temperature is fixed, reaching an impressive 350°C. With equivalent electrical power of around 35W. It features automatic shut-off when the end cap is replaced and fast refill time. It is a quality tool, sure to give you many years of trouble free operation.
- 1mm tip supplied.

Specifications:
Operating Time: 30 min (approx).
Refill Time: 10 sec (approx).
Ignition: Flint ignitor in end cap.
Dimensions: 196(L) x 26(W) x 19(D)mm.
Weight: 60g without gas.

TIPS TO SUIT

1.0mm	TS-1301	$14.95
2.4mm Chisel	TS-1302	$14.95
4.8mm Chisel	TS-1303	$14.95

	Cat.	Qty 1+	Qty 3+	Qty 6+
	TS-1300	$34.95	$31.45	$27.90

Duratech Solder

60% TIN / 40% LEAD

Two sizes available, 1.0mm and 0.71mm diameter

- 0.71mm is ideal for work with IC's and all fine soldering
- Both types are resin cored
- Hobbypacks are supplied in easy to use cannisters

1KG REELS

	Cat.	Qty 1+	Qty 4+	Qty 10+
0.71mm	NS-3002	$51.50	$46.45	$40.95
1.00mm	NS-3015	$51.50	$46.45	$40.95

200G REELS

	Cat.	Qty 1+	Qty 4+	Qty 10+
0.71mm	NS-3005	$10.95	$9.75	$8.75
1.00mm	NS-3010	$10.95	$9.75	$8.75

HOBBYPACK CANISTERS

- Contains 15-20 grams weight.

	Cat.	Qty 1+	Qty 4+	Qty 10+
0.71mm	NS-3008	$1.85	$1.65	$1.45
1.00mm	NS-3013	$1.85	$1.65	$1.45

Lead-Free Solder

- Lead free 15-20 gram plastic pack
- Diameter 0.71mm
- Contains 95% Tin, 5% Antimony
- Melts at 236° - 243°C

	Cat.	Qty 1+	Qty 4+	Qty 10+
	NS-3080	$6.95	$6.25	$5.50

Lead-Free Solder

Works just as well as ordinary solder but contains no harmful lead.

- Supplied on a 45gm roll with handy cover
- Available in two sizes. NS-3082: 0.8mm dia. NS-3084: 1.0mm dia.
- Melts at 243°C approx

JAPANESE MADE

	Cat.	Qty 1+	Qty 4+	Qty 10+
0.9mm dia	NS-3082	$29.95	$26.90	$23.70
1.0mm dia	NS-3084	$29.95	$26.90	$23.70

Desoldering Braid Dispenser Gun

An integrated desoldering tool for service and production work. Actuated by a thumbwheel, it dispenses braid to length and cuts it by trigger action. Solderability and safety are enhanced by eliminating contamination from skin contact. Saves time and money by reducing waste.

- ESD safe
- Safer - no more burned fingers
- Replacement braid: NS-3043

	Cat.	Qty 1+	Qty 4+	Qty 10+
Dispenser	NS-3042	$79.95	$71.95	$63.95
Replacement Braid	NS-3043	$19.95	$17.70	$15.70

Economy Desoldering Braid Dispenser

Keep your fingers out of harm's way and reduce static risk when desoldering. This dispenser takes 2.4m of 2.4mm Dri-Wick® desoldering braid and is actuated by a thumb-wheel for one-handed operation.

- ESD safe
- Disposable, economical

	Cat.	Qty 1+	Qty 4+	Qty 10+
Dispenser	NS-3040	$9.95	$8.75	$7.75

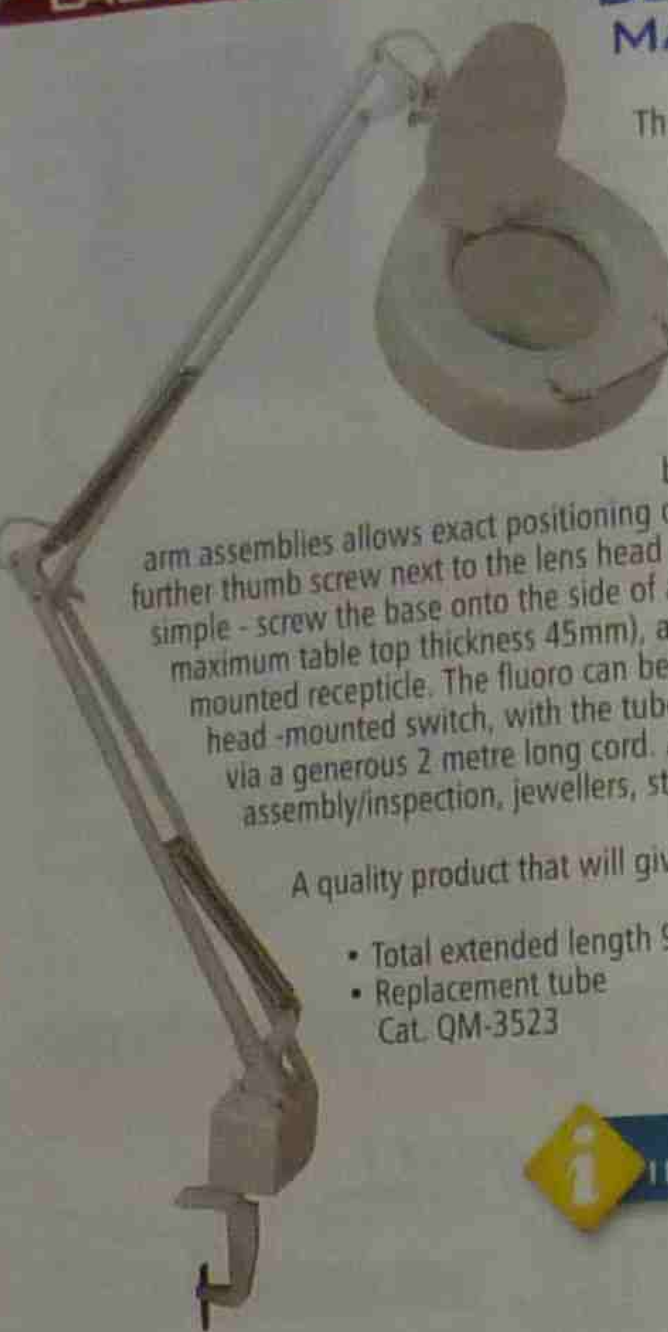
Laboratory Magnifiers & Lamps

Desk Mount Magnifier Lamp

This is a high quality, all metal frame construction magnifier which features a 22 watt circular fluoro inside the magnifier head. The magnifier itself is a 3 dioptre lens, with a metal handle mounted on the magnifier head for quick re-positioning. A thumb screw positioned in between the two flexible extension arm assemblies allows exact positioning of the lens, and is assisted by a further thumb screw next to the lens head itself. Mounting the magnifier is simple - screw the base onto the side of any desk or workbench (approx maximum table top thickness 45mm), and slot the magnifier into the bench mounted recepticle. The fluoro can be turned on or off with a magnifier head -mounted switch, with the tube itself plugged into a mains outlet via a generous 2 metre long cord. A great tool to assist PCB assembly/inspection, jewellers, stamp/coin dealers, etc.

A quality product that will give year after year of reliable service.

- Total extended length 990mm.
- Replacement tube Cat. QM-3523



NEW & IMPROVED

	Cat.	Qty 1+	Qty 2+	Qty 5+
Lamp	QM-3525	$119.00	$106.95	$94.95
Replacement Tube	QM-3523	$16.95	$14.95	$13.35

Bench Laboratory Magnifier Lamp

This magnifier lamp has a 115mm diameter lens that provides 3 dioptre magnification, and a stand that features a quality steel cantilever arm and two swivel joints enabling the magnifier lens to be positioned on almost any angle desired. Features a 22Watt circular fluoro light which is housed under a clear protective screen, a convenient On/off switch positioned on a quality ABS head. Magnifier position is lockable via three lever clamps and the maximum reach of head is over 450mm. The base measuring 200(W) x 280(D) x 40mm(H) and weighing a healthy 3Kgs for increased stability is designed for sitting objects on.

- Lamp is 240VAC operated.
- Replacement fluoro tube Cat. QM-3523

1 year warranty

NEW & IMPROVED

	Cat.	Qty 1+	Qty 5+	Qty 10+
	QM-3522	$149.00	$133.95	$118.95

Desk Top Magnifier Lamp

The perfect tool for coin/stamp collectors, jewellers etc. This desktop magnifier lamp features a 100mm glass lens that will provide you with 3x magnification. The lamp has a solid base with a bright 12W energy-saving fluorescent lamp with electronic ballast. The lamp also features a swivel joint enabling you to position the lens to suit your needs.

- Base 160mm(dia.)
- Stands 225mm high
- Input: 240VAC
- Replacement tube: QM-3521

	Cat.	Qty 1+	Qty 5+	Qty 10+
	QM-3529	$49.95	$44.90	$39.90

Illuminated Gooseneck Magnifier

The gooseneck magnifier is a valuable accessory for coin/stamp collectors, photography etc. The 2X main lens and 5X insert lens with the bonus of 2 LED lights, is a welcome addition for any hobbyist. The magnifier is clip-on or free standing. Suitable for most flat surfaces or workplace incline up to 38mm thick. The flexible metal arm ensures that you can position the magnifier just where it's needed. This handy magnifier also comes with a soft protective pouch for your lens to protect it from dirt and dust. It also has metal stablisers to ensure that your magnifier remains stable while working.

- Lens 110mm (dia.)
- Stands 225mm high
- Includes protective lens pouch
- Requires 3 x AAA batteries

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3532	$36.95	$33.45	$29.45

Magnifying Lens with LEDs

new

Ideal for the elderly or vision-impaired. Keep one in the glovebox for reading the street directory in the dark.

- Big, clear 90mm lens illuminated by two LEDs
- Requires 2 x AAA batteries
- Dual magnification: 3X & 5X
- Case and cleaning cloth included
- Size: 90(Dia) x 160(L)mm

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3537	$14.95	$13.35	$11.70

LED Headband Magnifier

This magnifying headset leaves both hands free and can be worn over prescription or safety glasses. It also has a built-in LED light that can be rotated up, down or side to side to illuminate your work. There are three lenses that can be used in combinations to give 1.5x, 3x, 8.5x or 10x magnification. The head strap is fully adjustable and locks into position and the lens assembly can be swivelled up out of the way when not in use. A multitude of uses including lapidary, jewellery, model-making, electronics and camera repair.

- Requires 2 x AAA batteries

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3511	$39.95	$35.90	$31.90

Magnifying Glass

This huge 4.5" diameter viewer allows hands free operation. It will fold into a neat and easy to store package.

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3505	$13.95	$12.35	$10.95

Eye Magnifier

This 3 times magnifier is handy for checking PC boards, looking at stamps or coins, jewellery, etc, etc.

- Every hobbyist should have one.
- Size: 28(H) x 22(dia view area) x 40(dia base)mm.

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3515	$8.95	$7.95	$6.95

Micro Magnifier with LED

Having trouble reading or working on your hobby in fading or poor light? This convenient pocket-size magnifier with super bright LEDs allows you to work or read for longer periods. Comes complete with a soft pouch for easy storage. Ideal to take travelling or camping.

- 2 Super bright white LEDs
- 3 X magnification with super 5 X magnifier
- One touch ON/OFF operation
- Measures: 90mm(H) x 58mm(W) x 6mm(D)

	Cat.	Qty 1+	Qty 2+	Qty 5+
	QM-3531	$9.95	$8.75	$7.75

Stethoscope

Listen for mechanical noises in equipment

- A very handy diagnostic tool with plenty of non-medical uses

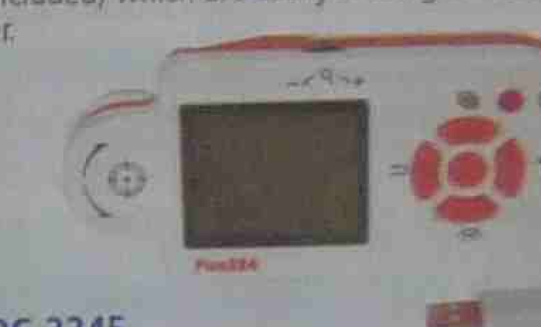
1 year warranty

	Cat.	Qty 1+	Qty 3+	Qty 6+
	QM-7255	$14.95	$13.35	$11.70

Digital Mobile Microscopes

These portable pocket-sized microscopes are surprisingly powerful with a magnification range of 24x - 90x. They use 3 bright-white LEDs to light up objects and feature an adjustable focus to sharpen images. You can also use image filters in real time. Invert the colour, emboss the image or use greyscale. Great for official inspections, scientific education, dermatology, automotive refinishing, forensics, printing, printed circuit board and integrated circuit analysis, jewellers, kit assembly and much more. It's powered by 3 Ni-MH rechargeable AAA batteries (included) which are easily recharged through your computers USB port or the mains power.

Two versions available:



DIGITAL MOBILE MICROSCOPE QC-3245

- Adjustable light intensity and LCD brightness
- 1.8" 65,000 colour TFT LCD
- Camera Resolution: 640 x 480 pixels
- Lanyard included
- Includes 3 Ni-MH rechargeable batteries
- Dimensions: 120(L) x 55(W)mm

DIGITAL MOBILE MICROSCOPE WITH IMAGE CAPTURE QC-3246

Viewed images can be easily transferred to a PC (QC-3246 only) for use in reports, articles, projects etc.

- Adjustable light intensity and LCD brightness
- Split view feature for comparing a saved picture and your current magnification
- Includes software to connect and download images to your PC
- 1.8" 65,000 colour TFT LCD
- Camera Resolution: 640 x 480 pixels
- Capture Resolution: 320 x 240 pixels
- Lanyard included
- Includes 3 Ni-MH rechargeable batteries
- Dimensions: 120(L) x 55(W)mm

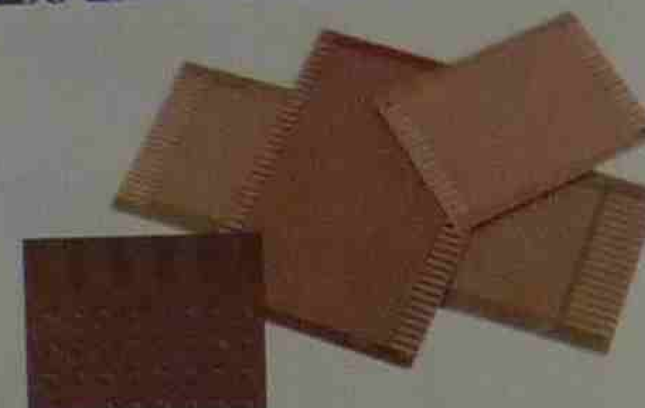
	Cat.	Qty 1+	Qty 2+	Qty 4+
Digital Mobile Microscope	QC-3245	$199.00	$178.95	$168.95
As above with Image Capture	QC-3246	$249.00	$225.00	$215.00

PC Boards - Vero Type Strip

Alphanumeric grid, pre-drilled 0.9mm, 2.5mm spacing.
- 95mm wide x 3 handy lengths

	Cat.	Qty 1+	Qty 4+	Qty 10+
76mm	HP-9540	$4.50	$3.95	$3.60
152mm	HP-9542	$8.50	$7.50	$6.75
303mm	HP-9544	$11.50	$10.35	$8.95

Universal Pre-Punched Experimenter's Boards



CLOSE UP

The low cost way to build prototype or one-off projects. We now stock 3 sizes of phenolic board that has a matrix of punched holes on 0.1" (2.5mm) pitch with 'donut' of tin plated copper around each hole. Finally the non-solder side of the board has an alphanumeric grid printed on it to assist in the component identification board layout.

SMALL
95 x 72mm, 25 x 30 holes (750) 3.6mm edge pitch.

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9550	$4.50	$3.95	$3.60

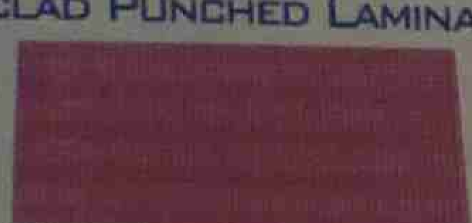
MEDIUM
140 x 95mm, 29 x 50 holes (1450) 2.5mm edge pitch.

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9552	$6.95	$6.25	$5.50

LARGE
210 x 78mm, 24 x 67 holes (1608) 3.6mm edge pitch.

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9554	$8.95	$7.95	$6.95

Unclad Punched Laminate



A single 150(L) x 70(W)mm strip of phenolic punched at 0.1" (2.5mm) grid with NO laminate (copper) on either side.

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9562	$5.50	$4.90	$4.30

Ultra Mini Experimenter's Boards

20 holes deep by 16 holes wide with "links" and "strips" (bus rails) every 2 holes. Board measures a compact 65 x 45mm and is extremely versatile. Supplied as a pair, end-to-end. Can be snapped apart to make two boards (dimensions above) or one 130 x 45mm 0.3" wide x 0.1" pitch.
- Ideal for DIL ICs in small numbers.

Close up

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9556	$4.95	$4.45	$3.90

IC Experimenter's Board

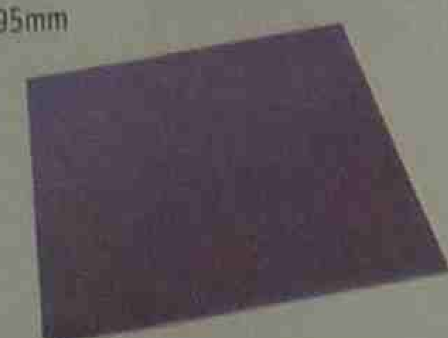
You can fit dozens of DIL ICs on this board. 30 holes deep x 50 holes wide, with a "link"/"strip" (bus rails) pattern enabling alternate V+ and earth rails. Very versatile.
- Alphanumeric markings on grid pattern.
- Size 140 x 95mm.

Close up

Cat.	Qty 1+	Qty 4+	Qty 10+
HP-9558	$6.95	$6.25	$5.50

Aluminium Sheet

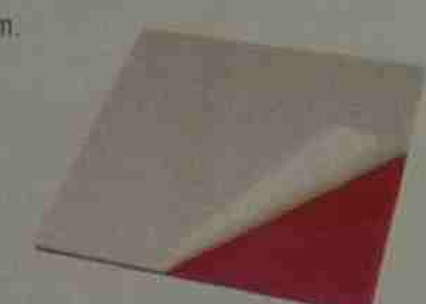
18 gauge 295 x 295mm



Cat.	Qty 1+	Qty 4+	Qty 10+
HM-9500	$13.95	$12.35	$10.95

Perspex - Red or Grey

- 195 x 195mm.
- 3mm thick.



	Cat.	Qty 1+	Qty 4+	Qty 10+
Red	HM-9505	$13.95	$12.35	$10.95
Grey	HM-9507	$13.95	$12.35	$10.95

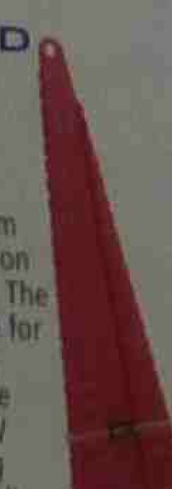
Thermal Transfer Tape 100 x 100 x 0.5mm - Pack 2

When it comes to attaching devices like star LED modules to heatsinks, it's often not easy as they don't have screw "eyes" like a TO220 transistor package. The material is self adhesive and provides good thermal conductivity making it easy for chipsets, LED modules, memory modules etc.
- 2pcs of 100mm x 100mm sheets supplied. 0.5mm thickness.
- Thermal Conductivity: 1.6 W/m-k.
- Flammability Level: UL94V-0.
- Working Temperature Range: -60 to +250°C.
- Insulation Strength: >7,000V/mm

Cat.	Qty 1+	Qty 4+	Qty 10+
NM-2790	$12.95	$11.35	$10.35

Component Lead Forming Tool

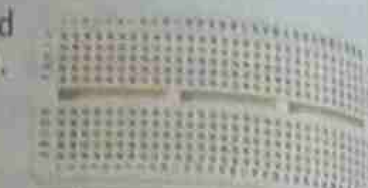
Get the hole spacing for your resistors and diodes perfect every time. This handy forming tool provides uniform hole spacing from 10 to 38mm. Suitable for production assembly, education and training. The tool is double sided with one side for use with DO47 outline diodes (eg 1N914) and 1W zener diodes, the other side being suitable for 1/5W resistors, DO41 outline diodes (eg 1N4004). An incredibly handy tool!
- Size: 138(L)mm
- Made in USA from engineering plastic.



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1810	$7.95	$6.95	$6.25

Mini Breadboard

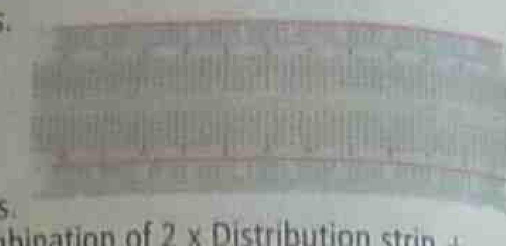
One terminal strip supplied which gives 30 holes x 10. Total 300 holes.
- Size 39(W) x 87(L)mm.



Cat.	Qty 1+	Qty 4+	Qty 10+
PB-8832	$9.95	$8.75	$7.75

Breadboard

- 840 tie points.
- 200 distribution holes.
- 640 terminal holes. (Simply a combination of 2 x Distribution strip + 1 Terminal strip).
- Size 65(W) x 175(L)mm.



Cat.	Qty 1+	Qty 4+	Qty 10+
PB-8814	$19.95	$17.70	$15.70

Breadboard

- 1680 tie points.
- 400 distribution holes.
- 1280 terminal holes.
- Mounted on a metal plate.
- 3 banana terminals.
- Rubber feet.
- Size 157(W) x 237(H)mm.
- Board size: 130(W) x 175(H)mm.

1 year warranty

Cat.	Qty 1+	Qty 4+	Qty 10+
PB-8816	$44.95	$40.45	$35.90

Breadboard Jumper Kit

This kit consists of 70 pcs of single core sturdy wire which has been stripped on each end and bent at right angles.
- Specifically made for breadboards.
- There are 5 pcs each of 14 different lengths.

Cat.	Qty 1+	Qty 4+	Qty 10+
PB-8850	$11.95	$10.70	$9.50

Economy Breadboard Jumper Kit

Solid core hookup cable, which is the ideal size for breadboards. Cut it to whatever lengths you require and strip the ends.

2 METRES EACH OF 5 COLOURS
Colours may vary from time to time.

	Cat.	Qty 1+	Qty 5+	Qty 10+
Per Pack	WH-3032	$3.45	$3.05	$2.70

Pin Extractor Press

A handy little pin-extractor/inserter press with a 08mm punch. Mainly intended for taking links out of watch bands, but endless other uses for jewellery making, modelmaking and hobbies.
- 2 spare pin punches
- Assortment of 12 pins

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2014	$13.95	$12.35	$10.95

YELLOW PRICEBAR DENOTES CLEARANCE ITEM

Riston UV/PCB Developing System

This is basically blank printed circuit board, with a photo-resist. A photo-resist is a coated chemical that changes composition when exposed to light. The PCB Riston is placed on a flat surface, and your negative artwork is then sandwiched between a sheet of glass and the Riston PCB. You can expose the PCB pattern and board in broad daylight, however a controlled source of UV is recommended. After the recommended exposure time, you remove the PCB pattern. Where the Riston was in shadow the photo-resist will not have changed chemical composition. Where it has been exposed it will. You then apply the Developer (HG-9964) to complete the process. The copper tracks will still have the Riston present, and this must be removed with Stripper CRS-40 (HG-9968).

Riston PCB

Blue negative acting. FR4 fibreglass material.
- Single sided.

152 X 152MM

Cat.	Qty 1+	Qty 4+	Qty 10+
HG-9970	$24.95	$22.35	$19.70

152 X 305MM

Cat.	Qty 1+	Qty 4+	Qty 10+
HG-9972	$59.95	$53.95	$47.90

Riston Stripper & Developer

Stripper required to clean board before soldering.
- 600ml bottle.

CRS-40/5 CRD30

	Cat.	Qty 1+	Qty 4+	Qty 12+
Stripper	HG-9968	$9.95	$8.75	$7.75
Developer	HG-9964	$7.95	$6.95	$6.25

Press 'n' Peel Film

This is by far the easiest way to make Printed Circuit Boards from CAD software or magazine PCB layout artwork. You simply make the PCB image by putting a Press 'n' Peel sheet of transfer film in the paper tray of a photocopier or Laser Printer (The sheet size is almost the same as the A4 paper tray). You then print or copy the PCB track image onto the film. Using a domestic clothes iron you press the image onto a precleaned piece of copper clad PCB. The image transfers to the copper and the image will not etch away. Ammonium persulphate or ferric chloride are both OK. You get 5 sheets of 215 x 280mm transfer film in each pack.
Full instructions are supplied.

4 Easy Steps:
1. Print or Copy 2. Iron on 3. Peel off 4. Etch

Cat.	Qty 1+	Qty 4+	Qty 12+
HG-9980	$35.00	$31.45	$27.95

Etch Resistant Pen (Dalo Pen)

This looks like a normal felt tip pen but it isn't. You simply draw on a clean piece of blank copper clad PCB. The ink from the pen is etch resistant so you have a quick and simple way of making that PCB! The ink will wash off afterwards with metho, turps, etc.

Cat.	Qty 1+	Qty 4+	Qty 10+
TM-3000	$9.95	$8.75	$7.75

Blank Fibreglass PCB

Pure copper bonded to quality fibreglass base.
- HP-9514, 9512, 9510 are single sided.
- HP-9515 is double sided.

mm	Cat.	Qty 1+	Qty 4+	Qty 10+
150x75	HP-9514	$5.95	$5.25	$4.75
150x150	HP-9512	$8.95	$7.95	$6.95
150x150	HP-9515	$9.50	$8.50	$7.50
300x300	HP-9510	$19.95	$17.70	$15.70

PCB Etching Kit

An ideal kit for anyone needing to etch a circuit board - complete with an assortment of copper boards, etchant, working bath and tweezers. It also includes a positive acting photosensitive PCB and developer.
Supplied with:
1 x Plastic bath to accept PCBs up to 180 x 110mm
1 x Photosensitive single sided PCB
1 x Sachet of positive developer for Riston
1 x 100g sachet of sodium persulphate etchant
1 x Plastic tweezers
Assortment of PCB's, Instructions

Cat.	Qty 1+	Qty 4+	Qty 10+
HG-9990	$27.95	$24.95	$22.35

Ammonium Persulphate - 400gm

The alternative to ferric chloride. Mix contents (400 grams) with 1.5 litres of hot water to etch copper boards. The contents will treat about 500sq/cm of PCBs.

Cat.	Qty 1+	Qty 4+	Qty 12+
NC-4254	$12.95	$11.35	$10.35

Etch Your Own PC Boards Booklet

Need just one or two printed circuit boards, for a magazine project or a prototype for your latest design? It's not too hard to etch your own single sided or even double sided boards, especially if you have access to a photocopier or a PC and laser printer. In this tech update we tell you all you need to know.

Cat.	Qty 1+
BI-8212	$2.00

DeoxIT ProGold Contact Cleaner & Rejuvenator - Pen Style

Simple cleaners wash away dirt, grime, and dust from your expensive equipment but are often ineffective at cleaning tough oxidation and metal sulphide contamination. This product will not only clean, but it will help restore your equipment to its original condition, improving its performance.

- Kit contains:
  DeoxIT Power Booster 7.4ml
  DeoxIT Pro Gold Conditioning Treatment 7.4ml
  2pc Brush Applicator
  4pc Swabs
  Lint Free Cloth

	Cat.	Qty 1+	Qty 4+	Qty 10+
Pen	NS-1430	$19.95	$17.70	$15.70
Aerosol	NS-1434	$24.95	$22.35	$19.70
Kit	NS-1436	$24.95	$22.35	$19.70

ProGold Contact Protector Pen Style

Simple cleaners wash away dirt, grime, and dust from your expensive equipment but are often ineffective at cleaning tough oxidation and metal sulphide contamination. This product will not only clean, but it will help restore your equipment to its original condition, improving its performance.

Cat.	Qty 1+	Qty 4+	Qty 10+
NS-1432	$18.95	$16.95	$14.95

related products

FOR SOLDER, LEAD FREE SOLDER, SOLDER PASTE AND FLUX SEE PAGES 37 - 38.

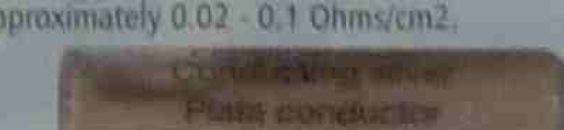
Silver Conductive Pen

This is a highly conductive polymer for use in a variety of applications and is an indispensable tool for fixing damaged circuit board tracks. The pen can also be used to draw new tracks on circuit boards, making jumpers, or even designing prototype circuit boards. It is suitable for use on most surfaces and has excellent adhesion and abrasion resistance properties. Tracks can even be soldered after heat curing.
- 1mm track trace width.
- 0.02 Ohm per mm resistivity.

Cat.	Qty 1+	Qty 4+	Qty 10+
NS-3033	$29.95	$26.90	$23.70

Silver Conductive Varnish

This silver conductive varnish adheres well to glass, PCB material, ceramics etc. Repairs printed circuits, window antennas, window alarm loops etc. Very good conducting properties.
- Approximately 0.02 - 0.1 Ohms/cm2.



Cat.	Qty 1+	Qty 4+	Qty 10+
NS-3030	$9.95	$8.75	$7.75

SOLDERING

Artwork Knife

[illegible] precision knife with safety cap. Ideal for fine [illegible] cuts, etching, hollowing, scoring, scraping, [illegible] stripping and trimming. Four piece jaw chuck [illegible] method of blade retention.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
Knife	HG-9955	$4.95	$4.45	$3.90
Blades Pkt 5	HG-9956	$3.95	$3.45	$3.15

Benchtop Work Mat

Protect your benchtop. This durable A3 size PVC cutting mat is just the thing to protect your work benchtop. You can cut on it, solder, write on it, and not damage your workpiece. The mat is 3mm thick, and measures 450 x 300mm.

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	HG-9100	$12.95	$11.95	$10.95

Polymorph Pellets

Polymorph will change the way you make parts. It's a commercial grade thermoplastic that softens enough to be formed into any shape at around 62 - 65°C. You simply heat the pellets in hot water. It hardens at room temperature to form a tough plastic material similar in consistency and colour to Nylon. It can be drilled, sanded, ground, machined, painted or heated and reformed again and again. Endless uses in model making, craft, single part manufacture, prototyping, engineering, science, lab and more.

- Supplied in a 100g bag of 3mm pellets.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
	NF-4260	$14.95	$13.95	$11.95

Epoxy Repair Putty 28g

Repair wood, brick, concrete plastics, ceramics or composites. You can also fabricate or mould small parts. Simply cut off as much as you need, knead together to mix. It has an open time of about 5 minutes and cures in about an hour. Storage tube included so it doesn't dry out before you use it. Essential for the boat, car or toolbox.

- Fast setting
- Grey coloured
- Can be drilled, tapped or machined
- Handy 28g size

new

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1520	$5.95	$5.25	$4.75

Tape & Adhesives

Coax Seal Tape

This versatile material looks like ordinary PVC electrical tape but is actually a handy sealing system that fuses together to form a removable, waterproof seal once it has been applied. The tape has many uses such as waterproofing outdoor coax connections, sealing low pressure vacuum lines, or even as an alternative to heat shrink tubing where access is difficult.

- 12700mm x 1.52mm

USED BY NASA & US MILITARY

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
	NM-2820	[illegible]	[illegible]	[illegible]

PVC Insulation Tape - 20m

High quality insulation tape suitable for wrapping electrical wires.

- Tape thickness: 0.18mm
- Roll Length: 20m
- Self-Extinguished within 2 seconds after being on fire
- Voltage breakdown: 8.5kV

NM-2852 - Yellow/Green
NM-2853 - White
NM-2854 - Red
NM-2855 - Blue
NM-2856 - Black
NM-2857 - Grey

↗	Qty 1+	Qty 4+	Qty 10+
All Colours	$1.35	$1.20	$1.05

PVC Insulation Tape

NITTO DENKO

Top quality PVC tape.

- Two sizes and six colours available.
- 18mm wide.

↗	Length	Cat.	Qty 1+	Qty 4+	Qty 10+
Black	5m	NM-2800	$0.78	$0.70	$0.60
Blue	20m	NM-2801	$2.25	$1.95	$1.80
Red	20m	NM-2802	$2.25	$1.95	$1.80
Black	20m	NM-2803	$2.25	$1.95	$1.80
Grey	20m	NM-2805	$2.25	$1.95	$1.80
White	20m	NM-2807	$2.25	$1.95	$1.80
Y'w/Gn	20m	NM-2804	$2.25	$1.95	$1.80

Insulation Tape - 5m 6 Pack

Contains one roll each of black, white, red, blue, grey and yellow/green.

- 19mm wide.
- Each 5 metres in length.
- Pack will vary from picture.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
Pk. 6	NM-2806	$2.95	$2.60	$2.30

Self Amalgamating Tape - 10m

This insulating tape will "cure" into a single mass (once applied) to help seal and waterproof cable, wires or even pipes. The tape has good corona resisting properties and can be used on high voltage cables. Useful where water and chemical resistance is important.

- Corona resistant.
- Chemical resistant.
- Self amalgamating.
- Length: 10 metres.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
10m	NM-2826	$23.95	$21.35	$18.95

Adhesive Backed, Closed Cell Polyethylene Tape - 2m

This UV stabilised closed cell polyethylene tape is 16mm wide and 3mm thick. It is great as a seal for mounting speakers or for making water tight joins when clamping 2 surfaces together.

- The 2 metre roll is sufficient for mounting 2 x 12" woofers.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
2m	NM-2840	$4.95	$4.40	$3.90

Double Sided Mounting Tape

Thousands of uses, from mounting PCBs in your projects to putting up your Britney Spears posters. Both are double-sided and the NM-2821 is also foam-backed.

- 12mm wide
- Two roll sizes

↗		Cat.	Qty 1+	Qty 4+	Qty 10+
10 metres		NM-2821	$4.95	$4.45	$3.90
25 metres		NM-2823	$3.95	$3.45	$3.15

Aluminium Foil Tape

To be used in any number of situations including as shielding.

- 50m roll
- Foil thickness: 0.04mm
- Raw material: Aluminium foil, Acrylic-based adhesive
- Width 25mm or 50mm
- Application Temp.: -20 120°C

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
50mm	NM-2860	$17.95	$15.95	$14.35
25mm	NM-2862	$11.95	$10.70	$9.50

Nashua Gaffer Tape - 40m

PREFERRED BY THE PROFESSIONALS

Genuine Nashua brand Gaffer tape. Professional quality 40 metre rolls in both black and silver. Leaves no residue behind and sticks to most clean surfaces, including carpet. 50(W)mm x 40(L)m.

Ideal for

- Outdoor concerts.
- Theatre.
- Television studios.
- Display exhibitions.
- School productions.
- Car racing.

↗	Cat.	Qty 1+
Black	NM-2812	$19.95
Silver	NM-2814	$19.95

Gaffer Tape - 25m

This is not your cheap plastic stuff. It is the genuine cloth-backed tape that you can split with your thumbnail. This waterproof tape is black in colour, low sheen and is the industry standard 48mm wide. Commonly called "100 mile an hour" tape, or "gaffer" tape, it was originally developed for the air conditioning industry.

- It is very adhesive and very strong.
- Ideal for taping down leads on stage.
- Repairing flexible duct work.
- Other public address setups.
- Holding things together in general. A must in every toolbox!

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
25m	NM-2810	$13.50	$11.95	$10.70

Heavy Duty PVC Tape -

Cordon off hazardous areas or create an unmistakable marker with this heavy duty PVC tape. The vivid colouring will draw anyone's attention to the barrier.

- Width: 50mm
- Thickness: 0.18mm
- 33m roll
- Raw material: PVC film, rubber-based adhesive

Two colours available:

- Red/White Cat NM-2864
- Yellow/Black Cat NM-2866

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
	NM-2864	$15.95	$14.35	$12.70
	NM-2866	$15.95	$14.35	$12.70

Tape & Adhesives (continued)

Easy-Tear Magnetic Mounting Tape

Ideal for craft projects, calendars, kids' artwork or to-do lists. Simply apply tape to a supporting surface and another piece to the item, then stick it anywhere you like. The adhesive can be removed without residue from any smooth surface - glass, wood, plastic etc. No cutting is needed as the tape simply tears to length, so it's completely safe, even for littlies.

new

- One side magnetic, one side adhesive
- Residue-free
- Easy tear-off, no scissors
- 3m x 12mm roll

↗	Cat.	Qty 1+	Qty 6+	Qty 20+
	LM-1608	$9.95	$8.75	$7.75

3 Second Super Glue

Ideal for plastics, wood, rubber, glass, metal and ceramics. It sticks in seconds and needs no clamping. Supplied in a small tube of 3 grams.

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1500	$1.90	$1.70	$1.50

Super Glue 15ml

Super glue in a handy 15ml bottle for hobby or industrial use. Bonds plastics, metals, wood, rubber, glass, metal and ceramics.

new

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1506	$3.95	$3.45	$3.15

25ml Metal Epoxy

new

Two-part metal epoxy. Bonds ferrous and non-ferrous metals including steel, stainless steel, aluminium, copper, brass and iron. Convenient 25ml syringe pack for accurate mixing. Fast setting and cures to a grey colour.

- Sets in 5 minutes
- Cures to usable strength in 3 hours
- 25ml syringe dispenser

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1516	$5.95	$5.25	$4.75

Epoxy Bond

Two part epoxy resin-based adhesive recommended for use where a quick set high-strength adhesive is required. Bonds in approximately 5 minutes to give a colourless, permanent bond. Maximum strength after 1 hour. Ideal for rigid plastics, metal, glass, concrete, china & pottery. Supplied in 24ml no-mess dual syringe.

NEW & IMPROVED

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1510	$5.95	$5.25	$4.75

Corrosion Buster Pen

This pen-type product enables you to remove rust, spot corrosion and clean burnt contacts etc. without abrasion utilising the 20,000 tiny glass fibres at the tip of the pen. It has an amply sized body with finger indent to provide a comfortable & secure grip while the adjustable length allows for more aggressive cleaning when required.

- 120mm long x 14mm dia.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
	NA-1410	$14.95	$13.35	$11.70

Fix-Lock Anaerobic Adhesive

Secure nuts & screws with this excellent thread locking compound. Excellent for locking screws & retaining fitted parts for assembly. Can even be used for sealing threaded pipe joints. Compound sets when parts are bought together and is resistant to chemicals, oils & gases. Will not shrink.

- Supplied in 7g pack.
- Non shrinking.
- Chemical resistant.
- Will allow disassembly.

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1502	$7.95	$6.95	$6.25

Liquid Electrical Tape/ Thread Locker

118ml Can

This liquid can be used instead of insulation tape. Simply brush it on and it will insulate and seal out moisture. Once dry, it is acid & salt resistant and won't crack, peel or harden even under extreme conditions. This is a very versatile product and can be used for such diverse applications as stopping screws from vibrating loose or ropes unravelling. Available in two colours.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
Black	NM-2832	$24.95	$22.35	$19.70
Red	NM-2834	$24.95	$22.35	$19.70

28g Tubes

- Handy 28g tube.
- Available in two colours.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
Black	NM-2836	$15.95	$14.35	$12.70
Red	NM-2838	$15.95	$14.35	$12.70

Wire Glue

A soldering iron in a jar. Wire Glue is a electrically conductive adhesive that enables you to make solder-free connections when you don't have a soldering iron handy, or aren't able to apply heat for some reason. Hundreds of hobby, trade and electronics uses. Lead-free, cures overnight.

- 9ml

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NM-2831	$9.95	$8.75	$7.75

Isopropyl Alcohol

Isopropyl alcohol is one of the most useful service aids you can have around. It has many of uses such as head cleaning, surface cleaning and prep, contact cleaning, stain removal in the laundry etc. It's also a medical-grade surface disinfectant and is ideal for cleaning your windscreen before you put your new rego label on. Dries quickly and is relatively non-toxic due to 50% concentration. A choice of two sizes:
NA-1062 20ml spray
NA-1064 200ml bottle

new

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
20ml	NA-1062	$2.50	$2.21	$1.95
200ml	NA-1064	$6.95	$6.20	$5.50

Isopropyl Alcohol Cleaning Pads

Removes contaminants from metal, plastic & painted surfaces.

- Also ideal for cleaning computers.
- Packet of 10.

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
Pkt 10	NA-1060	$4.95	$4.45	$3.90

INOX-MX3 Lubricant, Corrosion Inhibitor

This product is similar to popular water displacement products EXCEPT that it does not contain silicone or kerosene based solvents etc. Indeed, it is so benign it can be used around food. It is also ideal for seized nuts or screws, freeing up hinges, door locks, as a cutting or tapping & other machining agent. As the product is non-conductive, it is ideal for removing moisture from car distributors, switches, potentiometers, solenoids etc. It does not dry or wash out in water. You will be delighted with this product.
Two sizes are available:
Cat. NA-1022 125g Pump Pack (can be sent by post)
Cat. NA-1024 300g Aerosol Spray (can't be sent by post)

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
Spray Pump	NA-1022	$7.25	$6.25	$5.75
Spray Can	NA-1024	$11.50	$10.35	$8.95

Heatsink Compound - 5 Gram Tube

Non electrically conductive adhesive heatsink compound 5g tube - one 5g tube will adhere up to 16 TO-3 package transistors.

↗	Cat.	Qty 1+	Qty 4+	Qty 10+
	NM-2014	$7.35	$6.95	$6.25

Inox Premium Grade Machinery Grease

General-purpose synthetic grease in a handy 30gm tube size. Ideal for bearings, ball joints, chains, sprockets, O-rings, bushes & other rubber and plastic etc.

- Food grade
- High temperature, non-melt
- Highly resistant to water, salt, chemicals and drying
- Fully synthetic
- 30 gram tube

new

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
	NA-1020	$7.95	$6.35	$6.25

SOLDERING

WANT IT FAST? HOW ABOUT NEXT DAY DELIVERY - SEE PAGE 6

SOLDERING

AEROSOLS

PTFE Dry Lubricant Spray

This water displacing PTFE lubricant is formulated to provide a dry, lubricating film that is perfect for use with electronic & mechanical assemblies, chains & sprockets, rollers, runners, & power tools etc. It prevents rusting on ferrous metals and helps loosen stiff or sticking parts. The spray can be used in automotive, marine and household environments.

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1013	$17.95	$15.95	$14.35

Freezing Spray

Non CFC ozone safe propellant. Instant freeze spray for rapidly cooling components to detect intermittent thermal faults, dry joints and overheating problems.

- Non-flammable.
- 250g size.

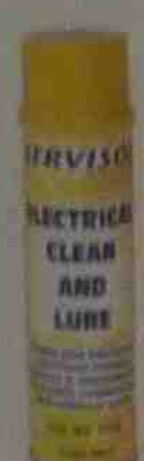
MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
250g	NA-1000	$17.95	$15.95	$14.35

Contact Cleaner Lubricant

A blend of non CFC solvents and RF contact treatment oil. Ideal for cleaning and lubricating non-arcing contacts and mechanisms. Can be used for cleaning & lubricating all types of contacts.

- Safe on most plastics.

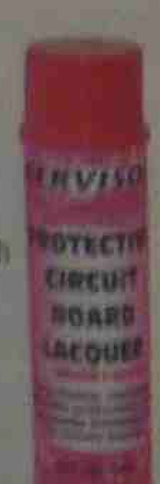


MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1012	$11.50	$10.35	$8.95

Circuit Board Lacquer

Non CFC ozone safe propellant. Premium quality flexible coating to protect printed circuit boards from humidity and environmental attack. Special solder through lacquer so previously coated boards can be soldered.



MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1002	$11.50	$10.35	$8.95

Electronic Circuit Board Cleaner

Non CFC ozone safe propellant. It may not be obvious, but many circuit problems occur when flux and other contaminents (even finger prints) remain on the PCB causing 'leakage' between tracks. If you work with RF, high impedance (i.e. FET Op-Amps etc), ioniser or other circuits, PCB cleanliness is a MUST. The electronic circuit board cleaner is a spray that dissolves flux residues & grime leaving the track work and board absolutely clean. We then recommend that you 'tropicalise' the board with circuit board lacquer (NA-1002).

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1008	$11.50	$10.35	$8.95

Water Displacement & Lube Spray

Multi-use water displacing and rust preventing lubricant specially formulated for use with electronic and mechanical assemblies. Displaces moisture from metal surfaces including: relays, switches, distributors etc. Quickly penetrates rusted and corroded metal parts like nuts and bolts, valves, tools. Also removes grease, chewing gum, adhesives, crayons from most surfaces.

ITS NOT WD40 BUT WE THINK IT'S JUST AS GOOD

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1025	$5.95	$5.25	$4.75

Spray-On Contact Adhesive

This pressure-sensitive contact adhesive is in an aerosol can, and bonds to paper, fabric, leather, rubber, foam, vinyl, concrete, metal, in fact almost any surface! No more spills and dribbles from a tube, or sticky screwdrivers (and plastic applicators) from applying glue to a surface.

Great for laying innerbond and speaker carpet in/on speaker cabinets etc.

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
400g	NA-1504	$16.95	$14.95	$13.35

Butane Gas

Use it for filling up your gas soldering iron, flame torches, cigarette lighters, etc. Includes five different adaptors, which cover most different gas valves.

- Includes filling instructions.

MADE IN AUSTRALIA

5 adaptors supplied

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
150g	NA-1020	$5.95	$5.25	$4.75

Dust Remover

Non CFC. Non flammable gas which allows removal of dust from electronic,electrical and optical devices. It does not leave residues and is non toxic and non conductive.

- 250 grams

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
250g	NA-1018	$17.95	$15.95	$14.35

Electronic Cleaning Solvent

Non CFC ozone safe propellant. A highly efficient fast drying solvent of high purity. For use on delicate electronic, electrical & precision mechanical assemblies (i.e. tuners, switches, plugs, sockets, relays, pots, etc).

MADE IN AUSTRALIA

↗	Cat.	Qty 1+	Qty 4+	Qty 12+
175g	NA-1004	$11.50	$10.35	$8.95

Ultrasonic Distance Meter with Laser

1 year

Measure distance quickly from a remote position. This ultrasonic measurer also calculates area, sums total readings and stores data for later use in imperial or metric units. Laser pointer for accurate placement of the measurement point.

- Backlit LCD
- Range: 0.6 - 16m (1.6 - 52ft)
- Accuracy: ±1% of reading
- Response time: 2 seconds
- Auto power-off: 100 seconds
- Backlight time-out: 7 seconds
- Auto or manual power-off
- Case and belt-clip included
- Dimensions: 175(L) x 62(W) x 45(D)mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	QP-2295	$44.95	$40.45	$35.90

Laser Level with Tape Measure

1 year

Whether you are a keen handy person around the house or a professional, you'll love this laser level with tape measure. Use this tool to hang pictures, paintings or mirrors in your home, install shelving, lay tiles etc. The range of the laser line projects up to 6m indoor and covers an angle arc of 45 and 90 degrees. The unit incorporates a horizontally spread laser to create illuminated guide lines along walls, horizontal and vertical levels. It also includes a handy 2.5m tape measure as well as a ruler on the side with inches on one side and millimetres on the other.

- Size: 158(L) x 57(H) x 27(D)mm
- Indoor Laser Range: 20 ft/6m
- Laser Diodes: red 640 to 660nm wavelength
- Levelling: 2 Bubble Vials
- Power Supply: 2 x "AAA" batteries (not included, use our SB-2426)

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	ST-3113	$14.95	$13.35	$11.70

Circuit Tester 90 - 300V

1 year

This small circuit tester allows you to test mains voltage for live circuits between 90V to 300V. It also has a pocket clip to prevent it from falling out while getting into those awkward places.

- Internal neon light to indicate a live circuit
- Pocket Clip
- Insulated Casing
- AC/DC indicator

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	QP-2240	$3.95	$3.45	$3.15

related products — SEE PAGE 25 FOR MORE TESTERS

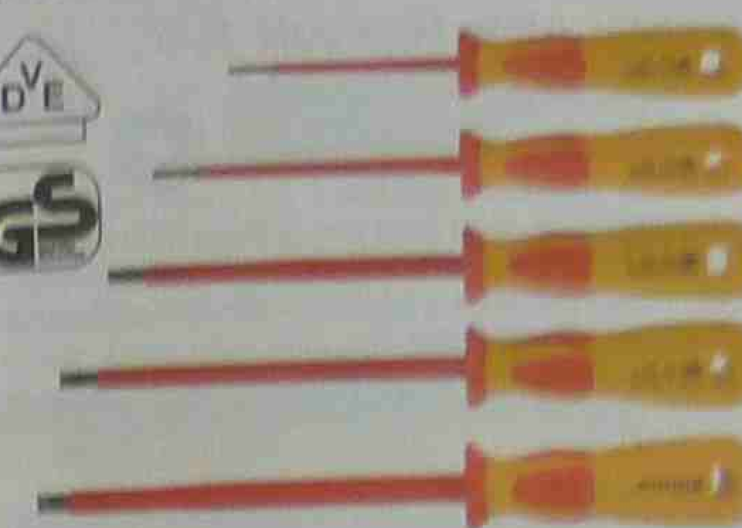
ELECTRICAL SCREWDRIVERS

This range of trade quality insulated screwdrivers is one of the most comfortable screwdrivers we have ever used. The ergonomic handles have a soft rubber coating for a secure, comfortable grip that you can use for hours on end. All are TUV and GS approved and rated up to 1kV.

- A size for any application

Slotted

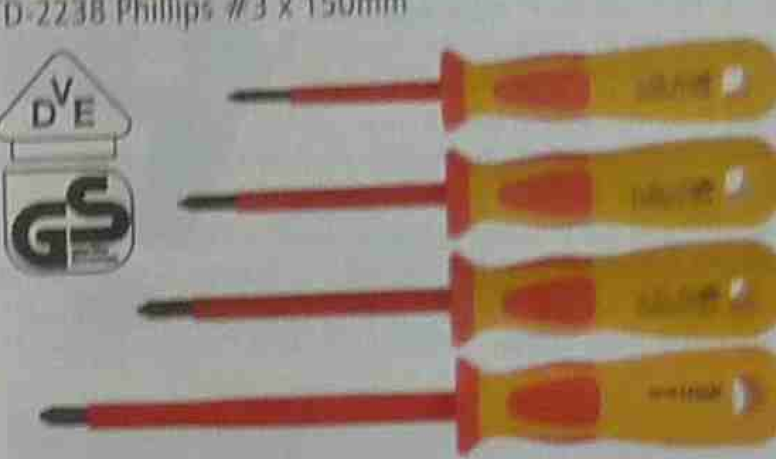
TD-2230 Flat Blade 2.5 x 75mm
TD-2231 Flat Blade 3.0 x 100mm
TD-2232 Flat Blade 5.5 x 125mm
TD-2233 Flat Blade 6.5 x 150mm
TD-2234 Flat Blade 8.0 x 175mm



↗ Wmm	Lmm	Cat.	Qty 1+	Qty 3+	Qty 6+
2.4	75	TD-2230	$4.50	$3.95	$3.60
4.0	100	TD-2231	$4.95	$4.45	$3.90
5.5	125	TD-2232	$5.65	$4.95	$4.45
6.5	150	TD-2233	$6.95	$6.25	$5.50
8.0	175	TD-2234	$7.95	$6.95	$6.25

Phillips

TD-2235 Phillips #0 x 60mm
TD-2236 Phillips #1 x 80mm
TD-2237 Phillips #2 x 100mm
TD-2238 Phillips #3 x 150mm



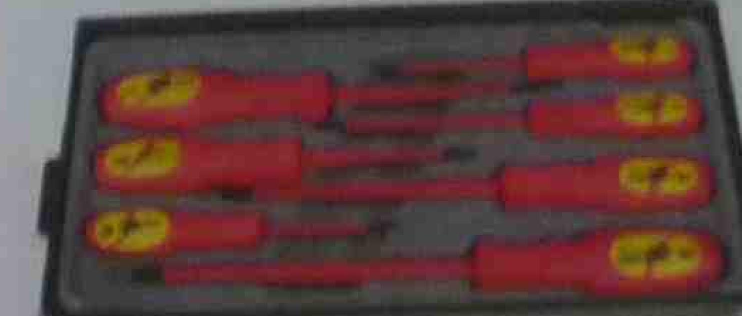
↗ Size	Blade	Cat.	Qty 1+	Qty 3+	Qty 6+
0	60mm	TD-2235	$4.95	$4.45	$3.90
1	80mm	TD-2236	$5.65	$4.95	$4.45
2	100mm	TD-2237	$6.95	$6.25	$5.50
3	150mm	TD-2238	$7.45	$6.50	$5.75

1,000V 7 Piece Screwdriver Set

A quality set of 7 screwdrivers with storage box that offers reliability & value without compromise. GS & VDE tested & approved to 1000V. With red handles and insulation right to the tip, you will find them amongst your tools quickly & easily.

- DIN EN 60900.

Drive sizes:
Flat: 1.2 x 6.5 x 150mm, 1 x 5.5 x 125mm, 0.8 x 4 x 100mm and 0.6 x 3.5 x 75mm
Phillips: #2 x 100mm, #1 x 80mm and #0 x 60mm



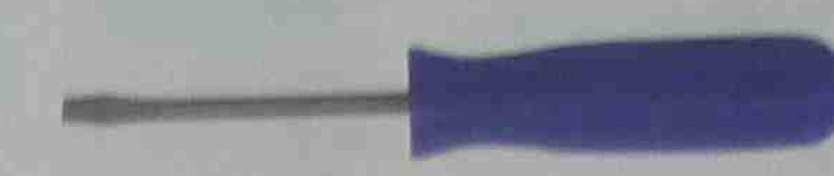
↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2022	$24.95	$22.35	$19.70

MARINE GRADE STAINLESS STEEL

MADE FROM #316 GRADE MARINE (3CR13) STAINLESS STEEL. EXCELLENT FOR HOSTILE ENVIRONMENTS LIKE SALT WATER, CHEMICALS ETC.

Screwdriver

- Slotted or Phillips



↗ Size	(L)mm	Cat.	Qty 1+	Qty 3+	Qty 6+
Slotted 3mm	75	TD-2340	$4.95	$4.45	$3.90
Slotted 4mm	100	TD-2342	$5.65	$4.95	$4.45
Slotted 5mm	125	TD-2344	$5.95	$5.25	$4.75
Phillips 1	80	TD-2350	$4.95	$4.45	$3.90
Phillips 2	100	TD-2352	$5.95	$5.25	$4.75

"Smart" Test Screwdriver

- Capacitor check
- Diode check
- Transistor check
- Globe/relay/fuse/speaker/resistor check
- Locating broken wire
- Picks up static radiation of TV or monitor
- Instantaneously checks AC power
- Earth disconnection check

It allows you to check / test the following safely:

- AC voltages: Contact method from 70 to 250V AC
  Non contact method from 70 to 250V AC
- DC voltage test up to 250V DC
- Continuity check - L = 0 to 5MΩ,
  - H = 0 to 2,000 MΩ
- Polarity check 1.5V to 36V DC
- Microwave leak detector greater or equal to 5MW/CM2
- Wrong mains connection check
- Check operating condition of negative Ion generators.
- It is also a screwdriver!!!
- Supplied with comprehensive instructions.
- Brightly coloured.
- Batteries included.

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2055	$11.95	$10.70	$9.50

Low Voltage Circuit Tester 6, 12, 24 Volts

Looks like a neon test screwdriver but instead of a blade on the end this tester has a probe and a 28" lead which clips to ground. Suitable for 6/12/24 volts for use on cars, trucks, boats, etc.

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2049	$3.95	$3.45	$3.15

Tool Magnetiser / Demagnetiser

Magnetised screwdrivers make life easier. This tool has two slots, one which will magnetise and the other to demagnetise. The holes are large enough for the largest of screwdrivers.

- 50 x 50 x 30mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2042	$6.95	$6.25	$5.50

TOOLS

LED Screwdrivers with 10 Bits

new

LED illuminated bit driver for working in spaces with poor lighting. The handle has four LEDs built in to provide working light. 10 bits are included, but any standard hex bit will fit. Great for fiddling around under the bonnet etc

- Four LEDS to eliminate blind spots
- Bits included: PH #0, #1, #2, slotted 3, 4, 5mm, T15, M6 pin drive, M4 hex, hex - 1/4" square converter
- Batteries included, plus a spare set

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2091	$22.95	$20.35	$18.35

4 in 1 Screwdriver Set

This handy screwdriver has four different bits and is about the size of a normal pen. The lid can also be used to extend the length of the handle. Encased in a durable plastic case.

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TH-1909	$3.95	$3.45	$3.15

5 Piece Screw Extractor Set

Also known by the trade name of 'Easyouts', these are the only real way to get a completely stripped or mangled screw, bolt or stud out, even if the head has broken off completely. You simply drill a hole in what remains of the offending fastener, then use the appropriate size screw extractor with a tap wrench. They have an aggressive left-handed thread that bites into the metal, enabling you to unscrew the damaged fastener. This is the kind of tool you need once in a blue moon, but when you need them, they're indispensable.

Sizes: 1/8", 5/32", 3/16", 1/4", 5/16"
Case size: 120(W) x 120(H) x 15(D)mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2079	$11.95	$10.70	$9.50

6-in-1 Compact Screwdriver

This innovative compact screwdriver has six of the most needed blades that are stored neatly in the handle until you need them. You rotate the collar on the handle to eject the blade you want. The blades can't completely come out and thus cannot get lost. They slide down to fit within the length of the handle. Compact and robust with hardened blade tips. Shafts made of a very strong, corrosion-resistant alloy of iron, vanadium and molybdenum.

- Slotted: 4, 5, 6mm
- Phillips: #1, #2, #3
- Size Compacted: 128(L) x 30(Dia)mm

↗	Cat.	Qty 1+	Qty 3+	Qty 6+
	TD-2028	$18.95	$16.95	$14.95

CARBON STEEL TOOLS

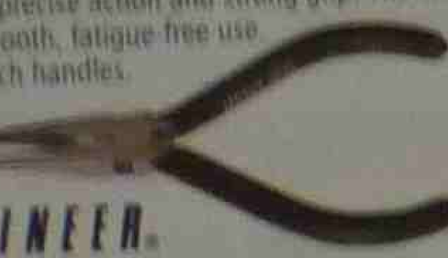
150mm Precision Side Cutters

These cutters are made from the same High Carbon Steel as our TH1885 long nose pliers and are designed for sharp cutting in precision wiring. They have insulated soft-touch handles and a coil return spring for fatigue free use.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1891	$44.95	$40.45	$35.90

125mm Precision Long Nose Pliers

These quality pliers are made in Japan from the same High Carbon Steel that is used to make professional chef's knives. The pliers feature serrated jaws and a box joint to provide a precise action and strong grip. The coil spring ensures smooth, fatigue free use. Insulated soft touch handles.



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1885	$39.95	$35.90	$31.90

When Stainless Steel is Not the Best

We are big fans of Stainless Steel. We love it in tools, fasteners & many building products. It is not, however, always the best material. Top chefs will tell you, for example that the best knives are not stainless. They are made out of carbon steel. Why? Carbon steel is far harder & tougher than stainless. The problem of course is that carbon steel will rust if not maintained.

In our search for the best hand-tools we have decided to introduce two very high quality Japanese-made products made out of carbon - steel. They are not cheap but they are for the professional user whose time is money. Like carbon steel knives, rust is not a problem if used constantly. They will, however, pay you back every day because they will stand up to heavy wear & tear.

For those who have only occasional use for hand tools, we sincerely recommend stainless steel products.

Fujiya 110mm Precision Side Cutters

Beautifully engineered side cutters for the discerning tradesman. These cutters feature high quality tool steel construction and are perfect for cutting super fine wire as well as general workshop use. They aren't just good to look at, they are tough as well and can cut steel wire up to 1.6mm without harm. The insulated handles are spring loaded for effortless use.

- 110mm long
- Blade hardness: Rockwell C Scale 57~61
- Soft grip handle

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2332	$39.95	$35.90	$31.90

5" Stainless Steel Precision Cutters

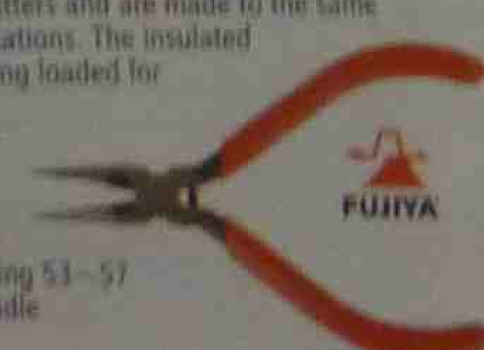
These Precision Cutters are just that - precision! The best cutters and pliers are box joint which these are. A box joint ensures a precise and positive grip and cutting action. They are of course, stainless steel with a polished finish. They have a leaf spring for smooth controlled action, and cut flush. Handles are black with a cushion grip and length is around 125mm (5").

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1886	$17.95	$15.95	$14.35

Fujiya 110mm Precision Long Nose Pliers

These are the perfect companion for our TH-2332 precision side cutters and are made to the same exacting specifications. The insulated handles are spring loaded for effortless use.

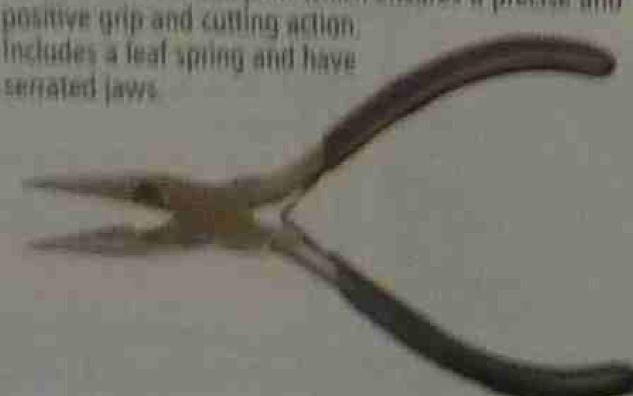
- 110mm long
- Hardness rating 53~57
- Soft grip handle



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2334	$39.95	$35.90	$31.90

5" Stainless Steel Precision Long Nose Pliers

These long nose pliers are made from polished stainless steel and have a box joint which ensures a precise and positive grip and cutting action. Includes a leaf spring and have serrated jaws.

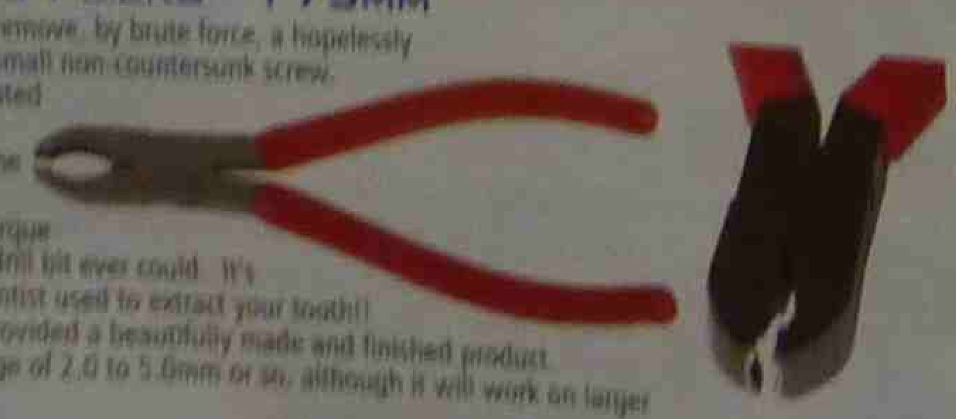


Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1888	$17.95	$15.95	$14.35

Screw Removing Pliers - 175mm

This tool is basically designed to remove, by brute force, a hopelessly stripped out head of any type of small non-countersunk screw. The secret is an oval-shaped serrated hole in the end of the plier. You place the plier over the head of the screw, grip and rotate the plier. This system applies a lot more torque to the head of the screw than a drill bit ever could. It's reminiscent of the plier that a dentist used to extract your tooth!! Its Japanese manufacturer has provided a beautifully made and finished product. Suitable for any screw in the range of 2.0 to 5.0mm or so, although it will work on larger screws to lesser effect.

- 175mm long
- Soft grip handle



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2330	$29.95	$26.90	$23.70

Stainless Steel Side Cutters

High quality small side cutters that have thick (2mm) blades and comfort soft plastic handles which are spring loaded.

- 115mm long.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1890	$13.95	$12.35	$10.95

Stainless Steel Long Nose Pliers

The partner to our TH-1890 stainless steel cutters. These have half round smooth gripping jaws perfect for adjusting and bending components, picking up that dropped nut, etc. Comfortable soft plastic handles which are spring loaded.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1893	$13.95	$12.35	$10.95

Precision 5" Angled Side Cutters

Precision side cutters that are ideal for fine PCB work. They will easily cut leads flush with the board's surface. The cutters are made from quality tool steel and have soft padded handles that are spring loaded for comfortable long term use.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1897	$11.95	$10.70	$9.50

Precision 6" Long Nose Pliers

Precision, slim line long nose pliers that are ideal for working in confined areas. They have serrated jaws so you can get a firm grip on the item you're holding. The pliers are made from quality tool steel and have soft padded handles that are spring loaded for comfortable long term use.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1887	$11.95	$10.70	$9.50

Tools for Your Trade

Jaycar is an approved supplier. This joint Australian Federal Government and Australian Apprenticeship initiative allows qualified participants to an $800 (including GST) allowance to purchase "Tools for Your Trade". Do you qualify? Find out at www.toolsforyourtrade.com.au

6" Side Cutters - Insulated

This drop forged alloy tool is made to the same standards as our TH-1984 combination pliers. It is just as tough and will also cut piano wire up to 1.6mm. This is a quality tool and designed for dedicated wire cutting. It has comfortable double inset handles.

- 160mm/6" long.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1985	$17.95	$15.95	$14.35

6.5" Long Nose Pliers - Insulated

You can see the quality German design in these strong and attractive pliers. They are drop forged to the same standards as the other tools in the range and have the same cutting capacity - up to 4.0mm electrical wire. A very attractive tool with comfortable handles. The high quality manufacture means they can take the work and last for ages. This could be the last pair of long nose general purpose pliers you need to buy!

- 170mm / 6.5" long.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1986	$16.95	$14.95	$13.35

180mm (7") Combination Pliers

Strong, Tough & Reliable.
These high quality drop forged alloy steel pliers are a professional tool and manufactured to the rigid German DIN standards for electrical safety & mechanical strength. They are for serious tradesmen who need a quality tool. These pliers will cut hard (piano wire) wire up to 1.6mm & soft (annealed copper, aluminium silver etc,) up to 4.0mm. Non slip double inset handles are comfortable to hold.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1984	$19.95	$17.70	$15.70

4.5" Bull Nose Pliers

These high quality miniature combination bull nose pliers are made from hardened carbon steel with a micro-nickel finish. The handles have a matt red vinyl coating. TUV and GS approved. 120mm / 4.5" long.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1889	$11.95	$10.70	$9.50

MARINE GRADE STAINLESS STEEL HAND TOOLS

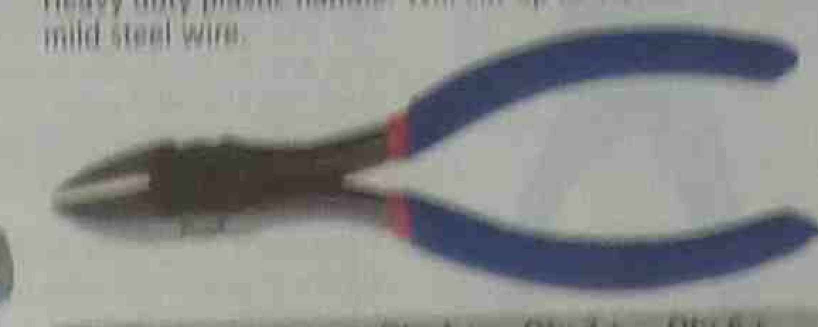
A range of quality hand tools in #316 marine grade (3Cr13) stainless steel.

Even the highest quality carbon steel tool will rust hopelessly in a fishing tackle box. How many times have your pliers seized up with rust? If you need to use tools in a hostile environment like saltwater, chemicals etc. then these tools are for you.

- Marine mechanical service
- Marine electrical service
- Swimming pool maintenance
- Hazardous wet area/ chemical maintenance
- Professional fishing
- Outdoor road maintenance
- Recreational fishing
- Hydraulic hose repair industry

150mm (6") Diagonal Cutters

Very strong jaws will enable this tool to last for years. Heavy duty plastic handle. Will cut up to 1.2mm mild steel wire.



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2304	$14.95	$13.35	$11.70

150mm (6") Long Nose Pliers

Heavy duty plastic handle with wire cutting facility. Bright polished finish.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2300	$12.95	$11.35	$10.35

170mm (7") Bull Nose Pliers

Always handy. Once again heavy duty plastic moulded handle.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2308	$14.95	$13.35	$11.70

Adjustable Wrench (Shifting Spanner)

A must in every toolbox.

- 200mm (8").

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2312	$14.95	$13.35	$11.70

Adel Nibbling Tool

The original and best. The Adel nibbling tool has been around for years and for good reason. It's still the best hand nibbler going around and is ideal for chassis bashing and all sorts of hobby applications. Cut, notch or trim simple or complex shaped holes in plastics, laminates, leather or metal.

- Capacity:
- Mild steel: 1.2mm
- Aluminium: 1.6mm
- Plastics: 2mm
- Spare punch use TH-1767
- Made in USA.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1765	$69.95	$62.95	$55.95
TH-1767	$44.95	$40.45	$35.90

SHEARS

High Quality Stainless Steel 7" Electrical Shears

Slice through soft electrical cable with ease. These precision spring loaded stainless steel electrical shears will cut soft wire up to 22sqmm and make light work of cable ties, heatshrink, light hook up wire and cable braid etc. The shears feature insulated handles and a convenient, thumb operated lock to keep them closed when not in use.

- Ideal for cutting ribbon cable

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1757	$7.95	$6.95	$6.75

High Quality 5.5" Electrical Shears

These shears offer impressive cutting ability with finely honed and serrated blades. Great for cutting insulation, heatshrink, spaghetti and light duty hook-up wire. These shears feature insulated handles.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1758	$6.95	$6.25	$5.50

Stainless Steel 6.5" Cable Shears

This set of cutters has a cutting capability of up to 10mm cable. It is constructed from stainless steel and has a hardened long-life cutting edge. For safety and ease of storage, it can be locked in the closed position.

- Stainless Steel
- Lightweight
- Lockable.

NEW & IMPROVED

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1898	$19.95	$17.70	$15.70

NIBBLING TOOLS

Nibbling Tool

Will cut any shape out of aluminium, plastic, copper and other unhardened metals up to 18 gauge. Tool is designed to fit into the palm of your hand for easy use, simply drill a 1/4" hole to start.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1768	$24.95	$22.35	$19.70

PC Drill Pack

Includes all the common types:
- 1 x 3.5mm for PC supports and relays
- 2 x 1.2mm - for PC pins and hook-up wire
- 3 x 1mm - for resistors, capacitors, etc.
- 4 x 0.8mm - for ICs etc.

TOTAL - 10 DRILLS

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2400	$18.50	$16.35	$14.70

Mini Drills

	Pkt	Cat.	Qty 1+	Qty 3+	Qty 6+
0.8mm	ea	TD-2408	$2.65	$2.30	$2.07
0.8mm	10	TD-2420	$17.95	$15.95	$14.35
0.9mm	10	TD-2422	$17.95	$15.95	$14.35
1.0mm	ea	TD-2410	$2.65	$2.30	$2.07
1.0mm	10	TD-2421	$17.95	$15.95	$14.35

HSS Engineering Grade Drill Bits

You can buy cheap drill bits from hardware stores but they are almost as soft as cheese. If you want a serious, tradesman quality drill bit you will pay over $20 for, say, a 1/2" bit. We have scoured the world for quality drill bits that won't break the bank! They are less than half the price of comparable quality bits. They are made from the finest quality high-speed steel and we dare you to find them inferior to the Aust/NZ made bits that are unfortunately so expensive now. (The big ones are made for the 10mm chucks).

	Cat.	Qty 1+	Qty 3+	Qty 6+
1/16"	TD-2700	$1.75	$1.55	$1.40
3/32"	TD-2703	$1.75	$1.55	$1.40
1/8"	TD-2706	$1.75	$1.55	$1.40
5/32"	TD-2709	$1.75	$1.55	$1.40
3/16"	TD-2712	$1.75	$1.55	$1.40
7/32"	TD-2715	$2.45	$2.15	$1.95
1/4"	TD-2718	$2.75	$2.45	$2.15
5/16"	TD-2721	$4.45	$3.95	$3.45
3/8"	TD-2724	$5.45	$4.90	$4.30
7/16"	TD-2727	$7.95	$6.95	$6.25
1/2"	TD-2730	$11.95	$10.70	$9.50
1.0mm	TD-2750	$1.75	$1.55	$1.40
1.5mm	TD-2752	$1.75	$1.55	$1.40
2.0mm	TD-2754	$1.75	$1.55	$1.40
2.5mm	TD-2756	$1.75	$1.55	$1.40
3.0mm	TD-2758	$1.75	$1.55	$1.40
3.5mm	TD-2760	$1.75	$1.55	$1.40
4.0mm	TD-2762	$2.25	$1.95	$1.80
5.0mm	TD-2764	$2.65	$2.30	$2.07
6.0mm	TD-2766	$2.75	$2.45	$2.15
8.0mm	TD-2768	$4.35	$3.90	$3.45
9.0mm	TD-2770	$5.45	$4.90	$4.30
9.5mm	TD-2772	$6.95	$6.25	$5.50
10.0mm	TD-2774	$6.95	$6.25	$5.50
11.0mm	TD-2776	$7.95	$6.95	$6.25
12.0mm	TD-2778	$9.95	$8.75	$7.75
12.5mm	TD-2780	$11.50	$10.35	$8.95
13.0mm	TD-2782	$11.95	$10.70	$9.50

PCB Drill Chuck

This chuck will allow you to use drill bits from 0.6mm up to 2mm on your normal drill / cordless drill / power screwdriver or screwdriver which accepts standard hex bits.

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2010	$9.95	$8.75	$7.75

Rotary Drill Parts Set

new

This 400 piece kit will service every bit you will ever need. It also has a base so you can turn your tool into a freehand router and comes housed in a fold-out case so you can see exactly where everything is.

Contents include:
- Sanding arbours: 2 x 6mm, 2 x 6mm
- Sanding belts (P80): 20 x 12mm, 24 x 6mm
- Collets: 2 x 3/32", 2 x 1/8"
- Drill bits: 2 x 1/16", 2 x 3/32", 2 x 1/8"
- Grinding stones: 2 each of 20 x 4mm wheel, 10 x 4mm wheel, 15 x 10mm cylinder, 12 x 10mm cylinder, 7 x 4mm cylinder, 20 x 10 bullet, 11 x 8mm tapered bullet
- 1 each of: 15 x 6mm cylinder, 15 x 8mm cylinder, 15 x 8mm bull-nose cylinder, 4 x 10mm cylinder,
- 10 x 8mm bull-nose cylinder 10 x 10mm inverted cone, 15 x 10mm cone, 10mm ball
- Tungsten carbide burrs: 3mm straight, ball, cone profiles
- Diamond coated burrs: 2 each of 2.5mm straight, 4mm ball, 2mm ball, 2mm fine cone taper
- Polishing wheels: 1 x 25mm, 4 x 12mm, 1 x 15mm bull nose cylinder (mounting shaft included)
- Steel, brass, fibre wire brushes: Wheel, cone and fine brush profiles
- Cutoff wheels: 5 x 30mm
- Grinding wheels: 6 each of P80 20 x 4mm, P120 20 x 4mm (2 x mounting shafts included)
- 1 x P1200 fine grinding stone
- 1 x buffing mop and buffing paste
- 1 x paint-removing wheel
- 1 x 3/8" collet spanner
- 1 x router base
- 1 x grinding wheel dressing stone
- Approx 250 assorted sanding discs
- Case measures: 370(W) x 300(H) x 65(D)mm

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2456	$69.95	$62.95	$55.95

3/8" Precision Keyless Drill Chuck

Features an ergonomic design that gives a more powerful locking action and a patented 'Click Lock' system to indicate that the chuck is properly locked. The chuck's unique locking system makes it ideal for use on hammer drills and other high-vibration applications.
- 3/8" - 24UNF mounting thread
- 1/32" - 3/8" (0.8 - 10mm) drill capacity
- Suitable for drills up to 1,200 watts

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2011	$23.95	$21.35	$18.95

Stepped Drill Bits

Drill multiple size holes with the one bit! You can drill through plastics and non-hardened metals (such as aluminium or copper sheeting etc) up to 4mm thick! Made from high speed steel (HSS), they will last for a very long time. Two sizes available:
4 - 12mm drill in 1mm steps - TD-2436
12 - 20mm drill in 1mm steps - TD-2438

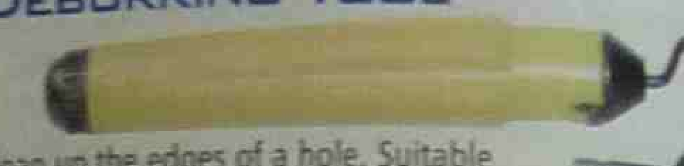
	Cat.	Qty 1+	Qty 3+	Qty 6+
4 - 12mm	TD-2436	$24.50	$21.95	$19.35
12 - 20mm	TD-2438	$34.50	$30.95	$27.45

4 Piece Countersink Set

The easy way to deburr or countersink. Simply drill hole, insert tool with appropriate bit, and turn for a countersunk angle of 90°. Bits include 12, 16 and 19mm - all hardened carbon steel.

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2027	$13.95	$12.35	$10.95

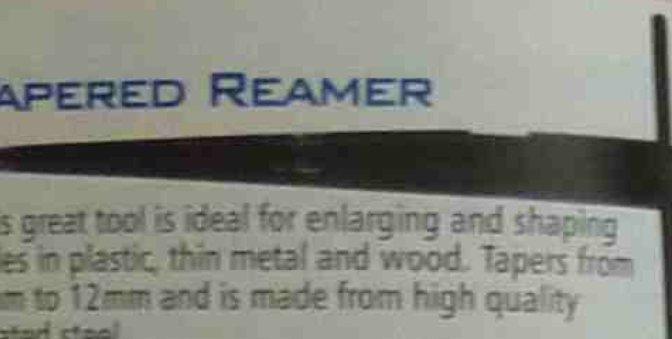
Deburring Tool



Clean up the edges of a hole. Suitable for steel, plastic, wood etc. Blades are fully ground for safety. SK2 steel blades are hardened to HRC64. Supplied with spare blade. Will deburr virtually anything.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1845	$9.95	$8.75	$7.75

Tapered Reamer



This great tool is ideal for enlarging and shaping holes in plastic, thin metal and wood. Tapers from 3mm to 12mm and is made from high quality treated steel.

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2130	$16.50	$14.70	$12.95

File Saw

This is a versatile tool that will find many uses around the home or workshop. Great for cutting odd shapes in plaster, PVC pipe, PVC sheeting, leather, plastic and many other soft materials. The tool will trim holes to any radius and get into places where others won't. The blade measures 175mm long and the large, easy grip handle is 120 mm long.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-2127	$15.95	$14.35	$12.70

M3 & M4 Metric Taps

Clean up a cross threaded nut or cut a thread into a chassis or heatsink etc. High quality tool steel. Two sizes available.

	Cat.	Qty 1+	Qty 3+	Qty 6+
3mm	TD-2440	$3.75	$3.30	$2.95
4mm	TD-2442	$4.45	$3.95	$3.45

T-Type Tap Wrench

Can also be used with twist drills and reamers, etc.
Accepts shafts up to 5mm diameter.

3 - 5MM
Suits TD-2440 & TD-2442 above

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2445	$12.95	$11.35	$10.35

10 Piece Needle File Kit

Every toolbox should have a set of needle files. This set contains every profile you could ever need. All have integrated plastic handles and come in a handy storage wallet. Each is 162mm long.
Profiles:
- Round • Elliptical
- Half-round • Double half round
- Triangle • Square • Flat
- Flat tapered • Knife
- Flat tapered safe back

	Cat.	Qty 1+	Qty 3+	Qty 6+
10 Piece	TD-2128	$15.95	$14.35	$12.70

TOOLS FOR COMPONENTS

Budget SMD Vacuum Pickup Tool

This handy workshop SMD component pickup tool fits into your pocket and uses vacuum pressure to pick up your sensitive components without damaging them. An amazingly useful tool!
- Easy to use, one-hand operation. • Picks up to 20g in weight.
- Supplied with 3 vacuum pads.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1978	$17.95	$15.95	$14.35

Pearl Catch

Incredibly handy tool for those moments when you need to pick up small fiddly bits and pieces.
- Will clip to and hold small nuts, bolts, washers, transistors, resistors, etc.
- Metal construction.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1842	$9.95	$8.75	$7.75

PLCC Extractor

There's really only one way to remove a PLCC safely - and this is it. Basically a spring assisted clip carrier extractor, specifically designed for the safe and hassle free removal of PLCCs from sockets.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1816	$11.95	$10.70	$9.50

Universal IC Pin Straightener

A simple and effective tool to straighten and align bent IC pins. Accommodates standard ICs from 8 to 48 pins.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1814	$11.95	$10.70	$9.50

Pin Extraction Tool

Take the frustration out of removing Molex connector pins with these time saving tools. Simply slide the tool over the connector and push the plunger to effortlessly remove the pin. Both male and female pin extractors available.

	Cat.	Qty 1+	Qty 3+	Qty 6+
Male Pin Extractor	TH-1730	$17.95	$15.95	$14.35
Female Pin Extractor	TH-1732	$17.95	$15.95	$14.35

IC Extractor

Deceptively simple looking device.
- One piece metal construction.
- 8 - 40 pins.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1818	$2.95	$2.60	$2.30

Component Lead Forming Tool

Get the hole spacing for your resistors and diodes perfect every time. This handy forming tool provides uniform hole spacing from 10 to 38mm. Suitable for production assembly, education and training. The tool is double sided with one side for use with DO47 outline diodes (eg 1N914) and 1W zener diodes; the other side being suitable for 1/5W resistors, DO41 outline diodes (eg 1N4004).
- An incredibly handy tool!
- Made in USA from engineering plastic.
- Size: 138 (L)mm

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1810	$7.95	$6.95	$6.25

Benchtop Work Mat

Protect your benchtop. This durable A3 size PVC cutting mat is just the thing to protect your work benchtop. You can cut on it, solder, write on it, and not damage your workplace.
- The mat is 3mm thick, and measures 450 x 300mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
HM-8100	$12.95	$11.35	$10.35

MAGNETS

Large Rare Earth Magnets - Pair

These rare earth magnets are exceptionally strong. They are made from NdFeB (Neodymium Iron Boron), providing the highest available magnetic energy of any material. Suitable for a wide variety of applications where a very strong magnet is required. In fact, these are the same magnets used in our Faraday shake torches. The nickel casing protects the magnets because the material is very brittle.
- Sold as a pair.
- Dimensions: 19mm dia x 28.2mm long
- Grade: NdFeB, N35
- Coating: Nickel

	Cat.	Qty 1+	Qty 6+	Qty 20+
Sold as a pair	LM-1652	$19.95	$17.70	$15.70

Rare Earth Magnets

These magnets feature rare earth compounds combined with ferrite (anisotropic) to produce incredibly strong attraction/ repulsion. They are encased in nickel jackets because the material itself is very brittle. We warn you that getting a finger between these could result in a nasty pinch! They are provided with a nickel plated iron "keeper".

	Cat.	Qty 1+	Qty 6+	Qty 20+
Small - 10mm dia x 3mm - Pkt 4	LM-1622	$11.95	$10.70	$9.50
Medium - 25mm dia x 5mm - Each	LM-1618	$15.95	$14.35	$12.70
Large - 35mm dia x 5mm - Each	LM-1620	$22.95	$20.35	$18.35

Magnets

Educational magnets. Ideal for hobbyists & children to learn more about magnetism.
- Each end is marked with "N" or "S".

	Cat.	Qty 1+	Qty 6+	Qty 20+
Bar magnet - 70L x 12W x 5Dmm	TH-1874	$1.00	$0.90	$0.80
U Shaped - 30mm x 30mm	TH-1873	$1.00	$0.90	$0.80

Ferrite Magnets

Ideal for hobby and craftwork, school projects etc. Can be glued to wood, plastic rubber etc.

	Cat.	Qty 1+	Qty 6+	Qty 20+
10L x 10W x 4Dmm - Pkt 12	LM-1616	$6.95	$6.25	$5.50
20L x 15W x 5Dmm - Pkt 6	LM-1614	$6.95	$6.25	$5.50

Super Strong Horseshoe Magnet

Have you ever done something stupid like drop your keys overboard while out boating? By having this retrieval magnet, you can alleviate much of the pain by attaching a rope to the handle and dropping it overboard where you lost your keys to collect them. Made from super-strong ceramic magnets, it has a lift capacity of 85kgs and is coated to make it resistant to wear and corrosion from salt water.

A must have for boating but has thousands of other applications! And while you're not retrieving your keys from the deep, your kids can even go "treasure hunting" with it to see what's at the bottom of the river.

Note: Keys not included

Cat.	Qty 1+	Qty 6+	Qty 20+
LM-1654	$39.95	$35.90	$31.90

TOOLS

MAGNETS CONTINUED

Flexible Adhesive Magnetic Rubber Strip

This self adhesive magnetic rubber strip can be cut down to suit an application and makes an ideal fridge magnet for adhering to promotional material such as menus, calendars etc.
- Size: 15(W) x 3000L mm.
- 2mm thick.

Cat.	Qty 1+	Qty 6+	Qty 20+
LM-1612	$5.95	$5.25	$4.75

Handy Magnet Strip

You'll find dozens of uses for this innovative magnetic storage system. Its simplicity itself and can be attached to walls, tables or other surfaces to hold tools, brushes, scissors, key rings or any other object that contains iron.

Cat.	Qty 1+	Qty 6+	Qty 20+
LM-1624	$19.95	$17.70	$15.70

Magnetic Wrist Tray

So useful you will wonder why nobody thought of it years ago. This simple yet ingenious wrist tray will hold small screws, washers, nuts, split pins, or other metal items so you won't put them down 'somewhere safe' and lose them. The tray has a Velcro strap and includes 2 x Phillips and 2 x slotted driver bits in their own storage slots. Every handyman should have one.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1971	$14.95	$13.35	$11.70

Telescopic Magnetic Pickup Tool with LED

new

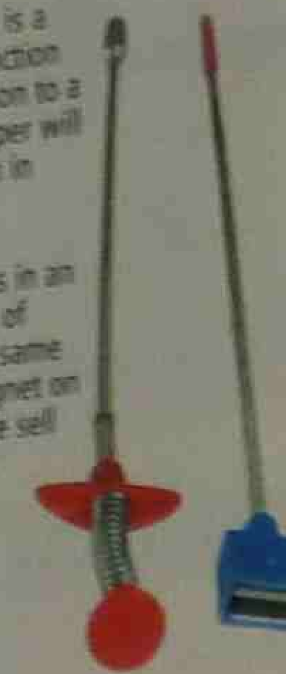
Sunday arvo, 5 o'clock and you drop the last bolt into the crankcase. No problem - with this handy tool, you can see where it is with the built-in LED torch and retrieve it with the magnetic tip. The torch comes on as soon as you extend it and it has a handy pocket clip so it doesn't get lost. A must for every toolbox.

- Picks up over 100g in weight
- Batteries included
- Extends to 870mm
- 170mm long (closed)

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1877	$17.95	$15.95	$14.35

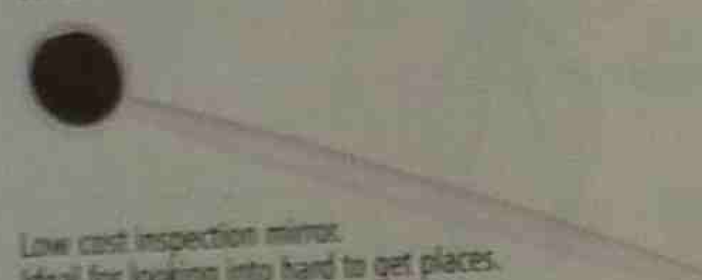
Super Extension Claw & Magnetic Pickup

Actually two tools. Tool No. 1 is a 600mm long flexible syringe-action claw gripper similar in operation to a pearl catch. The 4 prong gripper will capture an object up to 15mm in diameter with about 500g of gripping force. It is ideal for retrieving, say, a nut or screws in an engine compartment or piece of equipment. Tool No. 2 is the same thing except that it has a magnet on the end instead of a claw. We sell them as a set because they compliment each other.



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1868	$8.95	$7.95	$6.95

Inspection Mirror



Low cost inspection mirror.
Ideal for looking into hard to get places.
Length: 160mm, mirror diameter: 24mm.

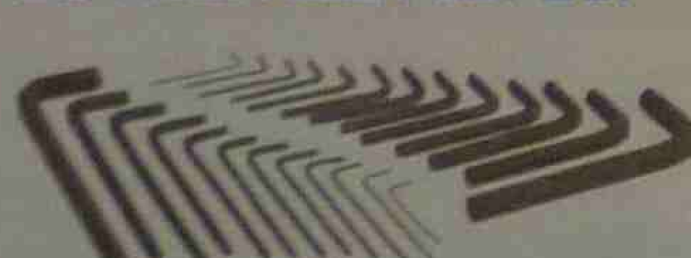
Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1872	$4.95	$4.45	$3.90

Magnetic Telescopic Bar

A VERY HANDY TOOL!
This looks a bit like a telescopic TV antenna section with a clip to hold in your pocket. It has 7 extensions, and measures 640mm when extended and 127mm when closed. The tip on the end is a powerful magnet. It's ideal for picking up that dropped nut, screw or component etc from inside a piece of equipment. The magnet is strong, we picked up a screwdriver which weighed 60 grams.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1870	$8.95	$7.95	$6.95

25 Piece Allen Key Set



25 different size Allen keys in a plastic wallet.
Both metric and imperial sizes.
- Metric: 1.27, 1.5, 2.0, 2.5, 3.0, 3.5, 4.0, 4.5, 5.0, 5.5, 6.0, 8.0, 10.0mm
- Imperial 3/8, 1/4, 7/32, 3/16, 5/32, 9/64, 1/8, 7/64, 3/32, 5/64, 1/16".

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2052	$11.95	$10.70	$9.50

Soldering Heatsink & Component Holder

This quality small tool is ideal as a heatsink when soldering to remove some of the heat from the semiconductor/component while soldering. It can also be used to hold small parts while soldering to avoid burnt fingers.
- Supplied with insulated handles, and length is 65mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2122	$6.95	$6.25	$5.50

Anti-Magnetic Precision Tweezers

These are high quality tweezers made from anti-magnetic, anti acid steel. Perfect for handling SMD components, jewels or general electronic components.
- Length 110mm. • Supplied with protective nylon cap.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1754	$4.45	$3.95	$3.45

Stainless Steel Tweezer Set

1 year

A set of 4 tweezers, three supplied with vinyl handles, comprising:
1 x self closing pointed type
1 x long curved pointed type
1 x long fine pointed type
1 x flat spatula type
Approximate length 115mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1752	$8.95	$7.95	$6.95

Forceps (Haemostats)

These clamps are made of stainless steel and can be locked on to the item they are holding. They are 130mm long, so they are ideal to reach into hard-to-get-at places. Also great for holding components etc., whilst soldering.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1755	$8.95	$7.95	$6.95

4 Piece Mini Pick & Hook Set

These hook tools have stainless steel heat treated points and are ideal for use on O-rings, springs, snap rings, washers, checking soldering joints, general hobbyist kit building, and probing into things, etc.
Hooks are:
- straight pick
- 90° hook
- full hook
- small angle hook.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1762	$15.95	$14.35	$12.70

10 Piece Alignment Tool Set

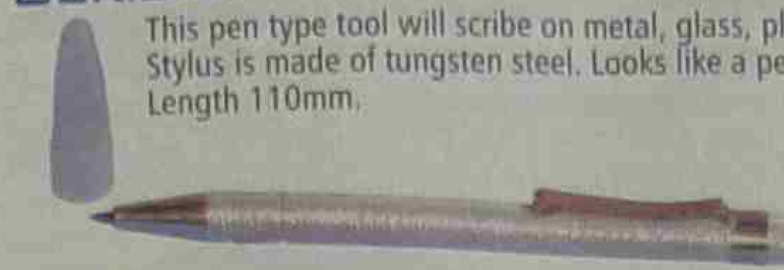
Set of 10 low cost plastic alignment tools. One of each pictured is supplied.
- Length 114mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1806	$9.95	$8.75	$7.75

TOOLS

Scriber

This pen type tool will scribe on metal, glass, plastic, ceramics and tools. Stylus is made of tungsten steel. Looks like a pen, and has a pocket clip. Length 110mm.



Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2120	$7.95	$6.95	$6.25

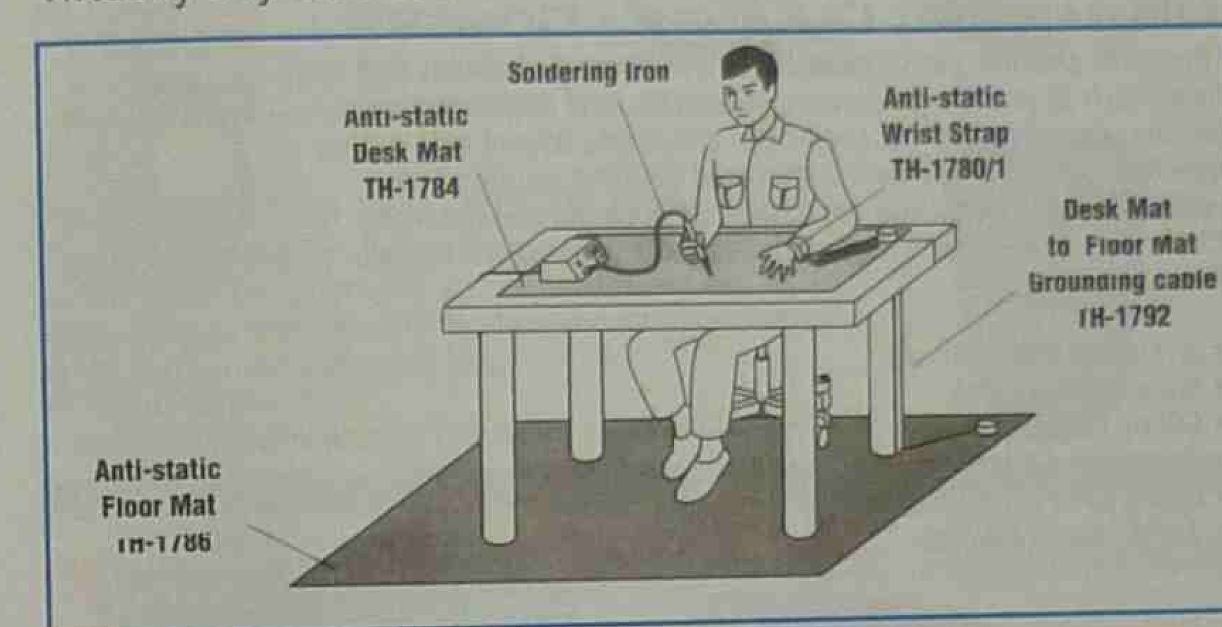
Automatic Centre Punch

This handy tool allows you to make a centre punch mark with one hand.
You simply hold the unit against where you would want a centre punch mark, apply downward pressure until the internal hammer mechanism trips. The pressure is adjustable, depending on the hardness of the surface. Sturdy brass body with hardened steel centre punch. • 125 mm long overall.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1770	$7.95	$6.95	$6.25

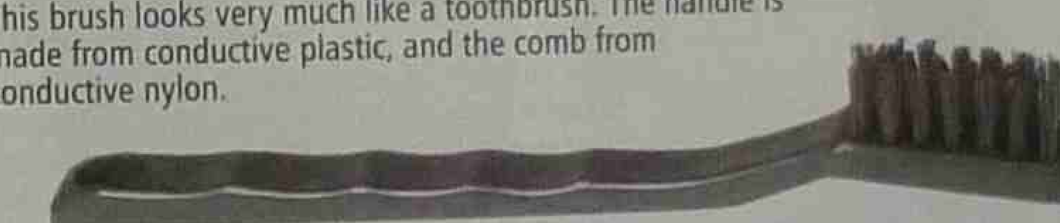
ANTI-STATIC PRODUCTS

Static control - both in the workshop/factory and in the field - is becoming increasingly important for reliable manufacture, repair and or installation of large surface mount LSI based boards. We offer a complete line of static control products for virtually any situation.



Conductive Brush

This brush looks very much like a toothbrush. The handle is made from conductive plastic, and the comb from conductive nylon.



Use it to clean anything where static is a problem.
- Resistivity: - 10^4 - 10^6 Ω.
- Length 178mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1775	$9.95	$8.75	$7.75

Anti-Static Workplace Desk Mat

This anti-static mat measures 555(W) x 290(D)mm and is ideal for anyone who manufactures, repairs or services sensitive electronic equipment. It is about 3mm thick, has a hard wearing face over a dense sponge rubber base. Can also be used as an anti-static base for computer keyboard.
- Top colour is grey.
- See opposite for connecting leads.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1783	$19.95	$17.70	$15.70

Large Anti-Static Workplace Desk Mat

This anti-static dissipative desk mat will cover the whole top of a desk or work station. It measures 1 metre x 0.5 metres x 3mm thick. The mat is grey in colour and made of anti-static foam plastic. The surface of the mat has a slight grain texture, whilst the underside has a checker pattern to prevent slipping on the bench top.
- Resistivity: Anti-static layer 10^8 - 10^{10} Ω/inch.
- Conductive layer 10^4 - 10^6 Ω/inch.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1784	$49.95	$44.90	$39.90

Anti-Static Floor Mat

This anti-static floor mat is made from conductive PVC sheet combined with high density PVC foam. The top layer is grey with green fleck in colour, and the bottom is black. Resistivity is: 10^9 - 10^{11} Ω/inch for the top layer, and 10^4 - 10^6 Ω/inch for the bottom layer.
- Size is 2 metres x 1 metre x 2mm thick.
- See below for grounding leads.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1786	$199.00	$178.95	$158.95

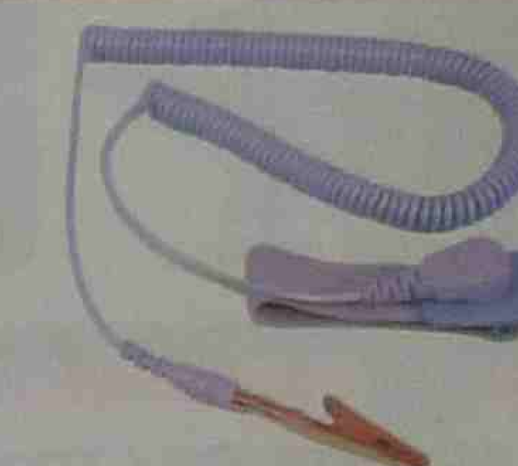
Anti-Static Wrist Strap

Use when static electricity is a problem when soldering, changing ICs and installing expansion cards in computers etc. Static charges can destroy ICs instantly. Consists of adjustable Velcro wrist strap, coiled lead and banana plug/alligator clip.
- Expanded lead length about 1.8m.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1780	$13.95	$12.35	$10.95

Anti-Static Wrist Strap

Similar to the unit above, this strap has an extra long coiled lead of 3m/10ft extended. This makes it ideal for moving about your workbench without having to unclip yourself to do so. An elasticised wrist strap is also included, along with banana plug/ alligator clip.
- Colour may vary.



Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1781	$17.95	$15.95	$14.35

Desk Grounding Cable

This lead is to ground workplace desk mats. It clips over the stud fastener on any mat, and has a receptacle for 2 banana plugs. The other end of the lead has a banana plug, and an alligator clip that slips over the banana plug for earth connection. The anti-static wrist strap then plugs into the socket above the stud fastener clip. 2 wrist straps can be plugged in if required.
- Cable length is approximately 3 metres.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1791	$13.95	$12.35	$10.95

Desk Mat to Floor Mat Grounding Cable

Use this lead if you are using both the floor mat and desk mat. (You will need to have grounded the floor mat). The anti-static wrist strap can then be plugged into this leads double adaptor. Lead has snap connectors both ends - one end with double banana socket (desk mat end).
- Cable length approximately 3 metres.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1792	$13.95	$12.35	$10.95

Anti-Static Field Service Mat/Bag

This mat is ideal for field service people, who need an anti-static work area when on the move. The mat folds out to a work area of approximately 600 x 600mm. At one end there are 2 pouches, and a ground lead and wrist strap are included.
A very useful tool.
- Pouch size approx: 200 x 300mm.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1776	$49.95	$44.90	$39.90

TOOLS

TOOL KITS

Tool Kit With Case 29 Piece

DIY minor repairs are a breeze with this 29 piece tool kit and every DIY guy should have at least one of these in easy reach. The tools are held secure in the stylish silver case so they won't get lost.

- 6 inch ratchet driver bit handle with magnetic tip
- Extension bar
- Screwdriver bits include 2 x Phillips, 2 x Pozi, 2 Slotted drivers, 3 Torx tips, 1 Adaptor for hex to square drive
- 8 Nut drivers from 5 mm -11mm
- Jewellers drivers 2x flat & 1 Phillips
- 2 metre tape measure with lock
- Spring loaded long nose pliers with wire cutter
- Spring loaded side cutters
- Flexible drive shaft
- Retractable knife blade
- Torch requires 2 x AA batteries (not included)
- Case measures 200(L) x 145(W) x 45(H)mm

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2066	$34.95	$31.45	$27.90

Computer Service Tool Kit

Kit includes:

- IC inserter/extractor
- Tweezers
- 1/4" Nutdriver
- 3/16" Nutdriver
- Parts tube for storage
- Pearl Catch
- Double ended 10/15 torx driver
- #0 Phillips screwdriver
- Zipper case
- #1 Phillips screwdriver
- 1/8" Slotted screwdriver
- 3/16" Slotted screwdriver
- Size 220 x 155 x 38mm

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2040	$29.95	$26.90	$23.70

149 Piece Pink Tool Set

new

Everything the handy woman could ever possibly need in one box. Finished in bright pink, the kit contains a hammer, long nose pliers, multigrips, tape measure, screwdrivers, shifting spanner, shears, driver with 20 bits, 8-piece Allen key set, 6 jewellers screwdrivers plus an assortment of nails, screws and other fasteners. Not only has it got the tools, it's got the instructions to go with it! There is an easy-to-follow 20 page instruction booklet on each of the tools and directions for some common household tasks you might use your tools for such as hanging pictures, assembling flat pack furniture and repairing shoes. Just too easy!

- Will turn any girl into a handywoman!
- Case measures: 250(W) x 322(H) x 65(D)mm

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2075	$49.95	$44.90	$39.90

23 Piece Pink Mini Tool Kit

Who says tools are only for blokes? Introducing our elegant 23 Piece Pink tool kit for the handy female! This tool kit is compact and easy to store, it comes in a trendy carry case and is perfect for doing repairs at home, around the office and for travellers. These are good quality tools with rubberised grips to protect the most delicate hands!

Tool kit includes:

- Pink Torch
- Pink Side Cutters
- Pink Long Nose Pliers
- Pink Tape Measure
- Pink Driver Bit Handle with Magnetic Tip
- Extension Bar
- 2 Jewellers Screwdrivers
- Tweezers
- Screwdriver Bits Set
- Socket Set

Pink - just for the ladies!

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2067	$24.95	$22.35	$19.70

HEADPHONES

12DB Noise-Cancelling Headphones

These headphones acheive a dramatic external/background noise reduction. This is done by detecting the external noise & inserting it into the line signal 180° out of phase to the wanted signal. This, and the efficient cushion earpads create an astonishing island of quietness in an otherwise noisy environment. Perfect for noisy environments such as airports, construction/demolition zones, shipping docks or train stations. Included is a standard 3.5mm cable for connection to most audio devices and a gig bag for easy transportation. It also has a 3.5mm to 6.5mm converter. Includes replacement earpads.

- Driver units: 40mm
- Impedance: 163 ohms +/-20%
- Sensitivity: 110dB/mW
- Frequency response: 20Hz to20kHz
- Frequency range of active noise attenuation: 40Hz to 1.5kHz
- Maximum noise reduction level: 12dB at 100Hz
- Operating voltage: DC 1.5V, AAA battery
- Battery life: up to 24 hours

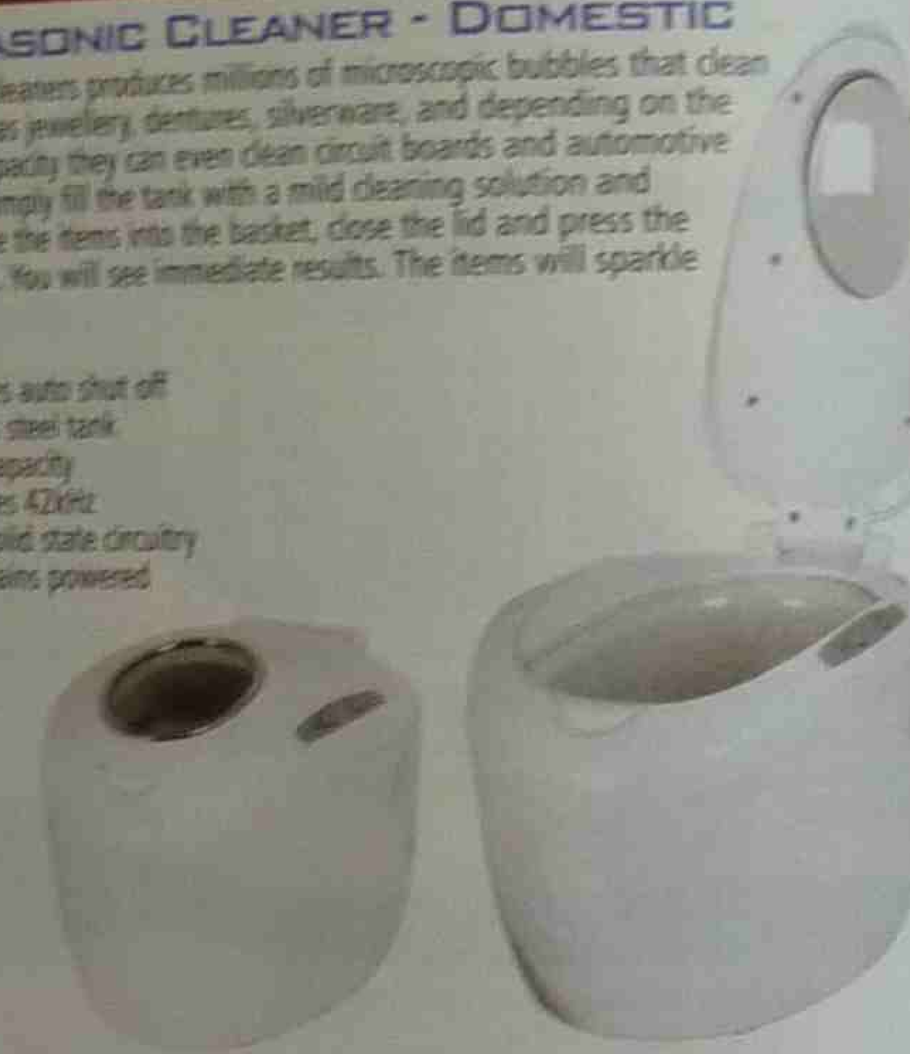
Cat.	Qty 1+	Qty 4+	Qty
AA-2058	$99.95	$89.95	$79.[illegible]

ULTRASONIC CLEANERS

Ultrasonic Cleaner - Domestic

Ultrasonic cleaners produces millions of microscopic bubbles that clean items such as jewellery, dentures, silverware, and depending on the cleaners capacity they can even clean circuit boards and automotive injectors. Simply fill the tank with a mild cleaning solution and water, place the items into the basket, close the lid and press the 'On' button. You will see immediate results. The items will sparkle like new!

- 3 minutes auto shut off
- Stainless steel tank
- 600ml capacity
- Generates 42kHz
- Quiet, solid state circuitry
- 240V mains powered



Cat.	Qty 1+	Qty 3+	Qty 6+
YH-5406	$115.00	$103.95	$91.9[illegible]

Ultrasonic Cleaner - Commercial

If you've been after an industrial ultrasonic cleaner but have been frightened off by the price, then this is for you! Its massive 100W transducer produces millions of microscopic bubbles that are small enough to penetrate the most microscopic of crevices, cleaning them thoroughly. Use this cleaner for automotive injectors, jewellery, glasses, circuit boards and more! The unit features a large LED display with real time countdown. You can also set the cleaning time in 5 minute increments.

- Power supply: 240VAC 50/60Hz
- Power: 100 Watts
- Tank capacity: 3L
- Tank dimensions: 265 (L) x 160(W) x 235(H)mm
- Unit dimensions: 265 (L) x 160(W) x 245 (H) mm (including lid and basket)
- Supplied with: 1 x 1.5m IEC mains lead
  1 x 250(L) x 155 (W) x 90(H)mm stainless steel basket
  1 x stainless steel lid

Cat.	Qty 1+	Qty 3+	Qty 6+
YH-5410	$399.00	$359.00	$319.[illegible]

YELLOW PRICEBAR DENOTES CLEARANCE

POWER TOOLS

4.8 Volt Two Speed 'T' Bar Cordless Screwdriver

You can really pile on the power with this unique T -bar cordless screwdriver. The clever design means that pressure is intrinsically applied along the axis of the screw to minimise slippage or head damage, while the T Bar handle gives you precise control. The power button falls naturally under your thumb and is easily operated for long periods without fatigue. A useful addition to any handyman's toolbox.

- 2 speeds 180 & 800 RPM
- Overall length 200mm
- Forward & reverse
- Mains charger included

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2492	$29.95	$26.90	$23.70

4.8 Volt Cordless Screwdriver

There are many cordless screwdrivers on the market, but few of them are as comfortable or easy to use as the little beauty. The driver has a comfortable pistol grip that gives you maximum control and allows your fingers to fall naturally on the forward/reverse control switch. The driver also has a bright LED lamp, a magnetic bit holder, and a handy security strap. There is even a handy LED battery level indicator that tells you how much charge is left in the battery.

- Voltage: 4.8V
- Bit holder size: 6.35mm
- Mains charger included

POUCH WITH BELT CLIP - SUIT TD-2498

Protect and keep your cordless screwdriver (TD-2498) within reach. This pouch has a strong metal belt clip, a Velcro strap to hold your drill in place and has 6 pockets at the front to store your drill bits.

- Colour: Yellow
- Approximate dimensions: 100(W) x 115(H) x 65(D)mm

	Cat.	Qty 1+	Qty 3+	Qty 6+
Screwdriver	TD-2498	$23.95	$21.35	$18.95
Pouch to suit	TD-2499	$5.95	$5.25	$4.75

3.6V Cordless Driver Drill With Charging Cradle

Everyone should have a spare drill floating around the garage or the toolbox. This one has forward and reverse with a 1/4" hex bit holder instead of a conventional chuck for quick bit changes. It also has a quality 3.6V lithium-ion battery, a desktop charger, charge status LEDs and includes 5 bits: 5mm flat, #2 Phillips, 2, 2.5 & 3mm twist drills plus a general-purpose 1/4" hex bit holder.

Specifications:

Speed:	150 RPM
Battery:	3.6V Li-ion
Charge time:	3 hours approx.
Size:	140 x 140mm

Cat.	Qty 1+
TD-2494	$29.95

3.6V Cordless Screwdriver With Pivoting Handle

Nothing beats the convenience of a cordless screwdriver when you have numerous screws to install or remove from a job.
And what makes this cordless screwdriver unique is the pivoting handle, folding from a straight position for easy storage in a toolbox, to an angle position for better grip. This rechargeable screwdriver also features a powerful motor, forward & reverse switch, safety power lock, 6 torque settings and reversible Phillips/Flat Blade bit. Supplied with a plugpack for recharging. Spare battery also available.

- Voltage: 3.6V
- Speed: >230RPM
- Bit holder size: 6.35mm
- Charging duration: 5 - 7hrs
- Weight: 350gm

	Cat.	Qty 1+	Qty 3+	Qty 6+
Screwdriver	TD-2495	$13.95	$12.35	$10.95
Spare Battery	TD-2496	$6.95	$6.25	$5.50

GLUE GUNS

Large Glue Gun - 240V

Our mini glue gun was so popular we decided to import its big brother. This gun performs just as well as the smaller version but uses a 40% larger diameter glue stick for bigger jobs. Great for quick and easy low stress repairs to timber, cardboard, paper, and many household materials. Powered by 240VAC and Standards Australian approved. Intermittent use only, not for production use.

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1992	$22.95	$20.35	$18.35

Mini Glue Gun - 240V

A glue gun is a handy tool to have around the house. Its fast, easy and simple to use with trigger controlled glue feed.

- Toys and models • Decorations
- Furniture • Woodwork
- Cardboard etc.

Plugs straight into 240V power point
Standards Australia approved
Use glue sticks with 7.4mm diameter
Intermittent use only, not for production use

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1990	$19.95	$17.70	$15.70

Portasol Glue Gun - Gas

A quality, butane powered hot glue gun that can be used almost anywhere. The gun features easy push button electronic ignition so you don't have to fumble with matches, and a translucent fuel tank that lets you see how much gas is left. The glue will bond to most household materials including ceramics, metal, fabric, leather, wood, & plastics.

- 125 Watt equivalent power
- Ready to use in 3 minutes
- 2 hours per refill
- 2 x 11mm dia. Glue sticks included - replacements use TH-1995 or TH-1996
- Weights 200 grams without gas
- Made in Ireland

Cat.	Qty 1+	Qty 3+	Qty 6+
TH-1330	$69.95	$62.95	$55.95

Glue Sticks

FOR MINI GUN

To suit the mini glue gun.
Size 7.4mm dia. x 100mm long

- Suits TH-1990 mini gun

FOR LARGE GUN

Size 11mm diameter x 100mm long.

- Suits TH-1330 & TH-1992 guns

Size 11 x 98mm long

- Suits TH-1992 gun

		Cat.	Qty 1+	Qty 3+	Qty 6+
Pkt 6	For Mini Gun	TH-1991	$3.95	$3.45	$3.15
Pkt 6	For Large Gun	TH-1992	$22.95	$20.35	$18.35
Pkt 50	For Mini Gun	TH-1994	$9.95	$8.75	$7.75
Pkt 6	For Large Gun	TH-1995	$4.95	$4.45	$3.90
Pkt 45 approx.	For Large Gun	TH-1996	$17.95	$15.95	$14.35

5 Piece Grinding Stones

Everything from finishing work to light deburring is easy with these grinding stones.

- 1/8th shaft.

Cat.	Qty 1+	Qty 3+	Qty 6+
TD-2435	$4.95	$4.45	$3.90

related products

LOOKING FOR STORAGE CASES?

Ideal for carrying small components and is an indispensable addition to any tool kit. See through plastic construction enables the contents to be easily identified and the 2 snap action latches secure the lid and keep everything in its rightful place.

See page 206 for the full range.

TOOLS

SHORT CIRCUITS VOLUME II

Cat. BJ-8504 $13.95

Once you have mastered the art of assembling a circuit with springs you will want to tackle projects that are more complex. The benefit of spring termination falls away once you get into projects that have over, say, 20 parts. In addition to this, projects that work on radio frequency, or RF for short, don't like springs as they start to act like little antennae all over the place! So it's time to graduate to the heart of modern electronics - the printed circuit board. This simple device fulfils the function of mounting board & circuit wiring in one. The critical element of success in printed circuit board - or PCB for short - assembly is good soldering. Soldering is not for 8 year olds but its fine if you are older or are under adult supervision. In Short Circuits II we teach you how to solder.

We teach you a lot more as well. Such as:

- How to use a Multimeter**
- What tools you will need.
- What sort of workspace is needed.
- How the circuit works (Tech talk).
- What to do next.
- How to make your circuit work in the real world.

Due to author's Copyright, instructions for each project are not included in the component pack. You can purchase the book, or buy a set of instructions separately.

**A MULTIMETER IS JUST THAT. IT ENABLES YOU TO MEASURE ALL SORTS OF ELECTRICAL UNITS, VOLTS, AMPS, OHMS ETC. THEY ARE THE MOST USEFUL PIECE OF TEST EQUIPMENT YOU WILL OWN AND BASE MODELS ARE NOT EXPENSIVE. JAYCAR HAS A GREAT RANGE WHICH STARTS ON PAGE 9. • ALSO SEE OUR SOLDERING STARTER KIT ON PAGE 72.

SHORT CIRCUITS IS A LEARNING SYSTEM

A key part of the Short Circuits learning system is that your mind is stimulated into learning electronics by building exciting projects and HAVING FUN AT THE SAME TIME. You will get a real "buzz" from the exciting projects offered - 22 in all. These projects have useful functions in the "real world" and you may choose to keep them for a long time because they are so useful. For example, you can make a Bionic Ear amplifier for a grandparent. Listed below are the projects in the book. They have all been designed by professional engineers and tested to work reliably.

Maybe one day you will be an engineer, designing projects for other people.

- Full colour - 148 pages. 205 x 275mm.

related products — *A SUITABLE 9V BATTERY FOR ALL PROJECTS IS OUR SB-2423 ON PAGE 227.

RED/GREEN LED FLASHER

PROJECT #1

Here's a simple and easy-to-build first project. It flashes two LEDs (light emitting diodes) - one red and the other green - alternately, a bit like the light on a police car. A great way to learn how a 555 Timer IC works.

- PCB and Components.
- Required 9V battery*

	Qty 1+	Qty 5+
Cat. KJ-8200	$7.95	$6.95
	Qty 1+	3+ mixed
Instructions KJ-8201	$2.50	$2.45
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

COUNTDOWN TIMER

This neat project enables you to set the time you want to elapse before you want something to happen, say the exposure of a camera shutter or an electrically operated door to stay open. At the end of the timed period (seconds to many minutes) a relay is energized which can switch something on or off.

- PCB and components.
- Requires 9V battery*

PROJECT #2

	Qty 1+	Qty 5+
Cat. KJ-8202	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8203	$2.50	$2.45
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

MINI STROBE LIGHT

No, this unit won't fill a disco with blinding white light. Those things are dangerous and run straight off 240V. This one runs off a 9V battery and gives a surprisingly intense white light - especially in darkness.

Uses state-of-the-art LED.

- PCB and components.
- Requires 9V battery*

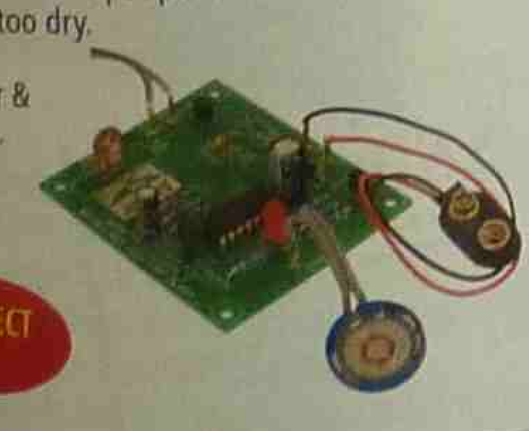
PROJECT #4

	Qty 1+	Qty 5+
Cat. KJ-8206	$18.95	$16.95
	Qty 1+	3+ mixed
Instructions KJ-8207	$2.50	$2.45
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

DROUGHT ALARM

Well, not really a big drought. This will test the soil moisture of a favourite pot plant and an alarm will sound if the soil gets too dry.

- PCB, speaker & components.
- Requires 9V battery*



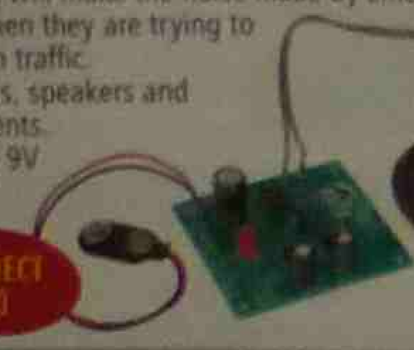
PROJECT #6

	Qty 1+	Qty 5+
Cat. KJ-8210	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8211	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

HEE-HAW SIREN WITH FLASHING LIGHT

This project will make the noise made by emergency vehicles when they are trying to get through traffic.

- PCB, LEDs, speakers and components.
- Requires 9V battery*



PROJECT #3

	Qty 1+	Qty 5+
Cat. KJ-8204	$9.95	$8.75
	Qty 1+	3+ mixed
Instructions KJ-8205	$2.50	$2.45
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

ON-OFF TOUCH SWITCH

This project allows you to turn things on or off simply by touching a small sensor pad with your finger. This could include operating an alarm, water pump, hi-fi speakers, or any other low voltage gadget.

- PCBs, relay components.
- Requires 9V battery*

PROJECT #5

	Qty 1+	Qty 5+
Cat. KJ-8208	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8209	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

LED AUDIO LEVEL DISPLAY

Make your own fancy volume level display you see on DJ mixing desks where the columns of light dance up and down with the music. Connect it to the output of your CD, tape player or radio. Build two of them for displays on your stereo hi-fi system.

- PCB, LEDs and components.
- Requires 9V battery*

PROJECT #7

	Qty 1+	Qty 5+
Cat. KJ-8212	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8213	$2.50	$2.45
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

YELLOW PRICEBAR DENOTES CLEARANCE ITEM

SHORT CIRCUITS II PROJECTS & KITS

ELECTRONIC ORGAN

You can play simple tunes easily once you have built this project! The organ 'keyboard' is actually part of the printed circuit board. It even has vibrato for extra realism.

- PCBs, speaker and components.
- Requires 9V battery*

PROJECT #8

	Qty 1+	Qty 5+
Cat. KJ-8214	$22.95	$20.35
	Qty 1+	3+ mixed
Instructions KJ-8215	$2.50	$2.45
RECOMMENDED BOX - UB1 Cat. HB-6011 $4.45 ea		

"NO BRAINER" AMPLIFIER

This amp is so easy to build, it's a no brainer! It will, however, give a powerful output to bring your Discman, sound card, etc alive! Two required for stereo.

- PCB, speaker and components.
- Requires 9V battery*

PROJECT #9

	Qty 1+	Qty 5+
Cat. KJ-8216	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8217	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

"BEAM ME UP SCOTTY" SIMPLE INTERCOM

This project enables you to talk (i.e. not shout) between parts of a house, or a house and a garage. You can even leave it on, say in the kitchen to monitor a baby's room.

- PCB, speakers, 6-metres of connecting wire and components.
- Requires 9V battery*

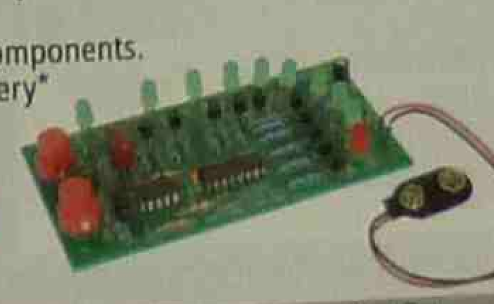
PROJECT #10

	Qty 1+	Qty 5+
Cat. KJ-8218	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8219	$2.50	$2.45
RECOMMENDED BOX - UB3 x 2 Cat. HB-6013 $3.95 ea		

SKEET SHOOT GAME

Here's a fun way to test your hand-eye coordination. The game sends a burst-of-light 'skeet' up along a string of ten LEDs, and you have to try to 'hit' it by pressing the FIRE button at exactly the same time as it reaches the red LED at the end.

- PCB, LEDs and components.
- Requires 9V battery*



PROJECT #11

	Qty 1+	Qty 5+
Cat. KJ-8220	$17.95	$15.95
	Qty 1+	3+ mixed
Instructions KJ-8221	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

SIMPLE FM BUG

Small portable FM microphone. Works surprisingly well. Use it as a baby minder/monitor as well.

- PCB and components.
- Requires 9V battery*

PROJECT #12

	Qty 1+	Qty 5+
Cat. KJ-8230	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8231	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

METRONOME WITH BEAT LED

This neat little metronome does everything that a traditional metronome does. It produces an adjustable 'tick' sound to help you play your music at just the right tempo. It also flashes a LED once per bar, to help you get the music's rhythm right as well.

- PCB, knobs, speaker and components.
- Requires 9V battery*

PROJECT #13

	Qty 1+	Qty 5+
Cat. KJ-8232	$18.95	$16.95
	Qty 1+	3+ mixed
Instructions KJ-8233	$2.50	$2.45
RECOMMENDED BOX - UB1 Cat. HB-6011 $4.45 ea		

JIMINY! WHAT'S THAT CRICKET NOISE?

This fun project makes the distinct sound of a cricket when triggered by a hidden microphone. Someone walks into a room, says something and the next thing this loud cricket starts to chirp! Hours of fun.

- PCB, mic, speaker and components.
- Requires 9V battery*

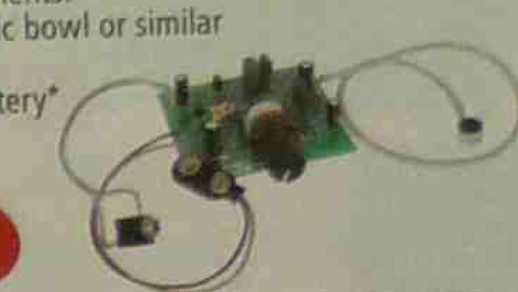
PROJECT #14

	Qty 1+	Qty 5+
Cat. KJ-8224	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8225	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

BIONIC EAR

This project enables you to listen to a conversation (or a bird in a tree, etc) beyond what you would normally hear. It uses the principle of a directional microphone.

- PCB and components.
- Requires a plastic bowl or similar for the reflector.
- Requires 9V battery*



PROJECT #15

	Qty 1+	Qty 5+
Cat. KJ-8226	$17.95	$15.95
	Qty 1+	3+ mixed
Instructions KJ-8227	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

CASINO! ELECTRONIC DICE

Well dice is really plural and this is just one dice - a die. At the press of a button "roll" 6 light emitting diodes flash briefly and only one stays lit, representing a number between 1 and 6. Truly, truly random and never rolls off the table!

- PCB, LEDs and components.
- Requires 9V battery*

PROJECT #16

	Qty 1+	Qty 5+
Cat. KJ-8222	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8223	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

SOUND LIKE A DALEK

This project will change your voice into that metallic sounding robot voice so familiar in the space adventure movies.

- PCB and components. Connect it to the No Brainer amp, (Project 9, KJ-8216) and get really scary!
- Requires 9V battery*

PROJECT #17

	Qty 1+	Qty 5+
Cat. KJ-8228	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8229	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

KNIGHT RIDER LIGHT SCANNER

PROJECT #18

Now you can make the light scanner that goes on the bonnet of that famous car. Looks fantastic in all sorts of places!

- PCB, LEDs and components.
- Requires 9V battery*

	Qty 1+	Qty 5+
Cat. KJ-8236	$19.95	$17.70
	Qty 1+	3+ mixed
Instructions KJ-8237	$2.50	$2.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

RESISTANCE / CAPACITANCE BRIDGE

Measures the value of resistors and capacitors, by comparing them against 'reference' resistors and capacitors. The measurement is carried out by turning a 'balance' potentiometer to adjust the bridge's balancing ratio, and then listening to the bridge's output with a pair of headphones. As you adjust the balance you can easily tell the value of your component from the setting on the balance potentiometer.

- Measures resistance from 10Ω up to over 1MΩ, and capacitor values from below 10pF up to over 10uF.
- PCB and components.
- Headphones required - use AA-2016
- 9V battery required*

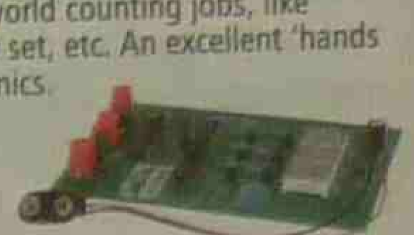
PROJECT #19

	Qty 1+	Qty 5+
Cat. KJ-8240	$27.95	$24.95
	Qty 1+	3+ mixed
Instructions KJ-8241	$2.50	$2.45

EVENT COUNTER 0 - 999

This project demonstrates how 3-digit counters work and can be used for real-world counting jobs, like counting laps on a slot car set, etc. An excellent 'hands on' introduction to electronics.

- PCB, 7 segments LEDs and components.
- Requires 9V battery*



PROJECT #20

	Qty 1+	Qty 5+
Cat. KJ-8234	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8235	$2.50	$2.45
RECOMMENDED BOX - UB1 Cat. HB-6011 $4.45 ea		

FM RADIO WITH ELECTRONIC TUNING

This is a true 88-108MHz FM radio with electronic station tuning and powerful amplifier included! It has a voltage-regulated power supply and works really well. Your friends won't believe you built it.

- PCB and components.
- Requires 9V battery*

PROJECT #21

	Qty 1+	Qty 5+
Cat. KJ-8238	$34.95	$31.45
	Qty 1+	3+ mixed
Instructions KJ-8239	$2.50	$2.45
RECOMMENDED BOX Cat. HB-6032 $7.95 ea		

FM RADIO STATION

This is a Three-Stage radio transmitter that is so stable it could be used as a small radio station. Your audience will be limited to your back yard, however, as there are rules about transmitting power! It includes a microphone but you can transmit other material as well.

- PCB, coils, mic and components.
- Requires 9V battery*

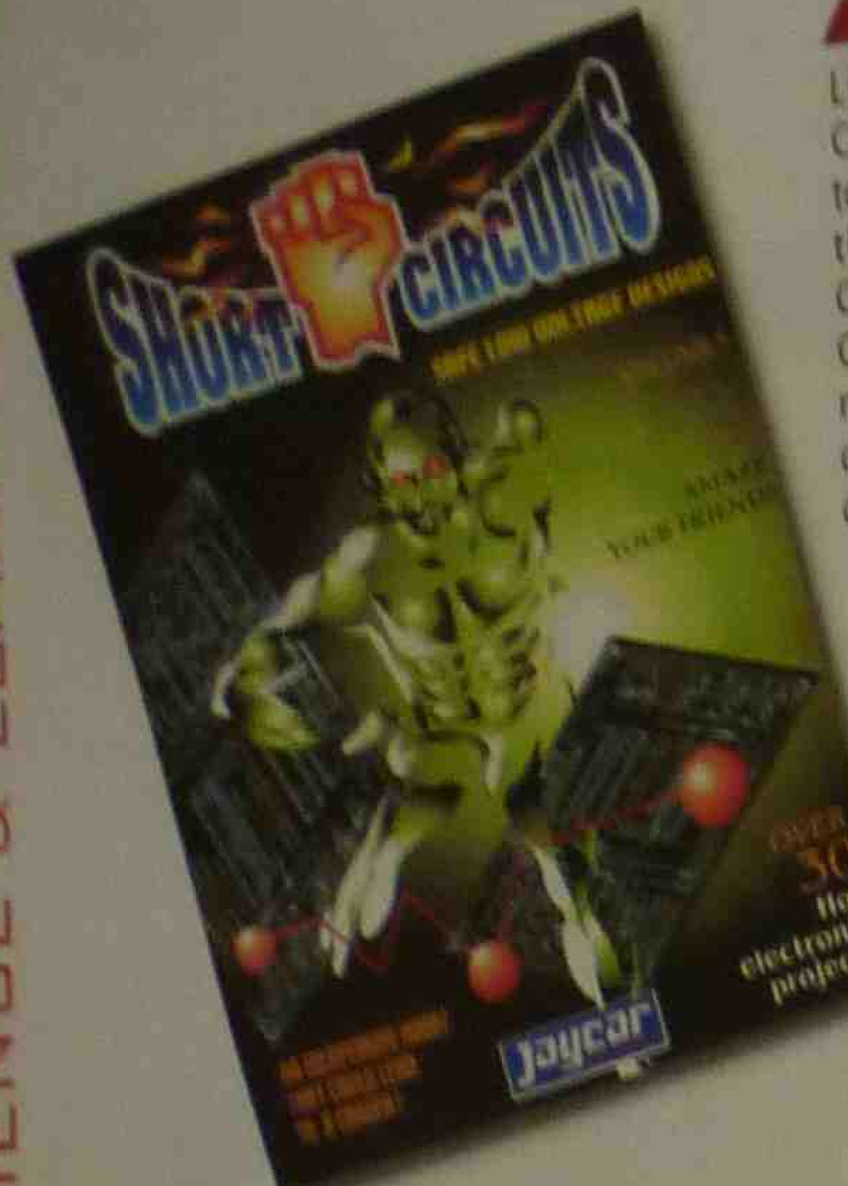
PROJECT #22

Instructions included in kit.

	Qty 1+	Qty 5+
Cat. KJ-8750	$23.95	$21.35
RECOMMENDED BOX - UB5 Cat. HB-6015 $2.95 ea		

SCIENCE & LEARNING

SHORT CIRCUITS VOLUME III

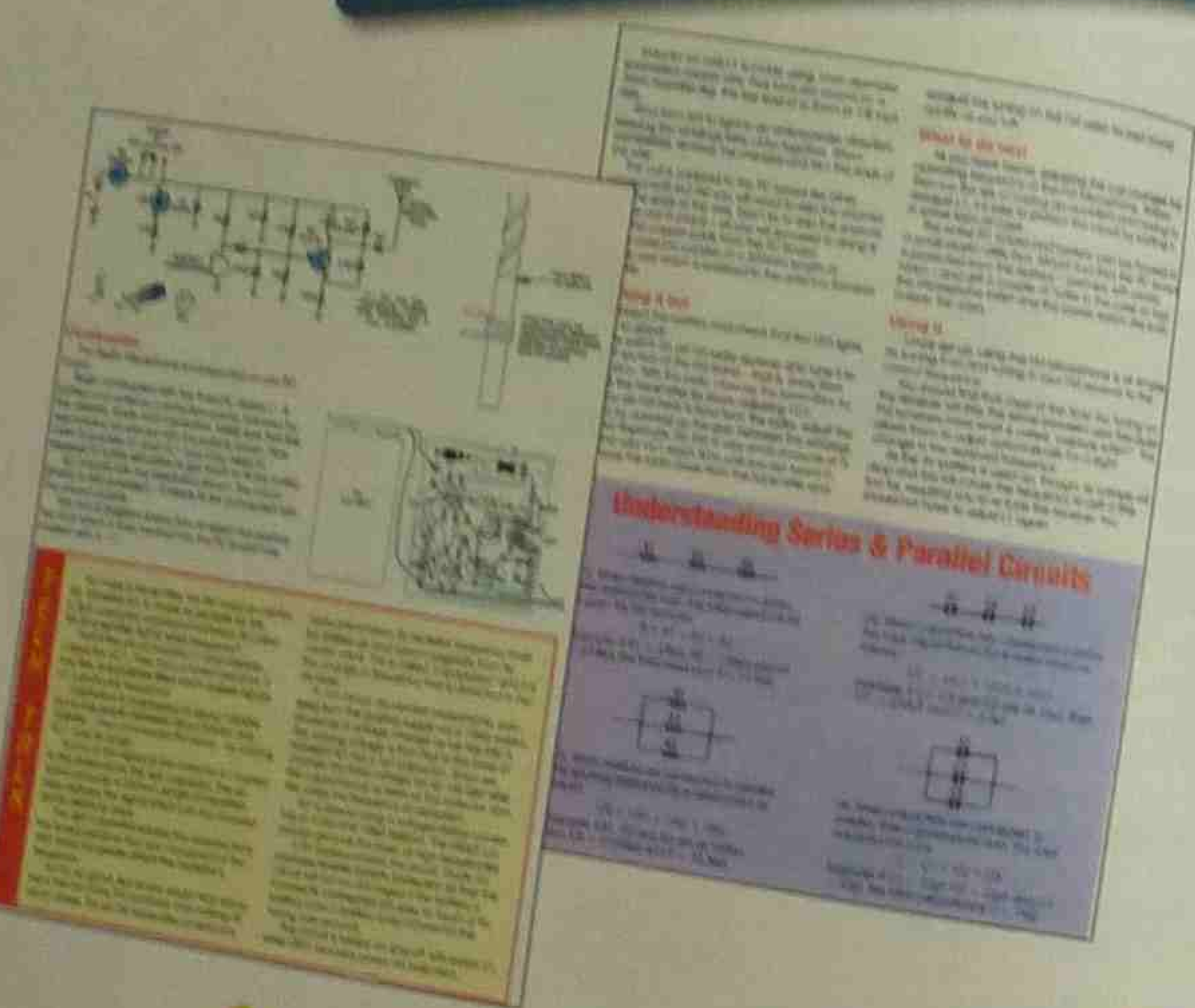


Like our Volumes 1 & 2 book, Short Circuits Volume 3 is sold independently to each of the 30-odd projects. Some of the projects you will build include a Guitar Sustain Unit, a Universal Timer, Car Alarm, Two Lamp Flasher plus many more! There's even a project that demonstrates how you can use your computer to interface to the realworld via your computer's parallel port. Building the projects described in Volume 3 will provide you with the confidence needed to tackle more complex projects, such as those described in the various electronics hobby magazines.

*Instructions to build all the projects are in the book or you may purchase individual instructions at a small charge due to copyright costs.

- Book: Cat. BJ-8505, $19.95
- 128 Pages - Full Colour. 205x275mm

DUE TO AUTHOR'S COPYRIGHT, INSTRUCTIONS FOR EACH PROJECT ARE NOT INCLUDED IN THE COMPONENT PACK. YOU CAN PURCHASE THE BOOK, OR BUY A SET OF INSTRUCTIONS SEPARATELY.



- Step by step instructions show you how to assemble each project
- Techtalk explains the operation of each circuit in detail, to help you gain a thorough knowledge of the "Building Blocks" used in circuit design.
- "What To Do Next" shows you how to modify or further improve your project.
- Basic circuit theory is provided to increase your level of understanding.

* NOTE: SHORT CIRCUITS 3 KITS DON'T CONTAIN BATTERIES. WE WOULD RATHER SEE YOU USE A FRESH BATTERY ONCE THE PROJECT IS COMPLETED.

*INSTRUCTIONS TO BUILD ALL THE PROJECTS ARE IN THE BOOK - IE. YOU NEED THE BOOK TO BUILD THE PROJECTS IF YOU DON'T BUY THE INSTRUCTIONS SEPARATELY.

DASH FLASHER FOR CARS

Deter thieves from breaking into your car with this lamp flasher. It mounts on your car dashboard and starts flashing when the ignition is off.
- PCB, lamp and components.
- 12VDC

PROJECT #1

↗	Qty 1+	Qty 5+
Cat. KJ-8050	$9.95	$8.75
	Qty 1+	3+ mixed
Instructions KJ-8051	$3.50	$3.45
RECOMMENDED BOX - UB5 Cat. HB-6015	$2.95 ea	

ELECTRONIC EYE

Mount across a path or doorway. When a person or object breaks the beam the loudspeaker will sound.
- PCB, LDR, speaker and components.
- 12VDC

PROJECT #3

↗	Qty 1+	Qty 5+
Cat. KJ-8054	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8055	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

INFRARED REMOTE LINK

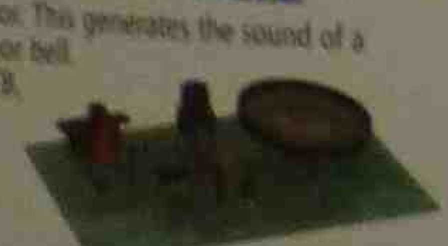
This can be used as either a light beam relay or a remote on/off switch. When the invisible infrared light beam is broken the relay will trigger.
- PCBs and components.
- 12VDC (receiver)
- 9VDC (transmitter, use battery SB-2423)

PROJECT #5

↗	Qty 1+	Qty 5+
Cat. KJ-8058	$28.95	$25.95
	Qty 1+	3+ mixed
Instructions KJ-8059	$3.50	$3.45
RECOMMENDED BOX - UB3 X 2 Cat. HB-6013	$3.95 ea	

DING DONG DOOR BELL

Ideal for the front door. This generates the sound of a classic Ding Dong door bell.
- Project includes PCB, speaker and components.
- 12VDC



PROJECT #2

↗	Qty 1+	Qty 5+
Cat. KJ-8052	$12.95	$11.35
	Qty 1+	3+ mixed
Instructions KJ-8053	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

SIMPLE FM ALARM

Recover your stolen goods using an FM radio. This project transmits a tone over the FM band so that you can track it down using an FM radio.
- PCB and components.
- 9VDC (use SB-2423 battery)

PROJECT #4

↗	Qty 1+	Qty 5+
Cat. KJ-8056	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8057	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

SIMPLE INTRUDER ALARM

This simple design features a normally open and normally closed input, 40 second siren duration and triggered LED indication.
- PCB, buzzer and components.
- 12VDC

PROJECT #6

↗	Qty 1+	Qty 5+
Cat. KJ-8060	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8061	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

SHORT CIRCUITS III PROJECTS & KITS

SCREAMER CAR ALARM

PROJECT #7

A more sophisticated alarm including entry/exit delay, flashing deterrent light, soft warning alarm and deafening internal siren. Thieves will make a hasty retreat!
- PCB, siren, lamp and components.
- 12VDC

↗	Qty 1+	Qty 5+
Cat. KJ-8062	$32.95	$29.45
	Qty 1+	3+ mixed
Instructions KJ-8063	$3.50	$3.45
RECOMMENDED BOX - UB2 Cat. HB-6012	$6.95 ea	

LIGHT CHASER

PROJECT #8

Dazzling display using 10 LEDs that appear to chase each other. Build for a shop, theater or advertising display.
- PCB, LEDs and components.
- 12VDC.

↗	Qty 1+	Qty 5+
Cat. KJ-8064	$12.95	$11.35
	Qty 1+	3+ mixed
Instructions KJ-8065	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

LIGHT SCANNER / CHASER

An impressive display using 20 LEDs that chase forward and backward - just like scanners seen in space movies.
- PCB, LEDs and components.
- 12VDC

PROJECT #9

↗	Qty 1+	Qty 5+
Cat. KJ-8066	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8067	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

REALLY BRIGHT FLASHER

Attract attention with this bright halogen light flasher. The flash rate can be adjusted between 0.5 to 3.5 seconds.
- PCB, 12V halogen lamp with base and components.
- 12VAC 2A (use MP-3051).

PROJECT #10A

↗	Qty 1+	Qty 5+
Cat. KJ-8068	$24.95	$22.35
	Qty 1+	3+ mixed
Instructions KJ-8069	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

TWO LAMP FLASHER

Alternate flashing halogen lamps with an adjustable flash rate between 1 to 4 seconds.
- PCB, two halogen lamps with bases and components.
- 12VAC 4A (use MP-3051)

PROJECT #10B

↗	Qty 1+	Qty 5+
Cat. KJ-8070	$29.95	$26.90
	Qty 1+	3+ mixed
Instructions KJ-8071	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

THE CHAMP AMPLIFIER

A cheap and handy amplifier that can improve low-level signal sources so they can be heard on headphones or a speaker.
- PCB, speaker, volume potentiometer and components.
- 12VDC.

PROJECT #11

↗	Qty 1+	Qty 5+
Cat. KJ-8072	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8073	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

UNIVERSAL STEREO PREAMPLIFIER

If the input signal to your amplifier is not sufficient you will need a preamplifier such as this one to boost the signal.
- PCB and components.
- 12VDC.

PROJECT #12

↗	Qty 1+	Qty 5+
Cat. KJ-8074	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8075	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

HIGH POWER 12V AMPLIFIER

This amplifier-on-a-chip can deliver about 13.5-watts into a 4-ohm speaker. That's quite loud!
- PCB, amplifier IC, heatsink and components.
- 12VDC
- Requires speaker

PROJECT #13

↗	Qty 1+	Qty 5+
Cat. KJ-8076	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8077	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

SIMPLE FM MICROPHONE

This FM transmitter has many possible uses including baby room monitor, wireless microphone or spy bug. It can be picked up on any FM radio.
- PCB, electret microphone and components.
- 9VDC - (use SB-2423 battery).

PROJECT #14

↗	Qty 1+	Qty 5+
Cat. KJ-8078	$12.95	$11.35
	Qty 1+	3+ mixed
Instructions KJ-8079	$3.50	$3.45
RECOMMENDED BOX - UB5 Cat. HB-6015	$2.95 ea	

MINI-MITTER FM TRANSMITTER

Transmit stereo audio from your tape deck or CD player to any FM radio elsewhere in your house. You could even tune in on a portable walkabout radio.
- PCB and components.
- 1.5VDC (use SB-2424 AA battery)

PROJECT #15

↗	Qty 1+	Qty 5+
Cat. KJ-8114	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8115	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

LOW-COST INTERCOM SYSTEM

A low-cost method of communicating between two rooms. The remote unit is hands free so it's also great as a baby monitor.
- PCB, speakers and components.
- 12VDC.

PROJECT #16

↗	Qty 1+	Qty 5+
Cat. KJ-8080	$15.95	$14.35
	Qty 1+	3+ mixed
Instructions KJ-8081	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

ELECTRONIC BONGOS

Sounds just like two traditional bongo drums. Use with the high power 12V amplifier (KJ-8076) for amazing results.
- PCB and components.
- 12VDC

PROJECT #17

↗	Qty 1+	Qty 5+
Cat. KJ-8082	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8083	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

VOX - SOUND SWITCH

This sound-activated switch closes a relay when a loud-enough sound is heard. It switches off when the sound stops. Ideal for radio transmitters or surveillance applications.
- PCB, relay, electret mic and components.
- 12VDC

PROJECT #18

↗	Qty 1+	Qty 5+
Cat. KJ-8084	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8085	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

TRAIN SOUND FX

PROJECT #19

Authentic sounding train whistle and horn sound effects for your model railway.
- PCB, speaker and components.
- 12VDC.

↗	Qty 1+	Qty 5+
Cat. KJ-8086	$27.95	$24.95
	Qty 1+	3+ mixed
Instructions KJ-8087	$3.50	$3.45
RECOMMENDED BOX - UB2 Cat. HB-6012	$6.95 ea	

WIND & RAIN SFX

Simulates the sound of falling rain or howling wind. Must be connected to an amplifier (use KJ-8072 or KJ-8076).
- PCB and components.
- 12VDC.

PROJECT #20

↗	Qty 1+	Qty 5+
Cat. KJ-8088	$19.95	$17.70
	Qty 1+	3+ mixed
Instructions KJ-8089	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013	$3.95 ea	

SHORT CIRCUITS III PROJECTS & KITS

GUITAR LINK
Play the guitar without the need of a guitar amplifier and speakers. This neat preamplifier allows you to connect to virtually any amplifier including your home Hi-Fi.
- PCB, 6.5mm sockets and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8090	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8091	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

GUITAR PRACTICE AMPLIFIER
This low cost amplifier allows you to practice your guitar without annoying the neighbours. It has a guitar type input socket, small speaker and volume.
- PCB, speaker, 6.5mm socket and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8092	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8093	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

GUITAR TWANG-O-MATIC
Add an unusual effect to your guitar that's different from the run-of-the-mill effects. It sounds somewhat reminiscent of an Hawaiian guitar.
- PCB, 6.5mm sockets and components.
- Requires amplifier (use project 13 - KJ-8076)
- 12VDC

	Qty 1+	Qty 5+
Cat. KJ-8094	$21.95	$19.70
	Qty 1+	3+ mixed
Instructions KJ-8095	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

GUITAR WAA-WAA
Add the popular Waa-Waa effect to your guitar to sound really distinctive.
- PCB, 6.5mm sockets and components.
- Requires amplifier (use project 13 - KJ-8076)
- 12VDC

	Qty 1+	Qty 5+
Cat. KJ-8096	$18.95	$16.95
	Qty 1+	3+ mixed
Instructions KJ-8097	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

UNIVERSAL TONE CONTROL
This tone controller can be set to enhance the low or high frequencies of your music. It can be used for guitar or any other audio source.
- PCB, 6.5mm sockets and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8098	$17.95	$15.95
	Qty 1+	3+ mixed
Instructions KJ-8099	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

GUITAR SUSTAIN UNIT
Add this sustain unit to your guitar to make it sound more fulfilling. Features adjustable Decay and Attack.
- PCB, 6.5mm sockets and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8100	$23.95	$21.35
	Qty 1+	3+ mixed
Instructions KJ-8101	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

GUITAR DISTORTION UNIT
Distortion is popular with many guitarists. It can make the music sound quite different.
- PCB, 6.5mm sockets and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8102	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8103	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

REGULATED +12V POWER SUPPLY
Many of the "Short Circuits 3" projects require 12VDC power. This low cost power supply is ideal to power these projects.
- PCB, +12V regulator and components.
- 11.5 - 0 - 11.5VAC 1.25A (use MP-3055)

	Qty 1+	Qty 5+
Cat. KJ-8104	$12.95	$11.35
	Qty 1+	3+ mixed
Instructions KJ-8105	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

REGULATED +/- 12V POWER SUPPLY
Some applications need a positive and negative supply. This circuit will supply a +12V and -12V rail.
- PCB, +/- 12V regulators and components
- 11.5 - 0 - 11.5VAC (use MP-3055)

	Qty 1+	Qty 5+
Cat. KJ-8106	$14.95	$13.35
	Qty 1+	3+ mixed
Instructions KJ-8107	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

UNIVERSAL TIMER
This timer is adjustable between 1 second to 3.5 minutes. An LED illuminates during the timing period and a buzzer sounds at the end.
- PCB, buzzer and components.
- 12VDC.

	Qty 1+	Qty 5+
Cat. KJ-8108	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8109	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

WAVEFORM GENERATOR
This low-cost signal generator produces adjustable square and triangular waves between 100Hz-20kHz. Ideal for testing, trouble-shooting and developing circuits.
- PCB and components.
- 12VDC.

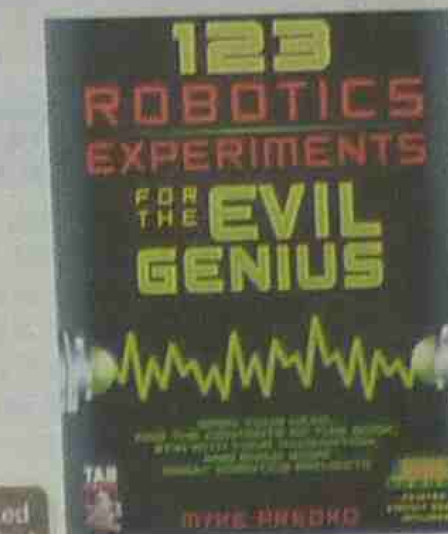
	Qty 1+	Qty 5+
Cat. KJ-8110	$16.95	$14.95
	Qty 1+	3+ mixed
Instructions KJ-8111	$3.50	$3.45
RECOMMENDED BOX - UB3 Cat. HB-6013 $3.95 ea		

SOLDERING STARTER KIT
After Short Circuits 1, the next two series' require you to start soldering components to the printed circuit board. So you'll need a good soldering iron. This Soldering Starter Kit is ideal...

- 1 x KC-5178 Clifford the Cricket kit
- 1 x TH-1886 Precision cutters
- 1 x TH-1502 Soldering stand
- 1 x TS-1465 240V 25W Soldering iron
- 1 x 15g pack 60/40 0.71mm solder

	Qty 1+	Qty 3+	Qty 6+
Cat. TS-1655	$49.95	$44.90	$39.90

123 ROBOTICS EXPERIMENTS FOR THE EVIL GENIUS
Frankly, the "Evil Genius" series of books started a few years ago and the publisher realised that it grabbed the attention of many. We reviewed one early title and were horrified at the VERY DANGEROUS projects described therein. This book, however, is an absolute delight. Written by the brilliant Myke Predko, the book starts with the basic concepts of robotics. In many ways it starts at "Too Simple". We mean this because Myke describes a couple of silly projects to start, i.e. a "doll" made out of the cardboard tubes from toilet paper rolls. He gets into gear quickly, however. (Start reading from page 15 and have an inches metric conversion table handy!) The projects & concept explanations come at a fast pace. You will learn about all the concepts needed for basic robotics technology & have all of the components described to you as well. You will get the basics at a bit of a rush - so that you can get to real robotic control - through the BASIC Stamp microcontroller. Before you know it, you will have serious knowledge of robotics. Indeed this book even includes a double-sided plated-through PCB that you will need for many projects. A board such as this could cost almost as much as the book itself if sold separately. So let's be tough & call the book 121 Robotics Projects. Either way it is THE book for someone who wants to get into this fascinating technology.

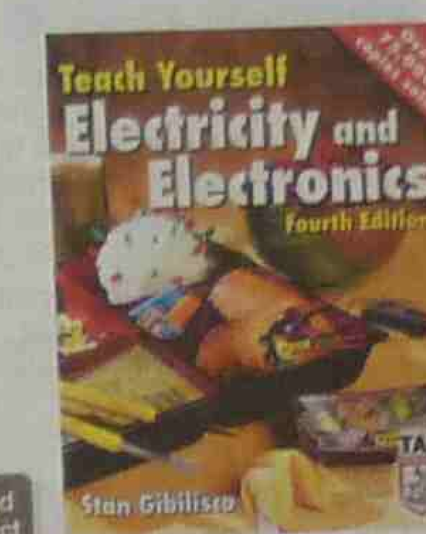


- 355 pages.
- Softcover 275(H) x 215(W)mm.

related subject book

	Cat.	Qty 1+
	BM-7104	$49.95

TEACH YOURSELF ELECTRICITY AND ELECTRONICS - FOURTH EDITION
Comprehensive, straightforward and well-written, this book covers everything from atomic structure to DC and AC theory, semiconductors, integrated circuits, digital electronics and communications. Each chapter has a quiz at the end so you can test your knowledge of each subject. One of the best entry-level self-teaching textbook on electronics available.

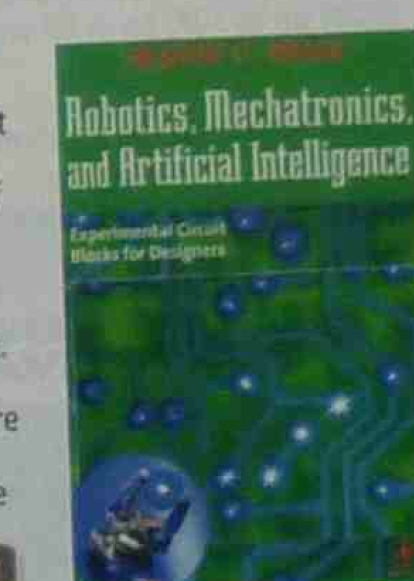


- Softcover, 699 pages with illustrations.
- 232 x 190mm.

related subject book

	Cat.	Qty 1+
	BM-7108	$59.95

ROBOTICS, MECHATRONICS, AND ARTIFICIAL INTELLIGENCE
If you are starting robotics from ground zero, or near it, this is your book. We have never seen a book before that encapsulates a subject from beginner to serious constructor in one volume. This book encompasses all of the basic control systems - along with the philosophy behind them - to enable the interested reader to get a serious grip on how to make a robot move, sense, lift, control and otherwise relate to an external environment. The format is extremely practical in that the author provides - literally - hundreds of useful circuits, which are invaluable to practical design. Indeed the components specified are mostly available from us. Also included are review questions at the end of each chapter to sharpen understanding.



- Softcover 230 x 150mm
- 297 pages

related subject book

	Cat.	Qty 1+
	BT-1370	$119.00

PIC ROBOTICS A BEGINNER'S GUIDE TO ROBOTICS PROJECTS USING THE PIC MICRO
This book enables you to master the specialised skill of programming the most popular robotic microcontroller, the PIC series. If you are serious about robotics, you will need to tackle this at some stage as robotics is all about machine intelligence. Luckily, no Assembly Language programming is required to complete the subject matter of the book. There are 14 chapters in all, including: Robot Intelligence, Testing PIC Performance, Speech Recognition, Colour Robotic Vision and 3 on specific robot controller construction projects.

- Softcover 230 x 185mm
- 274 pages

related subject book

	Cat.	Qty 1+
	BT-1367	$59.95

ROBOT BUILDER'S SOURCEBOOK
If you are going to really get serious about robot construction, you must have this massive book! Written by Gordon McComb, author of the classic (but slightly outdated) Robot Builders Bonanza, Robot Builders Sourcebook is totally new and up-to-date. It lists suppliers of all the bits 'n pieces, you are ever likely to need to pursue the construction of your dream machine, over 150 separate categories. It also includes lists of relevant books, societies, help groups, services, you name it. You not only get component sources, you get materials sources, as well as manufacturing facilities. In all, over 2,500 sources for robot parts. Apart from this, there are dozens of helpful primers on likely sources of useful parts, such as old VCRs etc. We think that book is also an unbelievable bargain.

- Softcover 280 x 215mm
- 711 pages

related subject book

	Cat.	Qty 1+
	BT-1365	$59.95

BOOK - PROGRAMMING & CUSTOMISING THE PIC MICROCONTROLLER
Tap into the latest advances in PIC technology with the fully revamped third edition of this brilliant book. Certainly the definitive text on the subject, this indispensable volume comes with more than 600 illustrations and provides comprehensive, easy-to-understand coverage of the PIC microcontroller's hardware and software schemes. With 100 experiments, projects and libraries, you get a firm grasp of PICs, how they work and the ins-and-outs of their most dynamic applications. This updated edition features a streamlined, more accessible format and concentrates on the three major PIC families, to help you fully understand Assembly, BASIC, and C programming languages. It also covers the latest program development tools and includes a refresher in electronics and programming as well as reference material.

new

Topics covered:
- Setting up your own PIC microcontroller development lab
- PIC MCU basics
- PIC microcontroller interfacing capabilities, software development, and applications
- Useful tables and data
- Basic electronics • Digital electronics
- BASIC reference • C reference
- 16-bit numbers
- Useful circuits and routines to get your applications up & running
- Softcover
- 1263 pages.

related subject book

	Cat.	Qty 1+
	BT-1347	$94.95

REDUCER MODULES - 24 OR 13.8VDC

1.1 AMP CONTINUOUS
With the aid of this module it is feasible to operate 12V devices on 24V truck and marine batteries.

Input voltage:	24 - 30 VDC
Output voltage:	12VDC approx
Maximum load:	1.1A
Dimensions:	60 x 55 x 20 mm.

related products SEE PAGE 239 FOR FULL RANGE OF DC - DC CONVERTERS

	Cat.	Qty 1+	Qty 5+
	AA-0265	$22.95	$20.35

3 AMP 24V TO 12V REDUCER
Many trucks and boats use 24V systems. These converters will allow you to run a reasonably sized 12V car-stereo or other devices from a 24V supply. Heatsink (not included) should be used for superior performance.

1 year warranty

- Short-circuit protection
- Thermal cutout.
- Input voltage: 24 to 26 VDC
- Output voltage: 12VDC
- Maximum load: 3A
- Dimensions: 135 x 50 x 24 mm
- Heatsink: 50 x 100 x 30 mm (rec. size) use Cat. HH-8572 or similar

	Cat.	Qty 1+	Qty 5+
	AA-0266	$29.95	$26.90

6.35mm (1/4") Shaft Coupler

Sturdy solid steel sleeve with black japanned finish, this device connects 2 x axial shafts together that are closely aligned by hex drive set screws.

- 19mm (3/4") long x 12.7mm (1/2")dia.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2790	$12.95	$11.35	$10.35

Spider Coupler Set

Connects a motor to a shaft that may be slightly misaligned or under length. You may also need a torsional shock absorbing coupling which this does as well.

- Each hub will accept a 6.35mm (1/4") shaft with set screws anchor.
- The hubs are coupled by a synthetic rubber spider.
- 20mm OD x 30mm long.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2782	$19.95	$17.70	$15.70

Aluminium Hub with Set Screws

This device will accept a 6.35mm (1/4") shaft, set screw fixed, and connect to the flat face of a gear, typically the flat side of the larger 48 pitch gears shown above.

- Coupling from the gear to the hub by 4 x set screws provided. This enables a motor to be securely fastened to a driving gear.
- Hub 7/8" (22.5mm) dia.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2784	$12.95	$11.35	$10.35

Quality German Made Power Transmission Components

These precision engineered components are a delight to play with. They offer solutions to small power transmission problems. We make no apologies for the price because well, they are German but they are magnificent quality.

Solid Shaft Couplers (Female)

Machining from solid brass stock – 16mm long, 12mm diameter. Type I will connect 2 shafts on a common axis. Type II will reduce from 6.0 diameter to 4.0 diameter. Shaft fixed with grub screws.
Cat. YG-2600 Type I
Cat. YG-2602 Type II

	Cat.	Qty 1+	Qty 5+	Qty 10+
Type I	YG-2600	$12.95	$11.35	$10.35
Type II	YG-2602	$12.95	$11.35	$10.35

Right Angle Shaft Couplers (Male)

Basically a right-angle cast housing with 45 degrees nylon bevel gears permanently coupled. One shaft is enclosed with a panel-mount style threaded bush (similar to a potentiometer mount) which is 10mm long. Two mounting nuts and a lock washer are provided. All metal parts are nickel plated.
Each shaft is 10mm long.

	Shaft	Cat.	Qty 1+	Qty 5+
Type I	2 x 6mm	YG-2610	$39.95	$35.95
Type II	2 x 4mm	YG-2612	$49.95	$44.90

Single Stage Classic Style Universal Joint (Female)

This unit consists of 2 nylon moulded fixed coupling pieces with brass ferrules moulded in for extra strength. They will accept 6.0 dia. Shafts and are fixed by grub-screws. The flexible coupling consists of a self lubricating brass block fixed with hardened pins. Couplers such as these work best when the shafts are not offset and the max. angle is not over 30 degrees.

Cat.	Qty 1+
YG-2606	$34.95

Double Stage Classic Style Universal Joint (Female)

Similar to YG-2606 but with 2 brass couplings set inside a nylon spider. It will couple up to 90 degrees but it will wear quickly at high speed at this angle.

- Works best between 80° and 45°.
- Grubscrews fixing 6.0dia shafts.

Cat.	Qty 1+	Qty 5+
YG-2608	$49.95	$44.90

All-Metal Universal Joint (Trunnion Style) Female 6.0 Dia.

This is a low torque coupler that will couple offset shafts by up to 2mm and 15° off-angle. Shafts are clamped by collet-type fixing i.e. a nut compresses a slotted bush compressing it onto the 6.0 dia. shafts.

- Nickel plated.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2604	$24.95	$22.35	$19.[illegible]

Drive Shafts

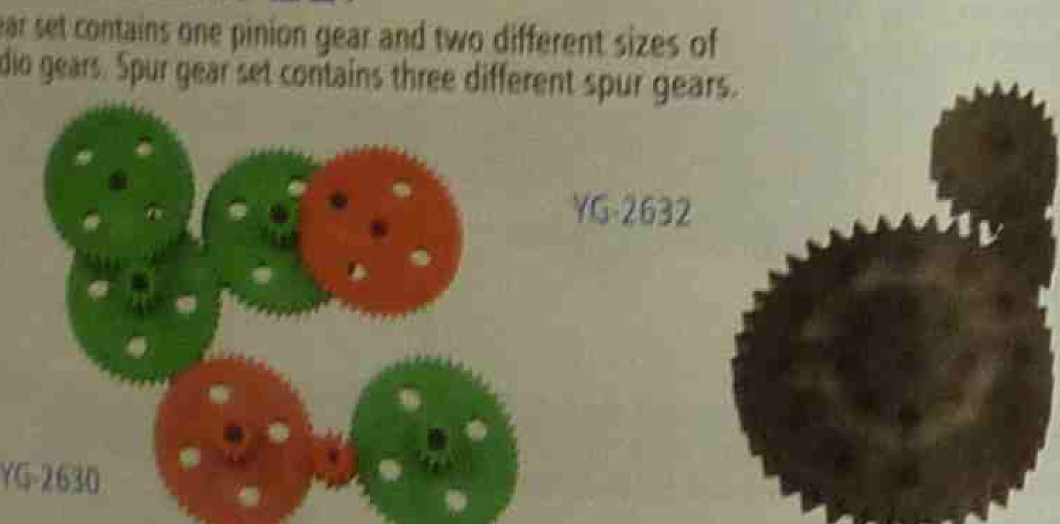
Plastic shafts, handy for use in drive systems or to use with a coupler to extend the length of a pot shaft. 6mm diameter.

Two lengths available:
YG-2620 60mm long
YG-2622 120mm long

	Cat.	Qty 1+	Qty 5+	Qty 10+
60mm long	YG-2620	$8.95	$7.95	$6.95
120mm long	YG-2622	$9.95	$8.75	$7.75

Gear Set and Spur Gear Set

Gear set contains one pinion gear and two different sizes of radio gears. Spur gear set contains three different spur gears.



	Cat.	Qty 1+	Qty 5+	Qty 10+
Gear Set	YG-2630	$10.95	$9.75	$8.75
Spur Set	YG-2632	$8.95	$7.95	$6.95

Worm Drive Set

This set consists of a worm gear 24mm long x 12mm dia with an 8 turn worm screw. The worm has a 2.8mm axle hole which could probably be carefully drilled out to a larger diameter. The spur gear is 1/4" (6.35 wide) and 1 & 5/8" (41.2mm) diameter with a number of regular pitch-concentric 2.8mm holes across the face of the gear including one dead centre. It would be drilled out to accommodate a larger shaft. This gear has 40 teeth. Both are moulded in an engineering grade plastic.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2736	$7.95	$6.95	$6.25

7 Piece Drive / Belt Set

This set consists of 4 x split pulleys. There are 2 sets of 30(dia.)mm; one has a 2.8mm centre hole, the other 4.0mm. These pulleys have brass & plastic bushing sets as well. The smaller pulley sets are 13.8 (dia.) & 8.0 (dia.)mm respectively. A 70 (dia.)mm rubber drive belt is also included.

Cat.	Qty 1+	Qty 5+	Qty 10+
YG-2640	$24.95	$22.35	$19.70

More than ever No.1 for kits

Jaycar Electronics

Our range of kits this year does not differ all that greatly from last years'. There are two reasons for this. One we have decided to concentrate on producing even better quality kits than ever and Two, we have focused on only the best projects that the electronics magazines have generated.

The new kits this year include a new 12/24v DC motor controller (40A!), a motorbike capacitor discharge ignition, a new 135W RMS power amp, an upgraded Theremin, a fantastic school 40km/hr speed zone alert and several others. So good is the School Speed Zone alert, we think that it may be commercially manufactured.

We are delighted to say that our Short Circuits series continues to do well along with the other ranges of "Science" based learning aids. As learning aids go, we implore you to look at our range of anatomically correct human figures with removable organs, etc. Where the definition of a "kit" begins and ends, we do not know!

As we said last year, if you are a parent or are a brother or sister etc, or just know a bright young person it could be a wonderful thing to make them a gift of one of these products. You could inspire the person to engage in a future career in engineering, science, medicine or some other worthwhile pursuit.

Jaycar kits are now even more superior to our diminishing competitors' products.
We put more into them, like metal film (not carbon film) resistors, pre-tested components and I.C. sockets (where technically feasible) and clear, plain English instructions. (Our kits are sold around the world and this is a must).

You are guaranteed to get a quality well-engineered product that will perform.
Trust only Jaycar kits.

Kit Index

Explanation of Kit Icons

SIMPLE - These kits are straightforward to assemble, generally built in one session and will require some basic electronics knowledge and soldering skills. Most are battery or AC adaptor powered. They may not have enclosures (i.e."boxes") or front panels.

DETAILED - Similar to the above but basically bigger. They are either "short forms" of larger projects or are fully operational projects with enclosures, front panels, etc. A greater knowledge of electronics is required to understand their operation and "troubleshoot" if necessary.

COMPLEX - All big projects which by their nature are complex, require several sittings to complete and may require some mechanical assembly. A good knowledge of electronics and manual dexterity is necessary to ensure success.

BATTERY INCLUDED - Batteries are included with the kit.

KWIK KIT - A new low cost fast assembly range of kits. Kwik kits are a fully imported range of kits that can be assembled in just one sitting, sometimes only minutes! All printed circuit boards are screened with a component overlay. The instructions assume component knowledge & the ability to follow simple schematic diagrams.

BUILD TIME - We understand that your time is valuable. We have therefore decided to provide you with an estimated build time for all our kits. Each kit includes a stopwatch logo that clearly indicates construction time. We know that you will find this useful.

240V POWERED - All mains kits have this logo. We do this because we feel it is necessary that only experienced kit constructors should attempt mains powered projects. It is generally a good idea to have a competent colleague check your work before applying power.

AUTOMOTIVE KIT - Kit is specifically designed for automotive (which could mean motorcycle, caravan or boat as well).

SMD COMPONENT - Kit involves soldering a SMD (Surface Mount Device). Suitable soldering iron for SMD soldering and proper handling techniques are required.

Guarantee on Kits

Most kits are successfully constructed with few problems. However, we recommend that if a kit builder does not have enough knowledge to diagnose faults, that the project should not be started unless assistance can be obtained. Unfortunately, one small faulty solder joint or wiring mistake can prevent a project from operating, which would take hours to locate and at normal service rates the service charge could well be more than the total cost of the kit. If you believe that you may have difficulty in building a kit (which is simply a complete set of separate parts made up to a list provided by the major electronics magazines) and you cannot get assistance from a friend, we suggest you return the kit to us in its original condition for a refund under our satisfaction guarantee. Unfortunately, kits cannot be replaced under our satisfaction guarantee once construction has been commenced. However, should you find any faulty parts (only), we will gladly replace them under warranty.

Back Catalogue of Kits

MORE INFO AVAILABLE ONLINE

If you can't find the kit you're looking for, try the Jaycar Kit Back-catalogue.
To keep up with the constant flow of new kits and maintain a contemporary in-store line-up of projects, older and slow moving kits are returned to our central warehouse.

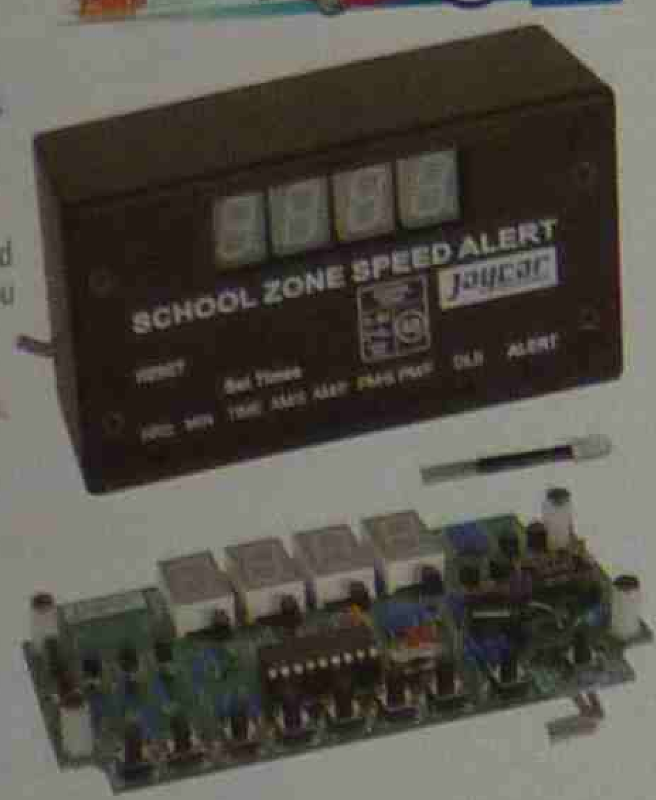
Back-catalogue kits are not available in Jaycar stores. You can only access them on our website. You can order online at www.jaycar.com.au or by calling our Techstore on 1800 022 888 or use the order form on page 433.

AUTOMOTIVE KITS

Pimpin' your ride has never been this easy and affordable. Boost performance? Check. Better monitoring? Check. An annoying-as-hell car alarm? We got that covered too. Whether you're a budding car enthusiast or a diehard DIY tragic, we're sure to have a well-engineered electronic kit to tickle your automotive fancy.

SCHOOL ZONE SPEED ALERT KIT

Ref: Silicon Chip Magazine April 2009
Basically a specialised timer that alerts you with a flashing LED when school zone-reduced speeds are in force. The unit will flash for the whole time the restrictions operate in the morning and afternoon. The only visible sign for you is a dashboard flashing LED; all other components can be mounted remotely, including the disable switch.



- PCB, programmed micro, case with machined and screen printed lid and components.
- 12VDC

new

	Cat.	Qty 1+	Qty 5+
	KC-5472	$49.95	$44.90

FUEL CUT DEFEATER

Refer Silicon Chip Magazine February 2007
Many factory turbo fitted cars have a limit to which the boost level can reach before a 'fuel cut' is activated by the vehicles ECU. This cheap and simple kit enables you to eliminate this factory fuel cut and go beyond the typical 15-17psi factory boost limit. The kit simply intercepts the MAP sensor signal, and trims the signal voltage above 3.9V to avoid the ECU cutting the fuel supply to the engine. No adjustment is required, and with only four wires to connect, installing this kit couldn't be easier. Note: care should be taken to ensure the boost levels and fuel mixture don't reach an unsafe level that will permanently damage your engine.

- PCB and components. • 12VDC

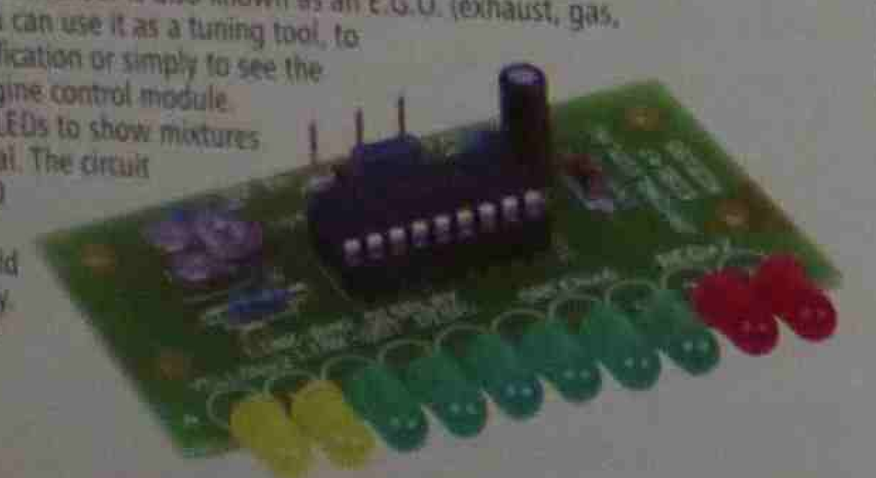
	Cat.	Qty 1+	Qty 5+
	KC-5439	$19.95	$17.70
RECOMMENDED BOX	HB-6065	$2.65 ea	

MIXTURE DISPLAY KIT FOR FUEL INJECTED CARS - IMPROVED

- Monitor your car's air-fuel ratio in real time.

THOUSANDS SOLD

Refer Silicon Chip Magazine November 1995 & Electronic Projects for Cars Vol 2 - Silicon Chip
This very simple kit will allow you to monitor the fuel mixtures being run by your car. This type of sensor is also known as an E.G.O. (exhaust, gas, oxygen) monitor. You can use it as a tuning tool, to help in vehicle modification or simply to see the behaviour of the engine control module. Indication is via 10 LEDs to show mixtures rich, lean and normal. The circuit connects to the EGO sensor mounted in the exhaust manifold and the cars battery. Now features extra circuit protection.



- PCB, LEDs and components.
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5195	$16.95	$14.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

VARIABLE BOOST KIT FOR TURBOCHARGERS

Refer Silicon Chip Magazine February 2007
Give your turbocharged car more giddyup with this variable boost controller. It's a very simple circuit with only a few components to modify the factory boost levels. It works by intercepting the boost signal from the car's engine management computer and modifying the duty cycle of the solenoid signal. Kit supplied in short form with PCB and overlay, and components.

- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5438	$19.95	$17.70
RECOMMENDED BOX	HB-6065	$2.65 ea	

HEADLIGHT REMINDER FOR CARS

- No more flat car batteries

Refer Silicon Chip Magazine August 2001
Nothing is more frustrating than getting into your car early in the morning, only to discover that you had left your headlights on the night before, running your car's battery flat. Features include a modulated alarm, ignition and lights monitoring, optional door switch detection, time-out alarm & a short delay before the alarm sounds. Build and install this hassle-saving kit & enjoy a feature in your car that many luxury vehicle owners have long taken for granted.

- PCB, case with screen printed lid and components.
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5317	$27.95	$24.95

COURTESY INTERIOR LIGHT DELAY MK II

- Interior light delay like those expensive models

Refer Silicon Chip Magazine June 2004.
Many modern cars feature a time delay on the interior light. It still allows you time to buckle up and get organised before the light dims and finally goes out. This kit provides that feature for cars which don't already provide it. It has a soft fade out after a set time has elapsed, and features much simpler universal wiring than previous models we have had.

- PCB and components.
- Suitable for circuits switching ground or V+
- Car & truck with negative chassis. • 12VDC or 24VDC

	Cat.	Qty 1+	Qty 5+
	KC-5392	$19.95	$17.70
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

DIGITAL FUEL MIXTURE DISPLAY

- Monitor your car's air-fuel ratio in real time.

Refer Silicon Chip Magazine Sept / Oct 2000
This brilliant dashboard mounting unit monitors and displays your car's air-fuel ratio in real time on a 3 digit display and at the same time on the programmable 10-LED bargraph. Indicates ratios between 11.8 - 20.6 for petrol and 12.7 - 21.5 for propane/LPG. Some features: fully lean and fully rich indication, fast 220ms update time for bargraph, 440ms upgrade time for 3 digit display, 0-1V display for setting up adjustment, dot or bar option for bargraph and auto display dimming for night driving. All this in a compact case measuring only 83 x 54 x 31mm. Leaded petrol engines will soon poison an EGO sensor, so it is recommended that this kit be used for tuning purposes on those cars.

- PCBs, black case with laser cut & screen printed front panel, programmed PIC micro, 7 segment displays, red acrylic display panel, hook-up wire and components.
- Requires EGO sensor with 0-1V output.
- 12VDC.

	Cat.	Qty 1+	Qty 5+
	KC-5300	$67.95	$60.95

12-24V HIGH CURRENT MOTOR SPEED CONTROLLER KIT

Refer Silicon Chip Magazine March 2008
Want to control a really big DC motor? This design will control 12 or 24VDC motors at up to 40A continuous. The speed regulation is maintained under load, so the motor speed is maintained even under heavy load. It also features automatic soft-start, fast switch-off, a 4-digit LED 7-segment display to show settings, an overload warning buzzer & a low battery alarm. All control tasks are monitored by a microcontroller, so the functionality is extensive.

- Double sided plated through hole PCBs, programmed micro and components
- 12VDC or 24VDC

	Cat.	Qty 1+	Qty 5+
	KC-5465	$89.95	$80.95
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

10A 12VDC MOTOR SPEED CONTROLLER

Refer Silicon Chip Magazine June 1997
Use this kit for controlling 12V DC motors in cars such as fuel injection pumps, water/air intercoolers and water injection on performance cars. You can also use it for headlight dimming and for running 12V DC motors in 24V vehicles. The circuit incorporates a soft start feature, which is used to reduce inrush currents, particularly if the kit is to control 12V incandescent lamps. As presented, the kit will control loads up to 10 amps, although the addition of an extra MOSFET transistor will double that capacity to an amazing 20 amps.

- PCB and components to build the 10A version.
- Extra MOSFET available separately, ZT-2450 • 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5225	$24.95	$22.35
OPTIONAL MOSFET TO SUIT	ZT-2450	$9.95 ea	
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

SPEEDO CORRECTOR MKII

Refer Silicon Chip Magazine December 2006
When you modify your gearbox, diff ratio or change to a large circumference tyre, it may result in an inaccurate speedometer. This project alters the speedometer signal up or down from 0% to 99% of the original signal. With this improved model, the input setup selection can be automatically selected and it also features an LED indicator to show when the input signal is being received.

- PCB, programmed micro and components
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5435	$54.95	$49.45

DIGITAL SPEEDOMETER/ SPEED ALERT

Refer Silicon Chip Magazine November/December 1999
A combined speedometer / speed alert that fits neatly into a small jiffy box (83 x 54 x 31mm) for mounting on your dashboard. This microprocessor based kit features a 3-digit LED display, 0-255km/h range with visual and audible alert & actual speed readout.

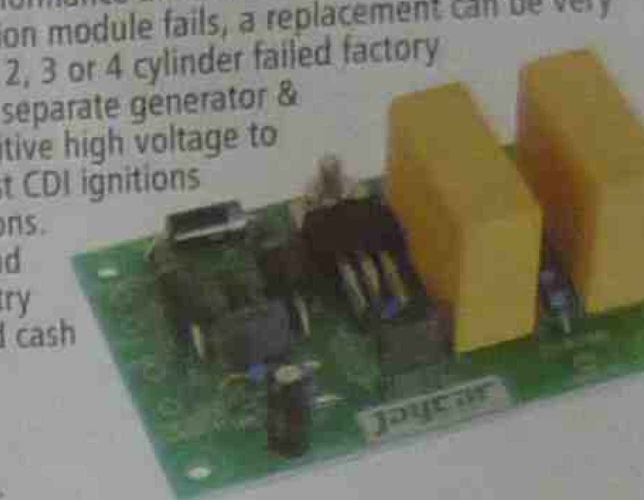
- PCBs, black jiffy box with laser cut & screen printed front panel, red acrylic display panel, pre-wound speed sensor bobbin, hookup wire, programmed microprocessor, magnets & all electronic components. Simple aluminum bracket required for speed sensor mounting.
- Beware of inferior kits which do not supply pre-wound bobbin.
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5279	$67.95	$60.95

CAPACITOR DISCHARGE IGNITION KIT FOR MOTOR BIKES

new

Refer Silicon Chip Magazine May 2008
Many small petrol engines fitted to modern motor bikes use a capacitor discharge ignition (CDI) to improve performance and enhance reliability. The only downside of this is that if the CDI ignition module fails, a replacement can be very expensive. This kit will replace most 1, 2, 3 or 4 cylinder failed factory units and is designed for engines with separate generator & trigger coils and which generate a positive high voltage to charge the capacitor before firing. Most CDI ignitions operate this way, but there are variations. Luckily, this CDI module uses cheap and readily available parts and is worth a try before shelling out lots of hard-earned cash for a genuine replacement module.



- PCB, case and components.
- Some mounting hardware required.

	Cat.	Qty 1+	Qty 5+
	KC-5466	$21.95	$19.70

PROGRAMMABLE HIGH ENERGY IGNITION SYSTEM

Refer Silicon Chip Magazine March, April 2007
This is Silicon Chip's most advanced and versatile ignition system yet and can be used on both two & four stroke engines. The system can be used simply to intercept and modify the factory ignition timing or turned into a stand alone ignition system with remapped timing, electronic coil control and anti-knock sensing. The unit will trigger from a range of sources including points, Hall effect sensors, optical sensors, or the 5 volt signal from the car's ECU. Timing can be mapped against engine load and RPM and adjusted in step as small as 0.5°.

- Timing retard & advance over a wide range
- Suitable for single coil systems
- Dwell adjustment
- Single or dual mapping ranges
- Two de-bounce settings
- Max & min RPM adjustment
- Suits 1 to 12 cylinder four stroke engines and 1 to 6 cylinder two stroke engines
- Optional knock sensing available
- PCB, programmed micro, components & die cast box with screen printed lid
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5442	$94.95	$85.50

IGNITION COIL DRIVER

Refer Silicon Chip Magazine March, April 2007
Add this ignition coil driver to the KC-5442 Programmable Ignition System and you have a complete stand-alone ignition system that will trigger from a range of sources including points, Hall effect sensors, optical sensors, or the 5 volt signal from the car's ECU. The kit can be mounted close to the coil to minimise energy loss and the ignition system can be fitted inside the car.

- PCB, case and components.

	Cat.	Qty 1+	Qty 5+
	KC-5443	$46.95	$42.45

KNOCK SENSOR

Refer Silicon Chip Magazine May 2007
Add this option to your KC-5442 Programmable High Energy Ignition system and the unit will automatically retard the ignition timing if knocking is detected. Ideal for high performance cars running high octane fuel. Requires a knock sensor that is cheaply available from most wreckers.

- PCB and components.

	Cat.	Qty 1+	Qty 5+
	KC-5444	$18.95	$16.95

KIT CONSTRUCTORS MANUAL

A must for amateur constructors. Contains much useful information for the more experienced. Huge amounts of information on construction and identification of parts.

- 8 pages.

related subject book

MADE IN AUSTRALIA

	Cat.	Qty 1+
	BI-8200	$2.00

Digital Tachometer Display

Refer Silicon Chip Magazine April 2000.

This PIC based tachometer is housed neatly in a small jiffy box (83 x 54 x 31mm), which mounts nicely on your dashboard. Its amazing features include 4 digit LED display showing up to 9,900rpm in 100rpm increments, 10 LED bargraph with optional dot or bar mode (showing 8 independent rpm thresholds), calibration options for 1 - 12 cylinder 4-stroke or 1 - 6 cylinder 2-stroke engines, anti-display flickering feature and automatic night time display dimming.

- PCBs, black case with laser cut & screen printed front panel, programmed micro, 7 segment displays, bargraphs red acrylic display panel, hook-up wire and components.
- 12VDC

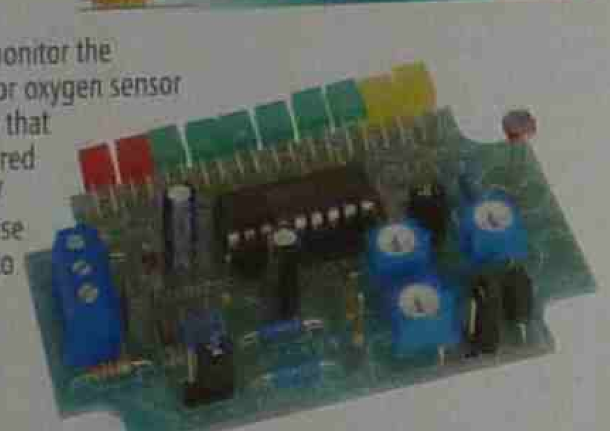
	Cat.	Qty 1+	Qty 5+
	KC-5290	$67.95	$60.95

Voltage Monitor

Refer Silicon Chip Magazine May 2006

This versatile kit will allow you to monitor the battery voltage , the airflow meter or oxygen sensor in your car. The kit features 10 LEDs that illuminate in response to the measured voltage, preset 9-16V, 0-5V or 0-1V ranges, complete with a fast response time, high input impedance and auto dimming for night time driving.

- PCB and components.
- 12VDC



	Cat.	Qty 1+	Qty 5+
	KC-5424	$22.95	$20.35
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Car Battery Monitor

Refer Electronics Australia May 1987

This simple electronic voltmeter lets you monitor the condition of your car's battery so you can act before getting stranded.

- 10 rectangular LEDs tell you your battery's condition.
- PCB and components.
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KA-1683	$17.95	$15.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Add-On Regulator For 12V Battery Chargers

Refer Electronics Australia July, 1997

Perhaps the most popular car battery chargers on the market today are readily available from supermarkets, but have substantial design limitations - usually reflected in their low price. When using one of these low-cost car battery chargers, you have to monitor the charge state of the battery and disconnect it from the battery when it's fully charged. Timer switches would be alright, but that doesn't take into account the charge level of the battery. A better (and safer) way is this add-on regulator kit so the charger can be left connected to the battery continuously. The PCB has been specifically designed to slide into an Arlec-brand 4A charger, although it could of course be mounted externally in its own separate case.

- PCB, heatsink sheet and components.
- Suits our MB-3528 battery charger.

	Cat.	Qty 1+	Qty 5+
	KA-1795	$27.95	$24.95

Screecher Car Alarm MKII - 12 Volt

Refer Electronics Australia January 1999

Featuring high deterrent capability for a minimum outlay, the Screecher MK II is more effective than its extremely popular predecessor from 1986. Using an ear-piercing piezo siren, this alarm will scare the pants off any would be thief. Unlike conventional car alarms, this unique design calls for the siren to be mounted inside the vehicle, the 'shrill' making it impossible for a thief to persist in stealing the vehicle or its contents. This project is easy to construct and features entry delay with a soft warning tone, exit delay and high intensity deterrent LED.

- PCB, siren, components and 2 adhesive warning stickers
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KA-1813	$29.95	$26.90
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

Ultrasonic Proximity Detector

Kwik Kit

Attention motorists!

Commercial products that carry out a similar task are, in some cases, worth over $100. Choose the minimum distance you wish to maintain between your vehicle and objects nearby. Useful at a distance of 10 to 80cm, the kit can also be used to sense animals straying into a particular area or even specific object security protection such as TVs and VCRs. An LED illuminates once an object strays into the detector's sensor range.

- Kit includes Kwik Kit PCB, one pair of ultrasonic sensors plus electronic components. This kit is designed to be used in a garage, not under the bumper bar.
- Sensors not waterproof.
- Suitable waterproof transducers/sensors Cat AU-5550.

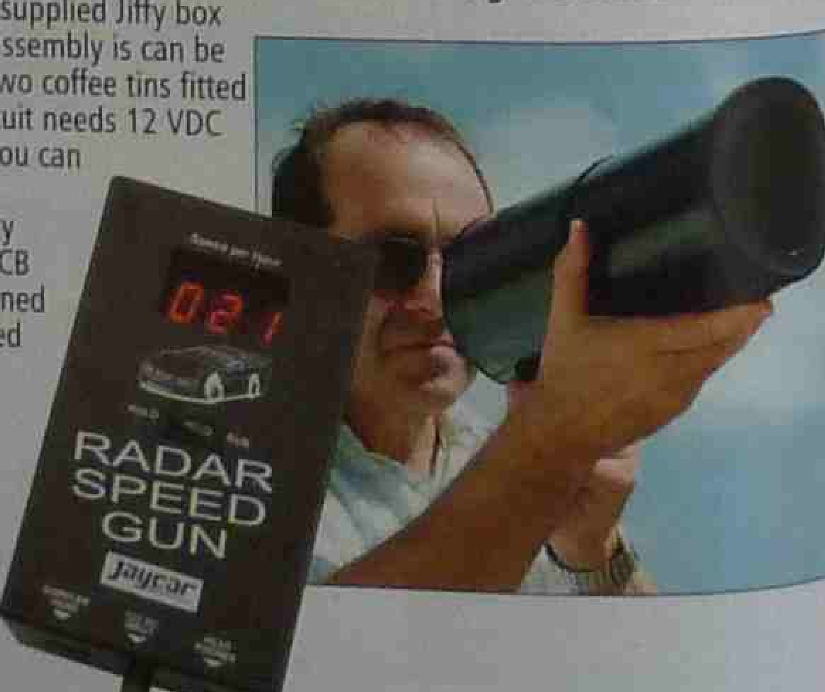
	Cat.	Qty 1+	Qty 5+
	KG-9158	$29.95	$26.90
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Radar Speed Gun Mk2 - New Revised Model!

Refer Silicon Chip Magazine Nov-Dec 2006

We have improved and refined this kit since it was published and it is now even more stable and accurate. If you're into any kind of racing like cars, bikes boats or even the horses, this kit is for you. This Doppler radar gun is similar to the units used by the police but at a fraction of the cost. It reads speed in km/h or mph up to 250 km/h or 15[illegible] mph. The resolution and accuracy we have obtained from this well priced kit is fabulous and it includes a hold switch so you can freeze the reading. The electronics & display are mounted in the supplied Jiffy box and the radar gun assembly is can be made simply with two coffee tins fitted end to end. The circuit needs 12 VDC at only 150mA so you can use a small SLA or rechargeable battery pack. Kit includes PCB with overlay, machined case and all specified components

- Some SMD components with red perspex panel.

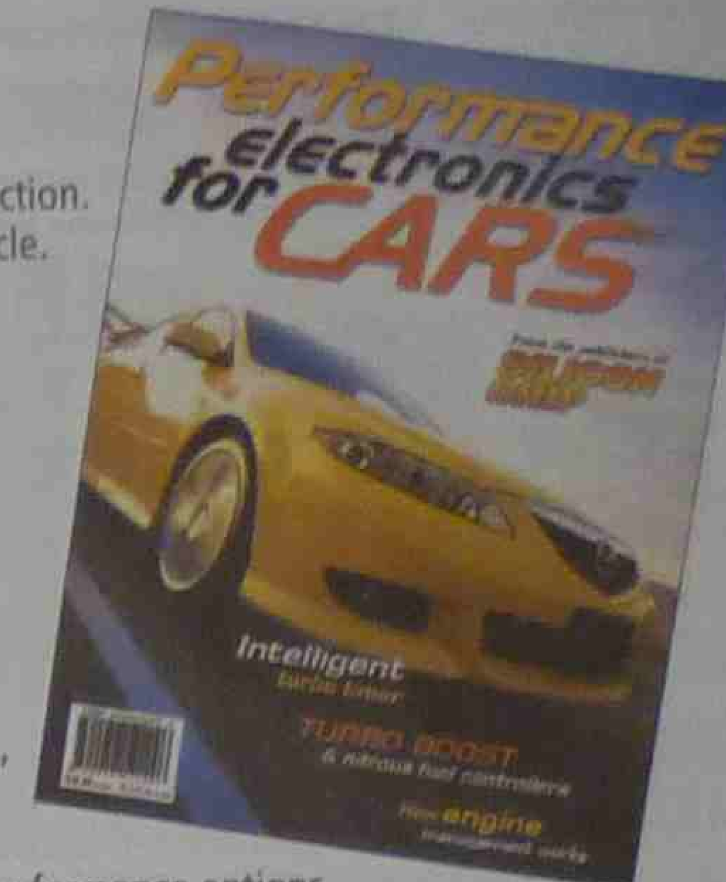


	Cat.	Qty 1+	Qty 5+
	KC-5441	$99.95	$89.95

Attention All Car Enthusiasts - Build Your Own 'Black Boxes'

Automotive High Performance - Just Build & Install

Without a doubt this book and the kits described in it have been very popular since their introduction. An absolute gold mine for DIY Car Enthusiasts who want to tweak the performance of their vehicle. The kits allow you to revamp fuel systems after injector upgrade, control turbo boost curves, or simply monitor exhaust gas temperatures. Although some of the projects use sophisticated electronics, both hardware and software, you can build the kits with relatively little electronics knowledge. You need to be able to solder and identify components but even if your skills in this area are a bit rusty, this book has a whole chapter set aside for you to have a refresher.



Performance Electronics for Cars Book by Silicon Chip Publications

Grab yourself a copy of the Performance Electronics for Cars project book, written by the Silicon Chip Publications team. The 160-page book describes the 16 projects shown on this and the following 2 pages in full detail. Each project includes full construction details, coloured overlays, circuit diagrams, testing, fitting and setup instructions, and even a 'How it Works' explanation. Even though each project will be supplied with a set of black & white instructions, the book is definitely a must-have to help ensure construction success and keep an overall perspective of performance options.

Chapters include:

- Understanding Engine Management
- Other Electronic Systems
- DIY Electronic Modification
- Using a Multimeter
- Coloured overays & circuit diagrams
- Over 23 chapters of projects and technical information
- Advanced Engine Management
- Modifying Car Electronic Systems
- Building Electronic Project Kits
- Full construction details
- Testing, fitting and setup instructions
- 'How it Works' explanation
- 160 page - full colour

SOME OF THE PRODUCTS ARE NOT COMMERCIALLY AVAILABLE OUTSIDE CAR FACTORIES!!

Cat.	Qty 1+
BS-5080	$22.50

Digital Pulse Adjuster

Take control over any pulsed solenoids in your car

Suggested uses:

- Create a custom boost curve
- Give better shift firmness in late model transmissions
- Modify action of the idle speed control valve to alter idle speed
- Control an extra fuel injector, water injector or toluene injector
- Modify steering control for better weight on speed-controlled systems

Refer: Performance Electronics for Cars - Silicon Chip Magazine Publications.

This unit is another huge step up in DIY automotive performance upgrades. It allows you to control and tune the operation of any solenoid that is run by the engine management system. This means that you could control turbo boost without an expensive boost controller, or alter automatic transmission line pressures for better shifts. Alternatively, it can be used to drive and control an extra fuel injector. Additional fuel injectors are commonly tapped into the throttle body to stop critical lean out under high load, high boost conditions. You can now add the injector with +/- 127 step mapping at 128 different duty cycle points. This allows superior fuel control, previously achieved only with an expensive fuel management system.

- PCB, machined case with processed panels, 2 programmed micros and components.
- Requires the Hand-held Digital Controller - KC-5386 and connecting cable WC-7502 (25 pin extension cable with all pins connected).

"The Digital Pulse Adjuster literally redefines the way in which car modifications can now be made." - Julian Edgar, Silicon Chip Magazine.

	Cat.	Qty 1+	Qty 5+
	KC-5384	$79.95	$71.95

Digital Fuel Adjuster

- Modify air/fuel ratios
- Allow air-flow meter or injector swaps
- Change closed-loop running characteristics
- Overcome Boost cuts

Refer: Performance Electronics for Cars - Silicon Chip Magazine Publications.

This unit is a huge revolution in DIY automotive performance. It gives you the power to completely tune the air/fuel ratio throughout the entire load range, at 128 load points, providing incredible mapping resolution and brilliant drivability. It uses the Handheld Digital Controller - KC-5386, so there is no need for a laptop, and it supports both static and real-time mapping. It can be used on 0-5V and 0-12V signals, so it is compatible with all voltage output airflow meters and MAP sensors. It can also be set to work with 0-1V signals, allowing modification of EGO sensor signals. This unit has been extensively tested on a wide range of cars including Subaru Impreza WRX & STi, Nissan 200SX, BMW 735i, Lexus LS400, & Nissan Maxima.

- PCB, machined case with processed panels, programmed micro and electronic components.

Kit requires the Handheld Digital Controller - KC-5386 and connecting cable WC-7502 (25 pin RS 232 extension cable with all pins connected).

"It is absolutely no exaggeration to say that the release of the Digital Fuel Adjuster is going to cause a revolution in budget engine management modification." Julian Edgar, Silicon Chip Magazine.

	Cat.	Qty 1+	Qty 5+
	KC-5385	$84.95	$76.50

Hand Controller for Digital Adjusters

- Used to program the Digital Pulse Adjuster, Digital Fuel Adjuster and Independent Electronic Boost Controller circuits

Refer: Performance Electronics for Cars - Silicon Chip Magazine Publications.

This hand controller is used for the mapping/programming of both Digital Adjuster kits. It features a two line LCD, and easy to use pushbuttons. It can be used to program the adjusters then removed, or left permanently connected to display the adjuster's operation. It is designed as an interface and display, and is not required for general adjuster functions after they have been programmed. If you have built and installed multiple adjuster kits, you will only need one controller to program, unless you want permanent displays for each adjuster function.

- PCB silk screened and machined case, LCD, and all electronic components.
- Must have all D25 pins connected.

	Cat.	Qty 1+	Qty 5+
	KC-5386	$67.95	$60.95
D25 MALE TO D25 MALE CABLE - 1.8M	WC-7502	$13.95	

Universal Voltage Switch

Economy Adjustable Temperature Switch

Frequency Switch

High Range Adjustable Temperature Switch with LCD

High Performance Timer

Smart Fuel Mixture Display for Fuel Injected Cars

Independent Electronic Boost Controller

Electronic Projects for Cars Volume 1

Audio Kits

Theremin Synthesiser Kit MkII

Theremin Synthesiser Kit

Studio 350 - High Power Amplifier

Why our version of this kit is so special

115W RMS/8 Ohm 170WRMS/4 Ohms Power Amp Module

135WRMS Ultra-Low Distortion Amplifier Module

new

Refer Silicon Chip Magazine August 2008

This ultra low distortion amplifier module uses the new ThermalTrak power transistors and is largely based on the high-performance Class-A amplifier which was featured in SILICON CHIP during 2007. The ThermalTrak transistors are a new version of the premium MJL3281A & MJL1302A and have an integral diode for bias compensation. As a result, the circuit has no need for a quiescent current adjustment or a Vbe multiplier transistor and has an exceptionally low distortion figure.

- PCB and components.
- Requires heat sink & power supply (KC-5471)

Output power: 135WRMS into 8 ohms and 200WRMS into 4 ohms
Frequency response at 1W: 4Hz to 50kHz
Input sensitivity: 1.26V RMS
Input impedance: 12k ohms
Harmonic distortion: <.008% from 20Hz to 20kHz
Signal-to-noise ratio: 122dB unweighted (22Hz to 22kHz)
Damping factor: <170 at 100Hz
Stability: Unconditional

	Cat.	Qty 1+	Qty 5+
	KC-5470	$94.95	$85.50

Power Supply Kit for 135W Ultra-LD MkII Amplifier (KC-5470)

Refer Silicon Chip Magazine September 2008

new

The amp module (KC-5470) is powered using an unregulated rail only. This power supply kit is specifically designed to provide a balanced +/- 55VDC supply to power this fantastic amp kit. It has two LEDs, which illuminate when power is present on the rails and the assist in slowly discharging the filter caps when power is switched off. It also provides another +/- 15V supply rail for optional powering of a preamplifier. Finally there is a +20V rail to power a speaker protector project (KC-5450).

- Supplied in short form with PCB and components.
- Requires a centre tapped transformer with 40V+40V windings for the +/-55V rails and 15V+15V windings for the auxilaries.

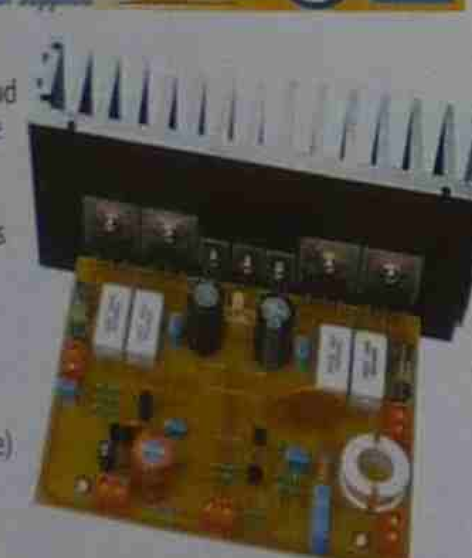
	Cat.	Qty 1+	Qty 5+
	KC-5471	$55.95	$50.50

50 Watt Amplifier Module SC-480 1 TO-218 Version

Refer Silicon Chip Magazine January and February 2003

Heatsink not supplied

You may remember the ETI-480 amplifier kits, introduced in December 1976. Silicon Chip have re-designed it, utilising its good features, to create the new SC-480. Some of the new features include less distortion (<0.05% 20Hz - 20kHz), inbuilt polyswitch cutout protection, and far better Signal to Noise figures than its predecessor. Power output is 50WRMS into 8 Ohms and 70WRMS into 4 Ohms.



- Short form with PCB and components.
- +/-40V (KC-5347, $17.95)
- Requires Heatsink, use HH-8555 (diecast type)

	Cat.	Qty 1+	Qty 5+
	KC-5345	$49.95	$44.90
HEATSINK TO SUIT	HH-8555	$19.95 ea	

Power Supply to suit SC-480

Refer Silicon Chip Magazine January 2003

Supplies +/-40V rails to power the SC-480 amplifier module kits. It also provides another +/-15V supply power to a preamplifier or equalizer etc.

- PCB and components
- +/-28V - (use MM-1095, 56V centre tapped transformer)

	Cat.	Qty 1+	Qty 5+
	KC-5347	$17.95	$15.95
TRANSFORMER TO SUIT	MM-1095	$42.95 ea	

50 Watt Amplifier Module

Refer Silicon Chip Magazine March 1994

This 50 watt unit uses a single chip module and provides 50 watts RMS into 8 ohms with very low distortion and extreme quietness.

- PCB size only 84 x 58mm.
- PCB and components
- Requires Heatsink

Specifications:
Output power: 50WRMS continuous into 8Ω
Power supply: +/- 37.5VDC
Signal/Noise ratio: -114dB (A-weighted)
-106dB (20-20kHz)
Distortion: 0.002% at 40W 1kHz
IHF power output: 60W @ 1kHz

Power Requirements:
- +/-35VDC rails @ 80VA per module
- MT-2114 25 + 25V 160VA toroid x 1
- ZR-1314 Bridge rectifier x 1
- RE-6241 2,200/50V electro caps x 2

	Cat.	Qty 1+	Qty 5+
	KC-5150	$29.95	$26.90
HEATSINK TO SUIT	HH-8590	$19.95 ea	

1 Watt Audio Amplifier Module

Kwik Kit

This tiny PCB measuring only 28 x 28mm incorporates all the electronics for a 1 Watt amplifier. 4 to 12VDC operating voltage, 50mV input sensitivity, 8 ohm output and 20 to 25000Hz output frequency. Optional connection for a 10k ohm - 100k ohm logarithmic potentiometer (not included) for adjustable volume control.

- PCB and components.
- 9VDC (use MP-3146)

	Cat.	Qty 1+	Qty 5+
	KG-9032	$9.95	$8.75
9VDC PLUGPACK TO SUIT	MP-3146	$17.95 ea	

"The Champ" 0.5 Watt Audio Amplifier

Refer Silicon Chip Magazine February1994

This tiny module uses the LM386 audio IC, and will deliver 0.5W into 8 ohms from a 9 volt supply making it ideal for all those basic audio projects. It features variable gain, will happily run from 4-12VDC and is smaller than a 9 volt battery, allowing it to fit into the tightest of spaces.

- PCB and components
- 4-12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5152	$7.95	$6.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

"Pre-Champ" Versatile Preamplifier

Refer Silicon Chip Magazine July 1994

This tiny preamp was specifically designed to be used with the "Champ" amplifier KC-5152. Unless you have a signal of sufficient amplitude the "Champ" will not produce its maximum power output. The "Pre-Champ" is the answer with a gain in excess of 40dB, which is more than enough for most applications. You can vary the gain by changing a resistor and there is even provision on the PCB for an electret microphone. (Use AM-4010)

- PCB and components.
- 6-12VDC.
- Can be battery powered.

	Cat.	Qty 1+	Qty 5+
	KC-5166	$8.95	$7.95
ELECTRET MICROPHONE TO SUIT	AM-4010	$2.65 ea	
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Bridge Mode Adaptor for Stereo Amplifiers

new

Refer Silicon Chip Magazine July 2008

This excellent kit will let you run a stereo amplifier in 'Bridged Mode' to effectively double the power available to drive a single speaker. There are no modifications required on the amplifier and the signal processing is done by the kit before the signals are fed to the stereo amp. The kit is perfect for say, using a stereo amplifier as an occasional PA amplifier for social functions or using an old amplifier to drive a sub-woofer in a home theatre system. It sounds like magic, but is just a clever adaptation of basic electronic principles.

- PCB and components
- +/- 15-60VDC

	Cat.	Qty 1+	Qty 5+
	KC-5469	$27.95	$24.95

Stereo Headphone Distribution Amplifier

Refer Silicon Chip Magazine November 2005.

Enables you to drive up to two stereo headphones from any line level (1volt peak to peak) input. The circuit features a facility to drive headphones with impedances from about 8-600 ohms.

- PCB and components
- PCB size: 134 x 103mm
- +/- 15VDC (use KC-5418)

	Cat.	Qty 1+	Qty 5+
	KC-5417	$34.95	$31.45
RECOMMENDED BOX - UB2	HB-6012	$6.95 ea	

Headphone Amplifier Power Supply For (KC-5417)

Refer Silicon Chip Magazine October 2005.

To ensure the best possible performance to the Headphone Amplifier Kit, this will provide regulated +/- 15V and +5 outputs.

- PCB and components
- PCB size: 56.5 x 80mm
- 15-0-15VAC 20VA (use MT-2086)

	Cat.	Qty 1+	Qty 5+
	KC-5418	$19.95	$17.70
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

Universal Stereo Preamplifier

- Perfect for phono, tape or microphones.

Refer Silicon Chip Magazine April 1994

Based around the low noise LM833 dual op-amp IC, this preamp is designed for use with a magnetic cartridge, cassette deck or dynamic microphone. The performance of this design is far better than most preamps in many stereo amplifiers, making it a worthy replacement if your current preamp falls short of expectation. It features RIAA/IEC equalisation, and is supplied with all components to build either the phono, tape or microphone version.

- Measuring only 80 x 78 x 30mm, it is ideal for incorporating into existing equipment
- PCB and components
- +/- 15VDC
- Use Cat KC-5038 and Cat MM-2007 if power is not available in your equipment.

	Cat.	Qty 1+	Qty 5+
	KC-5159	$16.95	$14.95
POWER SUPPLY KIT TO SUIT	KC-5038	$14.95 ea	
TRANSFORMER TO SUIT	MM-2007	$8.95 ea	
RECOMMENDED BOX - UB1	HB-6011	$4.45 ea	

Stereo VU and Peak Meter Kit

Refer Silicon Chip Magazine May 2007

Accurately monitor audio signals to prevent signal clipping and ensure optimum recording levels. This unit is very responsive and uses two 16-segment bargraphs to display signal levels and transient peaks in real time. There are a number of display options to select, and both the signal threshold and signal level calibration for each segment are adjustable.

- Accuracy within 1dB for signals above -40dB.
- PCBs, LCD, programmed micro and components.
- 9-15VDC 110mA (use MP-3147)
- Case not included (use HB-6082)

	Cat.	Qty 1+	Qty 5+
	KC-5447	$69.95	$62.95
REQUIRED POWER SUPPLY	MP-3147	$17.95 ea	
RECOMMENDED BOX - UB2	HB-6082	$11.95 ea	

Balanced to Unbalanced Audio Converter

new

Refer Silicon Chip Magazine June 2008

Using domestic audio equipment in a professional environment is complicated by the fact that standard audio gear does not have the balanced inputs and outputs found in professional systems. This kit overcomes the problem and will adapt an unbalanced input to balanced output and vice versa. This allows domestic equipment to be integrated into a professional installation while maintaining the inherent high immunity to noise pick-up on long cable runs provided by balanced lines.

- PCB and Components
- +/- 9-15VDC or 9-30VDC or 7-12VAC

	Cat.	Qty 1+	Qty 5+
	KC-5468	$32.95	$29.45

Bass, Treble & Volume Controller

Kwik Kit

This module enables you to add volume and tone control to any line level (IV) amp. The circuit consists of an input buffer, a Baxandall-type bass-treble network and output voltage divider type level control. It will give up to line level out.

- Mono - Two required for stereo.

	Cat.	Qty 1+	Qty 5+
	KG-9004	$16.95	$14.95
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

Parabolic Microphone

Kwik Kit

- Hear noises or conversations from a distance.

Whether you're James Bond or just an avid animal observer, this kit will allow you to hear noises from a few hundred metres away. Correctly identifying sounds over that distance is near impossible with normal, unaided hearing. With the use of a reflector (a plastic toy ball, for example), the microphone will detect the faintest sounds - then amplify them.

- Powered from 9VDC with an adjustable output for a small speaker or headphones.
- PCB dimensions: 55 x 44mm.
- Kwik Kit PCB, microphone and components.

A suitable reflector must be acquired, although it does not necessarily need to be parabolic. Most 'bargain basement' shops have inexpensive plastic balls that would be quite suitable.

	Cat.	Qty 1+	Qty 5+
	KG-9024	$24.95	$22.35

related products: SEE OUR RANGE OF 433MHZ & 2.4GHZ WIRELESS MODULES ON PAGES 111 & 112

Subwoofer Controller

Refer Silicon Chip Magazine August 2007

Refer Silicon Chip August 2007

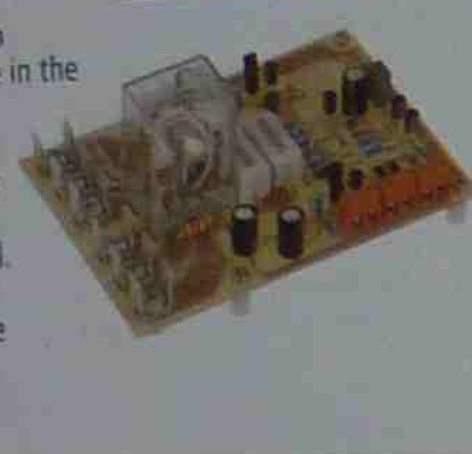
Adding a subwoofer to your home theatre or Hi-Fi or car sound system is the easiest way to extend its bass response. Using this kit to control your external speaker and sub-amplifier can give you loads of bass without taking up much space. The kit has all the features you could want, including low and high pass filters, parametric equaliser and auto-turn on for external equipment. The controller is 12VDC powered and can also be used in automotive applications.

- PCB, case with processed panels and components

Cat.	Qty 1+	Qty 5+
KC-5452	$99.95	$89.95

Universal Speaker Protection & Muting Module Kit

Refer Silicon Chip Magazine July 2007



The primary function of this versatile project is to protect your expensive speakers against damage in the event of catastrophic amplifier failure such as a shorted output transistor. In addition, the circuit also banishes those annoying thumps that occur when many amplifiers are switched on or off, especially when the volume is set to a high level. The design also incorporates an optional over temperature heat-sensor that will disconnect the speakers if the output stage gets too hot.

- PCB, 24VDC relay and components.
- 22-70VDC

Cat.	Qty 1+	Qty 5+
KC-5450	$34.95	$31.45

"Minivox" Voice Operated Relay

- Communicate "Hands Free".

Refer Silicon Chip Magazine September 1994

Voice operated relays are often used in "hands free" communication such as mobile radios, amateur tranceivers as well as some PA applications. The idea, of course is that instead of pushing a button on the microphone to talk, the sound of the voice activates it instead. This tiny kit measures approx 50 x 50 x 15mm and is designed to fit into the tightest spaces, has almost no turn-on delay, and has 3 second release time.

- PCB, electret mic, and components.
- 12VDC 35mA

	Cat.	Qty 1+	Qty 5+
	KC-5172	$15.95	$14.35
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Magnetic Cartridge Preamp

Refer Silicon Chip Magazine August 2006

This kit is used to amplify the 3-4mV signals from a phono cartridge to line level, so you can use your turntable with the CD or tuner inputs on your Hi-Fi amplifier - most modern amps don't include a phono input any more. Dust off the old LP collection or use it to record your LPs onto CD. The design is suitable for 12" LPs, and also allows for RIAA equalisation of all the really old 78s.

- S/N ratio: -84dB
- THD at 1kHz: 0.014%
- RIAA accuracy: within 1dB from 20Hz to 20kHz

Please note that the input sensitivity of this design means it's only suitable for moving-magnet, not moving coil cartridges.

- PCB and components.
- 12VAC
- Recommended enclosure HB-5042.

	Cat.	Qty 1+	Qty 5+
	KC-5433	$39.95	$35.90
RECOMMENDED BOX	HB-5042	$21.95 ea	

15dB Antenna Amplifier (Single Stage)

At last! A project for HAMS and anyone into RF. Featuring a strip line-type high-gain RF transistor, this kit will provide gain up to a respectable 15dB over the bandwidth of 50-1000MHz. This covers all TV, FM, marine and aircraft VHF as well as Police, mobile phone etc. Has saddle clamp in and out to mount coax to lessen VSWR losses. So stable is this amp you can put two in series for extra gain without the amp taking off. Must go in a screened box once built. Duh! *Screening must be added to the UB5 box • 6-18V.

	Cat.	Qty 1+	Qty 5+
	KG-9002	$24.95	$22.35
RECOMMENDED BOX - UB5*	HB-6015	$2.95 ea	

45 Second Voice Recorder Module

Refer Silicon Chip Magazine December 2007

Here's an improved version of the very popular voice recorder design published by Silicon Chip Magazine in May 2005. This improved version can now be set up easily to record two, four or eight different messages for random-access playback or a single message for 'tape mode' playback. Also, it now provides cleaner and glitch-free line-level audio output suitable for feeding an amplifier or PA system.

- PCB and components
- 9-12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5454	$43.95	$39.45
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

The Super Ear

- Hear those noises you normally never hear

Refer Electronics Australia May 1998

This kit assists people who have difficulty in hearing high audio frequencies, or for those who want to hear more than their normal unaided ear. By amplifying these high audio frequencies, not only will conversations be made clearer, you will be able to hear noises not normally heard such as insects or a watch ticking, for example. Built into a small case & powered from a 9V battery makes this kit totally portable. Use it as a hearing aid or for a fun & educational purpose.

- PCB case, processed panels, 9V battery, and components.
- Headphones required.

Note: Not a replacement for a proper hearing aid.

Cat.	Qty 1+	Qty 5+
KA-1809	$27.95	$24.95

Fog Horn

This kit generates a deep sounding noise similar to fog horns on ships. Use as a unique warning siren or to improve a child's toy.

- Output power up to 5 watts depending on the input voltage used.
- 4.5-12VDC
- Requires an 8ohm speaker - use our AS-3000.

	Cat.	Qty 1+	Qty 5+
	KG-9092	$12.95	$11.35
57MM SPEAKER TO SUIT	AS-3000	$3.50 ea	
RECOMMENDED BOX UB3	HB-6013	$3.95 ea	

Clifford the Cricket

Refer Silicon Chip Magazine December 1994

Clifford hides in the dark and chirps annoyingly until a light is turned on - just like a real cricket. Clifford is created on a small PCB, measuring just 40 x 35mm and has cute little LED insect eyes that flash as it sings. Just like a real cricket, it waits a few seconds after darkness until it begins chirping, and stops instantly when a light comes back on.

- PCB, piezo buzzer, LDR and components.
- 9VDC

Cat.	Qty 1+	Qty 5+
KC-5178	$16.95	$14.95

Radio & FM Kits

Video didn't kill the radio star. Rediscover the magic of radio through one of our three simple radio kits which take next to no time to assemble. But for something a little more challenging and infinitely more rewarding, pick up our popular Micromitter FM Transmitter. Hook it up to a CD or MP3 player and enjoy endless hours of listening to your favourite music or podcasts anywhere in your house through any humble FM radio.

Crystal Radio

- A radio without batteries!

Enjoy AM broadcasting without using battery or other power sources. Ideal for entry-level students or hobbyist with little electronics experience.

- Includes circuit explanation.
- Kit supplied with screen printed PCB (81x53mm), crystal, pre-wound coil, earphone and all components.

Cat.	Qty 1+	Qty 5+
KV-3540	$12.95	$11.35

Micromitter Stereo FM Transmitter

- Transmit you favourite music all over the house.

Refer Silicon Chip Magazine December 2002

We took the standard Silicon Chip design and improved it by replacing the original L-C filter components with a GFWB3 encapsulated 3 pin network to provide much better roll-off characteristics and reduce spurious interference outside the FM band. This is the third generation of this kit and is far more stable and compact than the original. You can connect your CD or MP3 player to the Micromitter and listen to your music all over the house through any FM radio. This model is also more stable than previous models and uses a surface mount BH1417F processor. It is crystal locked to a preselected frequency to eliminate frequency drift.

- Revised PCB, case with silk screened lid and components.
- 6VDC (4 x AAA) or 9-16VDC plug pack

Cat.	Qty 1+	Qty 5+
KC-5341	$49.95	$44.90

Miniature FM Transmitter

Refer ETI December 1985

This unit is a two transistor two stage transmitter that has the benefits of being VERY COMPACT.

- PCB with dual overlay, 9V battery and components.
- PC board size - 45 x 22mm.
- 9VDC

	Cat.	Qty 1+	Qty 5+
	KE-4711	$12.95	$11.35
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

100 - 200MHz VHF Converter

This simple to build kit makes it feasible to receive, for example, taxis, amateur radio operators, marine radio, television audio carriers, etc. The kit connects in-line with your VHF receiver's antenna avoiding messy installation and receiver modifications. Operating voltage 9V DC Sensitivity up to 0.8uV at 10dB S/N. Expand your knowledge of radio!

	Cat.	Qty 1+	Qty 5+
	KG-9128	$29.95	$26.90
RECOMMENDED BOX	HB-5970	$8.95 ea	

Video Kits

For a couple of dollars and a bit of sense, you can get so much more out of your home entertainment system. Extend the range of your remote so you can watch and operate your DVD or Foxtel from other rooms in the house. Or perhaps install a full function programmer for your wafer card. Either way our video kits will certainly enhance your home viewing pleasure.

IR Remote Extender MKII

Refer Silicon Chip Magazine October 2006

Operate your DVD player or digital decoder using its remote control from another room. It picks up the signal from the remote control and sends it via a 2-wire cable to an infrared LED located close to the device. This improved model features fast data transfer, capable of transmitting Foxtel digital remote control signals using the Pace 400 series decoder.

- PCB, case with screen printed panel and components
- 2-wire cable for extending the IR-Tx lead (use WB-1702).
- 9VDC 150mA

	Cat.	Qty 1+	Qty 5+
	KC-5432	$26.95	$23.95
9VDC PLUGPACK TO SUIT	MP-3146	$17.95 ea	
2-WIRE CABLE TO SUIT	WB-1702	$0.40 ea	

Smart Card Programmer Kit for Gold, Silver and Emerald Wafer Cards

Refer Silicon Chip Magazine July 2003

This full function programmer allows you to program both the microcontroller and EEPROM in the popular Gold, Silver and Emerald wafer cards. It hooks up to the serial port of your PC and can be operated as a free-standing unit or installed in a PC drive bay. Cards used need to conform to ISO-7816 standards, which includes ones sold by Jaycar.

- Instructions outline requirements for software that are available on the Internet.
- PCB, wafer card socket and electronic components.
- PCB measures 141 x 101mm
- 9-12VDC (battery or use MP-3130)

Jaycar Electronics and Silicon Chip Magazine will not accept responsibility for the operation of this device, its related software, or its potential to be used for unlawful purposes.

	Cat.	Qty 1+	Qty 5+
	KC-5361	$54.95	$49.45
REGULATED PLUGPACK TO SUIT	MP-3130	$29.95 ea	

Kit Constructors Manual

A must for amateur constructors. Contains much useful information for the more experienced. Huge amounts of information on construction and identification of parts.

- 8 pages.

Cat.	Qty 1+
BI-8200	$2.00

HOUSEHOLD PROJECTS

We're proud of our stable of clever DIY home projects. Last year's water tank level meter and telemetry base station met a growing need for many households and businesses. This year we introduce the Tempmaster MKII – our upgrade of the original electronic thermostat kit; as well as a Remote Switch Kit with a range of up to 200m. Just a couple of the many bright ideas we have for making your life at home just a little easier.

TEMPMASTER FRIDGE CONTROLLER MK II

new

Refer Silicon Chip Magazine February 2009

Want to convert an old chest freezer into an energy-efficient fridge or beer keg fridge? Or convert a spare standard fridge into a wine cooler? These are just two of the jobs this low-cost and easy-to-build electronic thermostat kit will do. It can also be used to control 12V fridges or freezers, as well as heaters in hatcheries and fish tanks. It controls the fridge/freezer or heater directly via their power cables, so there's no need to modify the internal wiring.

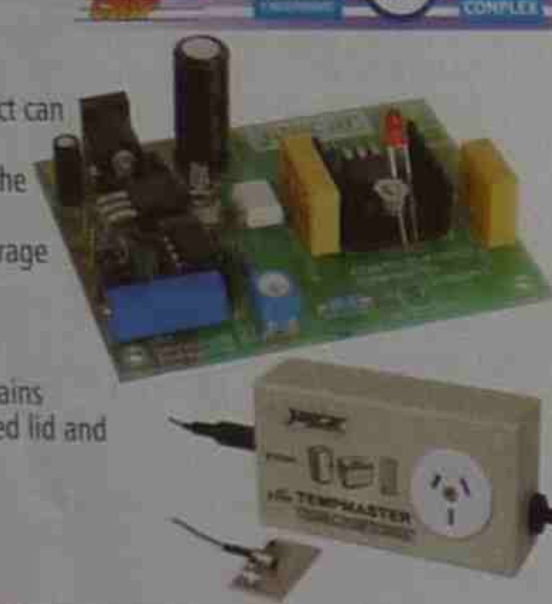
- Short-form kit contains revised PCB, sensor and components.
- You'll need to add your own 240V GPO, switched IEC socket and case.

Cat.	Qty 1+	Qty 5+
KC-5476	$32.95	$29.45

TEMPMASTER

Refer Silicon Chip Magazine June 2005

Need accurate temperature for a wine cooler or beer brewing heater? This project can be hardwired to turn a regular fridge or freezer into a wine cooler by controlling the temperature to make it suitable for wine storage. Or wire it for heating and encourage fermentation while brewing beer.



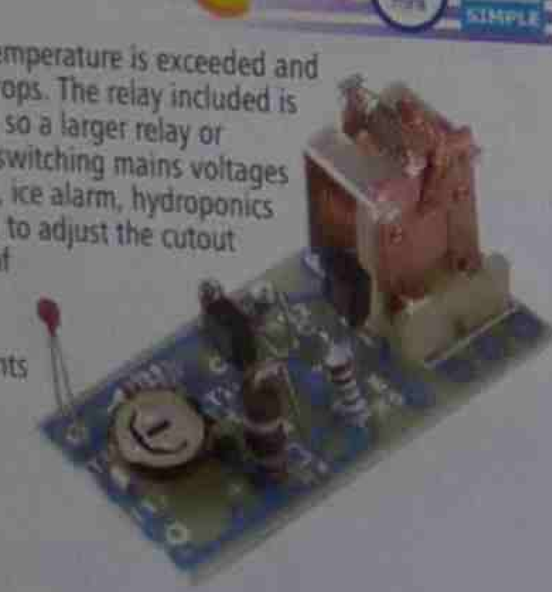
- Temperature range: 2.5 to 33°C.
- PCB, Panel mount mains socket and mains lead, machined case with screen printed lid and components.
- 12VDC, 15mA.

Cat.	Qty 1+	Qty 5+
KC-5413	$42.95	$38.45

TEMPERATURE SWITCH

- Ideal for turning fans or heaters on at certain temperatures.

This kit operates a relay when a preset temperature is exceeded and drops-out the relay when temperature drops. The relay included is capable of switching small currents only, so a larger relay or switching device should be employed if switching mains voltages or heavy currents. Ideal as a thermostat, ice alarm, hydroponics applications, etc. A small trimpot is used to adjust the cutout temperatures for the relay in the range of approx -30 to +150° celcius.



- PCB NTC thermocouple and components
- 12VDC, 100mA

	Cat.	Qty 1+	Qty 5+
	KG-9140	$24.95	$22.35
12VDC PLUGPACK TO SUIT	MP-3147	$17.95 ea	
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

433MHz REMOTE SWITCH KIT

Refer Silicon Chip Magazine January 2009

Suitable for remote control of practically anything up to a range of 200m, for example as a replacement for a dead garage door opener. The receiver has momentary or toggle output and the momentary period can be adjusted. The receiver can also be used to drive a 12V relay. Up to five receivers can be used in the same vicinity and additional transmitter kits are available.

A versatile kit with endless applications.
- 433MHz • TX PCB and RX PCB and components
- 9 -12VDC • Optional enclosures • Extra transmitter kit: KC-5474

Cat.	Qty 1+	Qty 5+
KC-5473	$44.95	$40.45
KC-5474	$22.95	$20.35

UHF REMOTE CONTROLLED MAINS SWITCH

Refer Silicon Chip Magazine February 2008

There are many situations where it would be convenient to switch on an appliance remotely rather than switching it on manually. These could include turning on pathway lighting when you arrive home late at night, controlling garden lighting, or pool lighting and pumps etc. Remote switching can also be very convenient for appliances that are difficult to access. Commercial remote control mains switches are available but these are generally limited to a range of less than 20m. This UHF system will operate over a range up to 200m and is perfect for water pump control systems etc. The switch can be activated using the included hand held controller or our KC-5461 water tank level sensor base station for automatic water tank level control.

- 2 PCBs, 2 cases, RF modules, 2 programmed micros and components.

Cat.	Qty 1+	Qty 5+
KC-5462	$99.95	$89.95

PIR CONTROLLED MAINS POWER SWITCH

Refer Silicon Chip Magazine February 2008

You've seen those lights fitted with PIR detectors that turn on when someone approaches. Well now you can do the same thing with just about any mains-powered device you like. You can use it to activate security systems, decorative lighting, fountain pumps or commercial advertising etc. Just about anything that can be plugged into a standard power point can be triggered by people moving in the vicinity of the passive infrared detector. The system uses a standard PIR to safely turn on 240VAC mains device(s) for an adjustable pre-set period. It's compact, easy to build, and will cost you much less than commercial PIR-triggered switches with similar features.

- Machines case with screen printed lid, PCB, mains sockets and components

Cat.	Qty 1+	Qty 5+
KC-5455	$79.95	$71.95

ELECTRONIC FLOW RATE METER WITH LCD

new

Completely self-contained, this unit only measures 60 x 40mm and includes a full LCD information display. It's supplied with a reed switch and piezo alarm. It operates from 2 x AAA batteries and a battery holder is included. When used in conjunction with the FS-300AH Digital Flowmeter (Cat no. ZD-1202), it will count down (in litres) from a predetermined volume, for example 500 litres. When 500 litres have gone through the flowmeter an alarm will sound. The alarm signal can be used to trigger another slave device. As it goes up to 99,500 litres, it could be used in larger-scale applications such as irrigation or just to let you know when it's time to clean a filter. It also reads flow rate, will remember data and operate in time mode. An unbelievably fantastic product! Full data sheet and instructions included.

- PCB/LCD size: 60(L) x 40(W)mm

Cat.	Qty 1+	Qty 10+	Qty 25+
ZD-1204	$69.95	$62.95	$48.90

"CLOCKWATCHERS" LED CLOCK

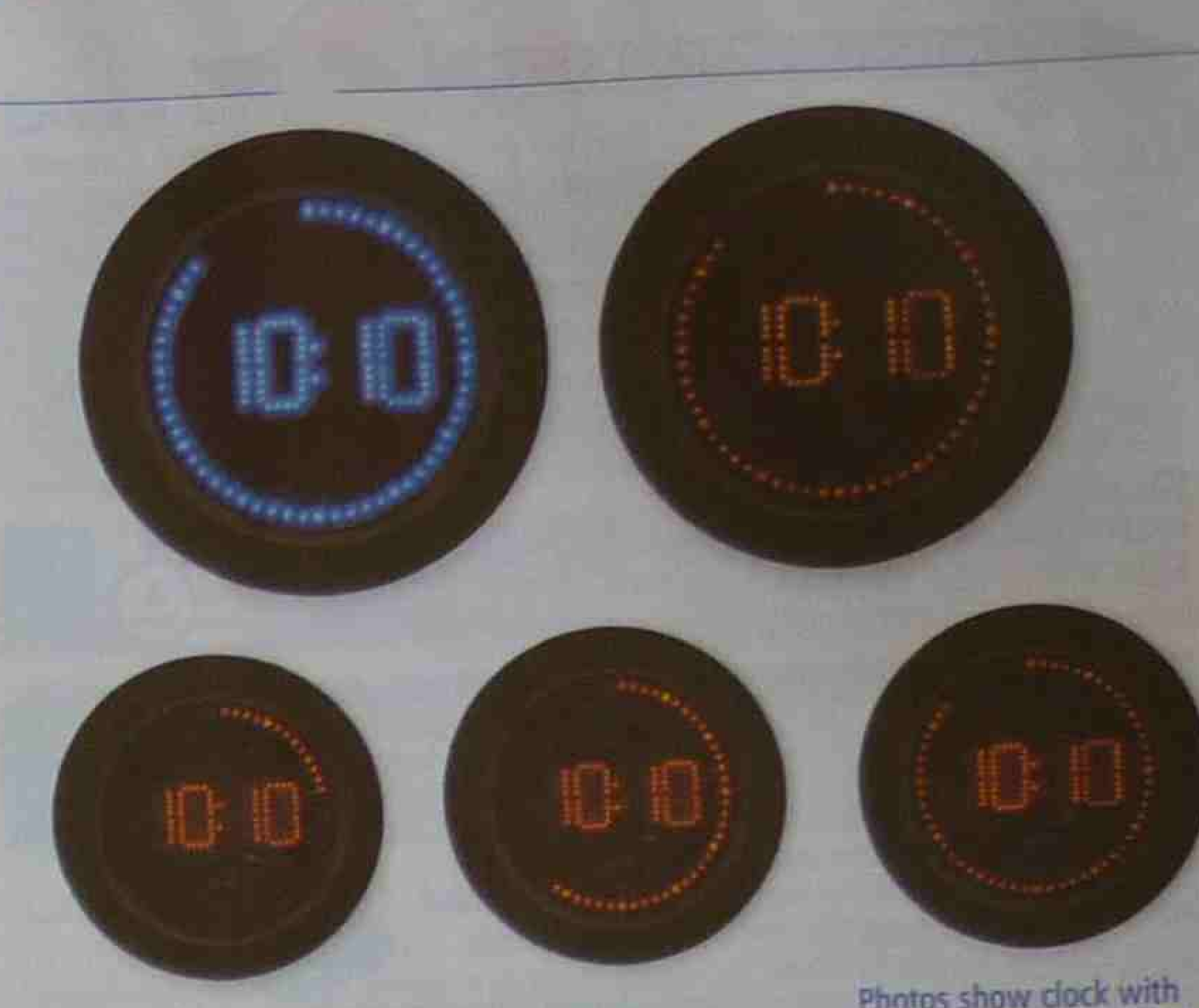
Refer Silicon Chip Magazine June 2005

It consists of a n AVR driven clock circuit, and produces a dazzling display with the 60 blue LEDs around the perimeter. It looks amazing, but can't be properly explained here. We have filmed it in action so you can see for yourself on our website, so check it out! Kit supplied with double sided silk screened plated through hole PCB and all board components as well as the special clock housing

WE HAVE FILMED THIS CLOCK IN ACTION SO YOU CAN SEE FOR YOURSELF ON OUR WEBSITE, SO CHECK IT OUT!

See page 419 for our built unit (Note: it's not quite the same).

	Cat.	Qty 1+	Qty 5+
RED	KC-5404	$129.00	$115.95
BLUE	KC-5416	$189.00	$169.95
12VAC POWER SUPPLY	MP-3020	$18.95 ea	



For a full dynamic demonstration see our website, keyword search on KC-5404 and then follow the prompts.

Photos show clock with red filter fitted (supplied).

FLEXITIMER/ INTERVAL TIMER

Refer Silicon Chip Magazine June 2008

Here's a new and completely updated version of a very popular project: an easily programmed low cost electronic timer. It operates from 12VDC, with low current drain: <50mA when relay is on, <5mA when relay is off. It is link programmed for either a single ON timing period, or continuous ON/OFF cycling and can be link programmed for any of 48 different time periods, separately for ON time and OFF time. Selectable periods are 1/2/3/4/5/6/7/8/10/20/30/40/50/60/70/80 seconds, minutes and hours. It may be restarted at any time simply by pressing reset pushbutton.
- PCB, programmed micro, relay and components. • 12VDC 50mA

Cat.	Qty 1+	Qty 5+
KC-5464	$34.95	$31.45

THE FLEXITIMER

- Handy timer from a few seconds to an entire day.

Refer Electronics Australia March 1991

This simple timer kit may be used as a parking meter reminder, or the basis of a full blown watering system. This kit uses just a handful of components to accurately time intervals from a few seconds to a whole day. It can switch a number of different output devices.

- Jaycar revised the circuit and PCB to allow selection of one shot or interval (50% duty cycle) mode via a jumper
- The time period is set with another jumper and increases / decreases by power of 2
- PCB and components!
- 12-15VDC

	Cat.	Qty 1+	Qty 5+
	KA-1732	$19.95	$17.70
12VDC PLUGPACK TO SUIT	MP-3147	$17.95 ea	
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

LIQUID LEVEL SENSOR

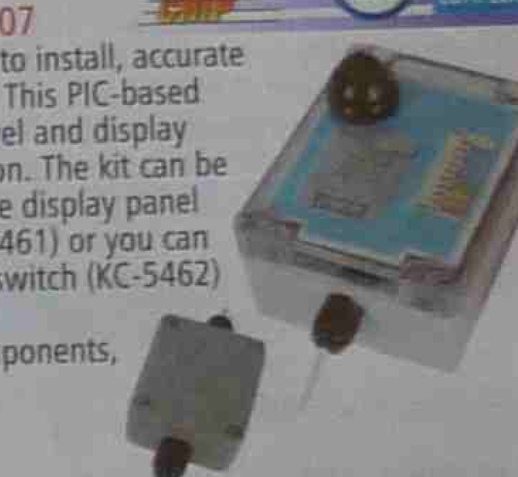
When two contacts are shorted by liquid, an LED will illuminate. Use in applications such as an overflow alarm and rain detector. Connect Relay Card KG-9142 for a relay output to operate lights, sirens or other warning devices.
- PCB size: 45 x 17mm.
- Kwik Kit PCB and components.
- 9VDC

	Cat.	Qty 1+	Qty 5+
	KG-9138	$9.95	$8.75
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

PIC BASED WATER TANK LEVEL METER KIT

Refer Silicon Chip Magazine November 2007

Looking for a water tank level meter that's easy to install, accurate and doesn't need a complicated in-tank sensor? This PIC-based unit uses a pressure sensor to monitor water level and display tank level via an RGB LED at the press of a button. The kit can be expanded to include an optional wireless remote display panel that can monitor up to ten separate tanks (KC-5461) or you can add a wireless remote controlled mains power switch (KC-5462) to control remote water pumps.
- Basic tank monitor kit includes electronic components, case, PCB and pressure sensor.
- Some fitting may be required to suit your particular tank.



Cat.	Qty 1+	Qty 5+
KC-5460	$109.00	$97.95

TELEMETRY BASE STATION FOR WATER TANK LEVEL METER

Refer Silicon Chip Magazine January 2008

The ability to monitor water tank levels from a remote location can be very useful, especially if you have several water tanks or if the tanks are hard to access. This Base Station is intended for use with the telemetry version of the KC-5460 water tank level meter and can handle data transmissions from up to 10 level meters and display the results on a 2-line 32-character LCD module. In bargraph mode, it can show up to 10 tank levels simultaneously, while the digital readout mode shows individual tank levels to 1%. As a bonus, it also includes an option for electric pump control.

- PCB, electronic components, receiver module and the RF transmitter upgrade for one tank level meter.
- Requires transmitter module to control KC-5462 remote switch.
- 9-12VDC 100mA.

Cat.	Qty 1+	Qty 5+
KC-5461	$84.95	$76.50

LED WATER LEVEL INDICATOR MKII KIT

Refer Silicon Chip Magazine July 2007

This clever circuit illuminates a string of LEDs to indicate the water level in a rainwater tank. The more LEDs that illuminate, the higher the water level inside the tank. The input signal is provided by ten sensors that are connected to the indicator unit via light duty figure-8 cable.
- Requires: 20mm PVC hose/pipe (length required depending on depth of tank)
- 12-18V AC or DC plugpack 500mA
- PCB case with machined and screen printed lid and components

Cat.	Qty 1+	Qty 5+
KC-5449	$39.95	$35.90

KITS

SECURITY PROJECTS

There's more to security than just deterring would-be intruders with wailing sirens and flashing lights. Our infrared security kits serve many household and commercial purposes, including detecting and alerting you of customers entering your shopfront, cars or vans entering your driveway, blockages in your assembly line and even whether or not your IR remote control is working.

ROLLING CODE INFRARED KEYLESS ENTRY SYSTEM

SMD

Refer Silicon Chip Magazine October 2007

Features two independent door strike outputs and will recognise up to 16 separate key fobs. The transmitter uses a fixed algorithm to calculate and transmit a different code each time the button is pressed. The receiver uses the same algorithm to compare the received code with the expected code and if they match, the door will open. The system incorporates an auto match facility that keeps the coded key fobs synchronised to the receiver and compensates for random button presses while the fobs are out of range.

- PCBs, two programmed micros, battery & electronic components.
- The receiver requires a 12VDC 1.5A power supply. Some SMD soldering is required.

	Cat.	Qty 1+	Qty 5+
	KC-5458	$64.95	$58.50

5 METRE IR LIGHT BEAM

With a range of about 5 metres, this kit will indicate using an LED when a person or object interrupts the infrared light beam. Use it across a doorway or across an assembly line. Connect a relay or use the Relay card KG-9142 for a relay output.

- PCB dimensions: 58 x 45mm.
- Kit supplied with solder masked PCB with overlay, infrared transmitter/receiver diodes, and all electronic components.
- 12VDC 100mA - (use plugpack MP-3011)

NEW IMPROVED MODEL	Cat.	Qty 1+	Qty 5+
	KG-9094	$14.95	$13.35
12VDC PLUGPACK TO SUIT	MP-3011	$19.95 ea	
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

50M IR LIGHT BARRIER

Covering a distance of up to 50 metres this light beam relay is ideal for protecting areas that have wide entrances, including driveways, shops, offices and storerooms, etc. Once the beam is broken the relay will trigger an onboard LED which will illuminate.

- TX 9VDC (use MP-3130) RX 12VDC (use MP-3011)
- PCB, infrared transmitter diodes, receiver transistor, magnifying lens and components.
- PCB: Receiver 56 x 45mm, Transmitter 17 x 55mm

	Cat.	Qty 1+	Qty 5+
	KG-9196	$49.95	$44.90
12VDC PLUGPACK TO SUIT RECEIVER	MP-3011	$19.95 ea	
9VDC PLUGPACK TO SUIT TRANSMITTER	MP-3130	$29.95 ea	

INFRARED FLOODLIGHT

- Let your CCD camera see in the dark!

This infrared light is powered from any 12-14VDC 300mA source and uses 32 x infrared LEDs to illuminate an area of up to 5-metres (will vary with light conditions). The PCB measures 56 x 75mm and draws a current of about 300mA.

- Screen printed/gold-plated/solder-masked PCB, 32 x infrared LEDs and all electronic components.
- Use our plugpack supply, MP-3147.

Note: Not suitable for CMOS cameras.

	Cat.	Qty 1+	Qty 5+
	KG-9068	$29.95	$26.90
RECOMMENDED BOX - UB3	HB-6013	$3.95	

INFRARED DETECTOR

This kit will switch on LED when it detects infrared light from sources such as IR remote controls. Connect it to the Relay Card kit KG-9142 to make an infrared remote controlled relay. Project requires 9VDC

- PCB dimensions: 55 x 15mm
- Can be battery powered.
- Kwik Kit PCB, Infrared receiver and all electronic components.
- Use to test if your remote control is working.

	Cat.	Qty 1+	Qty 5+
	KG-9086	$9.95	$8.75
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

LIGHTING PROJECTS

We offer several lighting projects which include smart solutions for your stage lighting, party lighting and outdoor home lighting needs. We even have a full-sized basketball electronic scoreboard – perfect for schools, sporting clubs and crazy b-ball fanatics who just need to practice those heart-stopping buzzer-beaters over and over again.

EMERGENCY 12V LIGHTING CONTROLLER

Refer Silicon Chip Magazine January 2008

This easy-to-build project is designed to automatically supply power for 12V emergency lighting during a blackout. The system has its own 7.5Ah SLA battery which is maintained via an external smart charger. The system includes a manual override switch and automatic protection to prevent over-discharge of the battery. Build your own emergency lighting system so you won't have to search for candles or your torch in the event of a blackout.

- PCB, case with screen printed panel and components
- Charger, SLA battery and lights available separately.

	Cat.	Qty 1+	Qty 5+
	KC-5456	$69.95	$62.95

12V LIGHT OPERATED RELAY

This kit can operate as a twilight on/off switch or as a light trigger relay. Operated from 12 volts, this versatile project triggers a 6 amp relay when the light intensity falls below an adjustable threshold. Turn lights on around the house when it goes dark or trigger an alarm when a light is switched on.

- Kwik Kit PCB, relay and all electronic components.
- 12VDC required - use plugpack MP-3147.

	Cat.	Qty 1+	Qty 5+
	KG-9090	$24.95	$22.35
12VDC PLUGPACK TO SUIT	MP-3147	$17.95 ea	

FLICKERING FLAME LIGHTING

- Great for the stage or unique lighting effects at home.

Refer Silicon Chip Magazine October, 1997

Theatrical productions often call for flaming torches and similar real flame lighting effects. The problem is that for obvious safety reasons, most theatres and halls have strict rules regarding the use of naked flames. Enter The Flickering Flame kit. This lighting effect uses a single 20W halogen lamp (the same as those used for domestic down lights) to mimic its' namesake. Fitted to a compact PCB, it operates from 12V DC and uses just a handful of readily available components.

- PCB, 20W halogen lamp, ceramic base socket, and components.
- 12VDC 2A

	Cat.	Qty 1+	Qty 5+
	KC-5234	$16.95	$14.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

NEON TUBE SOUND DISPLAY

- Make your neon light flash to the music.

Refer Silicon Chip Magazine November 2001

Following the tremendous success of the original 1997 Sound Mod kit, this latest version overcomes some of the original's limitations. With this latest kit you can now use any output from your car stereo - it is not limited to being exclusively driven by a subwoofer output, unlike its predecessor. This kit drives any colour neon tube in the Jaycar range (see website) and has the option of turning the tube either on or off to the beat of the music.

- PCB and components.
- 12VDC

	Cat.	Qty 1+	Qty 5+
	KC-5322	$19.95	$17.70
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

YELLOW PRICEBAR DENOTES CLEARANCE ITEMS

DMX CONTROLLER USB INTERFACE

new

Add computer control to your DJ or stage show. The DMX protocol was developed with the purpose of controlling dimmers, scanners, moving heads and other lighting devices with simple wiring. It is mainly used in theatres and discos, but you can use it in any place where a central or automated lighting is needed. Use this kit to control DMX fixtures such as spotlights using a PC and USB interface. A comprehensive kit that includes software, USB cable and enclosure. There's also a DLL provided so you can write your own software if you like. It can also be operated in stand-alone mode that outputs all 512 channels at the same time (9V battery required for stand-alone mode).

- Test software and DMX Light Player software included
- 512 DMX channels with 256 levels each
- 3 pin XLR-DMX output connector
- Windows 98SE or higher compatible
- Optional 9V battery needed for stand alone test mode
- Dimensions: 106(L) x 100(W) x 44(H)mm

	Cat.	Qty 1+	Qty 5+
	KV-3610	$149.00	$133.95

DMX RELAY CONTROL KIT

new

Control a relay with the DMX512 protocol. It is actually a bus-controlled power driver. The relay will be activated when the DMX value of the set channel equals 140 or more and turns off when the value is 120 or less. Team it with KC-5482 to make a computer-controlled automation system. Short form kit contains DMX-512, XLR plug, PCB and all specified components.

- 512 unique addresses, DIP switch settable
- LED indication for power supply, relay output & error status
- Relay Hold function in case of DMX signal loss
- Switching capacity: 8A
- Supply voltage: 12VDC
- Power consumption: max. 100mA

	Cat.	Qty 1+	Qty 5+
	KV-3612	$49.95	$44.90

DMX CONTROL DIMMER KIT

new

This kit allows you to control a lamp or group of lamps through a DMX signal. You can use the USB Controlled DMX Interface kit (Cat No. KC-5480) or any other control console compliant with the DMX-512 protocol as a controller. It will drive resistive loads like incandescent lamps and mains voltage halogen lighting. Short form kit contains XLR socket, PCB and all specified components.

- 512 unique addresses, selectable with DIP switch
- Status LED for power and error detection
- Toroidal filtering for low noise
- Stand alone mode for testing
- Load capacity: 1000 W @ 230V (5A) or 500W @ 115V
- Power Input: 115/230 VAC
- Dimensions: 150(L) x 60(W) x 45(H)mm

	Cat.	Qty 1+	Qty 5+
	KV-3614	$79.95	$71.95

MAKE YOUR OWN BASKETBALL SCOREBOARD

This kit enables you to make a full-sized electronic scoreboard, in particular for basketball but also adaptable for netball and other games.

- Silicon Chip Magazine March, April, May, August 2005.

It can be built for a fraction of the cost of commercial equivalents but has the fantastic feature of a completely wireless scoring console. You can mount the scoreboard high up in a court and all you need is a convenient 240V power point (115V available on request). You can then control the scoreboard from a table courtside with no messy wiring.

The scoreboard will operate under NBA, FIBA, & NCAA time rules. It features Home/Away team scores 0 to 199, Game period, and Countdown time which reverts to 1/10 second readout when there is one minute left in the quarter.

- All scoreboard functions are on the console with a safety feature of not being able to reset the game unless certain deliberate keyboard protocols are done.
- The console and the scoreboard operate from safe 12V AC adaptors. They can even be operated from a 12 volt car battery!
- The Jaycar kit comes complete with all pre-cut scoreboard woodwork, screen printed face, display filters, mounting plates, pre-programmed microcontroller, printed circuit boards, 2.4GHz transmitters and receivers, pre-punched control console with special piezo end-of-game/quarter sounder, all PCB components, connectors, instructions, solder, etc.

NEW UPGRADED MODEL WITH FOUL SCORING

SCOREBOARD CONSOLE

"HOME TEAM" AND "AWAY TEAM" GRAPHICS FOR ILLUSTRATION ONLY. ACTUAL TEAM NAMES AND SIZE OF NAME ARE UP TO THE CONSTRUCTOR

- The scoreboard measures 900 x 600mm and is approx. 70mm deep.
- The console measures 185(W) x 130(D) x 55(H)mm at rear x 30(H)mm at front.
- The console can control more than one scoreboard simultaneously.

Shipping weight - ask for a quote if ordering through Tech Store.

COMPLETE SCOREBOARD & CONSOLE KIT

Cat.	Qty 1+	Qty 5+
KC-5408	$799.00	$719.00

EXTRA SCOREBOARD

Cat.	Qty 1+	Qty 5+
KC-5409	$649.00	$585.00

LASER LIGHT SHOW

Generate a dazzling laser display using our new laser module Cat. ST-3115 and ST-3117. Using two speed adjustable motors that are fitted with mirrors, patterns similar to a spirograph toy can be projected onto a wall. Great for parties!

- Operating voltage is 6VDC.
- PCB size 100 x 74mm.
- Kit supplied with screen printed gold-plated PCB, 2 motors and mirrors plus all electronic components.
- Laser not included.

related products SEE PAGE 118 FOR OUR LASER MODULES.

	Cat.	Qty 1+	Qty 5+
	KG-9098	$39.95	$35.90
RED LASER MODULE	ST-3115	$17.95 ea	
GREEN LASER MODULE	ST-3117	$99.95 ea	

LUXEON STAR LED DRIVER

Refer Silicon Chip Magazine May 2004

Luxeon high power LEDs are some of the brightest LEDs available in the world. This kit allows you to power one or two 1W or 3W Stars, or a single 5W Star, from 12VDC, efficiently for minimum battery drain. It even features a low battery cutout, making it perfect for the car, boat or caravan.

- PCB and components.
- 12VDC 1A (use MP-3210)

related products SEE PAGE 116 FOR OUR STAR LED RANGE

	Cat.	Qty 1+	Qty 5+
	KC-5389	$29.95	$26.90
12VDC PLUGPACK TO SUIT	MP-3210	$36.95 ea	
RECOMMENDED BOX - UB3	HB-6013	$3.95 ea	

KITS

COMPUTER KITS

Don't chuck out that obsolete piece of junk just yet. Whether at home or at the office, we have a range of clever little computer kits that can extend the capabilities of your PC or Mac. Use your computer as an infrared transceiver to transfer files from your PDA or digital camera; or remotely control your electrical appliances at home via SMS or through your printer port! And for you musos, drag your PC into the garage/studio and turn it into a sophisticated audio test instrument to measure and monitor any sound.

AVR ISP Serial Programmer

SILICON CHIP · 60 mins · DETAILED

Refer Silicon Chip Magazine October 2002

Program, erase and rewrite the program and data memory in your AVR microprocessor without even removing it from the application circuit. This kit connects to the computer serial port, uses royalty-free software available on the Internet and allows you to program a multitude of micros in the AVR 8-bit RISC family, including:

ATTiny12, ATTiny15, AT90S1200, AT90S2313, AT90S2323, AT90S2343, AT90S4414, AT90S4433, AT90S8515, AT90S8535, Atmega83, Atmega103, Atmega161, Atmega163.

- PCB, case with screen printed lid and components.
- Can be used with KC-5421

	Cat.	Qty 1+	Qty 5+
	KC-5340	$44.95	$40.45

AVR Adaptor Board Kit (To Suit KC-5340)

SILICON CHIP · 60 mins · DETAILED

Refer Silicon Chip Magazine March 2006

A low cost method of stand - alone programming for when the application board is unavailable or doesn't include an ISP (or ITAG) header. Program, erase and rewrite the program and data memory in your AVR microprocessor with this socket board. Kit includes everything you need to support in-system programming.

- PCB, IC sockets and components
- In conjunction with KC-5340 AVR ISP Serial Programmer Ref: (Silicon Chip October 2002) • 12VDC 150mA.

	Cat.	Qty 1+	Qty 5+
	KC-5421	$34.95	$31.45
12VDC PLUGPACK TO SUIT	MP-3147	$17.95 ea	
RECOMMENDED BOX - UB2	HB-6012	$6.95 ea	

USB Experimenter's Interface

KwiK Kit · 45 mins · DETAILED

Interface your computer to the real world with this sophisticated USB Kit. There are five digital inputs to monitor the state of external switches or controls and two variable gain analogue inputs that can be connected to sensors and used for measuring temperature, humidity, voltage, current, & other variable parameters. Eight digital and two analogue outputs are available for controlling external equipment. The kit is supplied with all components, silk screened PCB, assembly manual, and software.

- PCB measures 145 x 87mm.
- 5 Digital inputs
- 2 Analogue inputs with variable gain
- Analogue input range 0 to 5VDC
- 8 Open collector digital counters
- 2 Analogue outputs PWM or 0 to 5VDC
- On-board test buttons for inputs & LED
- USB powered
- Win98SE or above (Not NT)

	Cat.	Qty 1+	Qty 5+
	KV-3600	$69.95	$62.95
RECOMMENDED BOX - UB2	HB-6012	$6.95 ea	

SMS Controller for Nokia Phones

SILICON CHIP · 4+ hours · DETAILED

- Control appliances or receive alerts from anywhere!

Refer Silicon Chip November 2004.

Would you like to be alerted via SMS when your burglar alarm has been activated, and which sectors too? How about being able to also reset the alarm if you are confident all is fine? It may seem futuristic, but it is all possible with the SMS controller module. By sending plain text messages, you can control up to eight devices. At the same time, it can also monitor four digital inputs. It works with old Nokia handsets such as the 5110, 6110, 3210, and 3310, which can be bought quite cheap if you do not already own one.

- Requires a Nokia data cable which can be found in mobile phone accessory stores.
- PCB, programmed microcontroller and components
- 12VDC 500mA

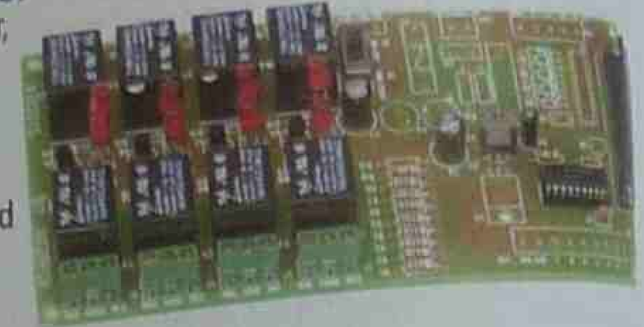
	Cat.	Qty 1+	Qty 5+
	KC-5400	$57.95	$51.95

PC Link for Automatic Control

KwiK Kit · 60 mins · DETAILED

- Automate your house and show your friends how clever you are!

By using our parallel port controller, you can switch up to eight separate devices on or off via your computer's printer port. Could be used for model trains etc. Each SPDT relay can handle 10 amps and has an LED indication to show whether it is on or off.



- Basic software is provided on a 3.5" disk.
- Kit includes PCB, relays, software, and all electronic components.
- 8 - 12VDC power required - use plug pack MP-3136
- Put an old out of service machine back to work!
- Windows 95, 98, 2000 and XP compatible
- Requires direct port access

	Cat.	Qty 1+	Qty 5+
	KV-3590	$69.95	$62.95
9VDC PLUGPACK TO SUIT	MP-3484	$21.95 ea	

Test Instrument Interface for PC Sound Cards

60 mins · MADE IN AUSTRALIA · DETAILED

- Turn your computer into an oscilloscope, audio monitor or spectrum analyser!

Refer Electronics Australia August 1998

Plug the kit into your computer's soundcard and by using software available over the Internet, measure or monitor just about any audio signal. Most PC sound card inputs have a limited dynamic range, relatively low input impedance and poor overload protection - this kit will eliminate these limitations and allow tiny signals, like those from op-amps, to be measured without degradation Or use it for amplifying a high quality microphone for sampling. Power is derived from the soundcard's D15 Joystick output socket.

- PCB, case with screen printed lid, and components.
- Optional CRO probe to suit - use our QC-1902.

	Cat.	Qty 1+	Qty 5+
	KA-1811	$29.95	$26.95
CRO PROBE TO SUIT	QC-1902	$39.95 ea	

PC Infrared Transceiver

SILICON CHIP · 60 mins

- For IRDA connection to your mobile phone etc.

Refer Silicon Chip Magazine December 2001

Did you know that most Pentium-class motherboards include infrared support right out of the box? This quick and easy kit plugs into your motherboard to provide IR support for external devices such as notebook computers, PDAs, digital cameras, data samplers - and the list goes on!

- PCB, wire, and components.

	Cat.	Qty 1+	Qty 5+
	KC-5323	$21.95	$19.70

Get Your SILICON CHIP at Jaycar

Silicon Chip is Australia's world-class magazine for the electronics enthusiast. As well as a great range of fun DIY kits, SILICON CHIP has interesting features articles and popular columns such as "Serviceman's Log" and "Circuit Notebook" etc. Jaycar gets its stock of SILICON CHIP straight off the printing press, so we usually have it a couple of days earlier than the newsagents. Don't forget to call in to a Jaycar store for the latest issue hot off the press.

	Cat.	Qty 1+
	BE-5025	$8.50

SILICON CHIP

POWER KITS

This year we introduce an upgrade to the handy universal motor speed controller kit, enabling you to vary the motor speed of your power tools and other 240V motors. Our neat range of power kits also includes DC relay cards and switches, battery voltage indicators and chargers, and voltage adaptors, regulators and converters.

Universal Drill / Motor Speed Controller Kit

SILICON CHIP · 3 hours · COMPLEX · 240V · new

Refer Silicon Chip Magazine February 2009

Drill speed controllers are nothing new, and in spite of the availability of variable speed power tools, there is still a need for a stand-alone motor speed controller. Apart from power tools, it's often handy to be able to control the speed of other 240V motors. Suitable for brush motors up to 10A, the circuit is a revised version of the popular 5A speed controller from October 2002.

- You'll need a garden-variety IEC lead as well.
- PCB, case with screen printed lid and components

	Cat.	Qty 1+	Qty 5+
	KC-5477	$64.95	$58.50

240V 10A Deluxe Motor Speed Controller Kit

SILICON CHIP · 3 hours · DETAILED · new

Refer Silicon Chip Magazine April 2009

Another motor speed controller? This one is a little bit different and offers boutique performance in the world of speed controllers. While the simpler circuit of our other motor controller kit (KC-5477) will control a motor's speed fairly well, it can't provide maximum speed as the output voltage is limited to about 160VAC maximum. It also doesn't provide great speed regulation and torque at low speeds. The deluxe kit addresses all these issues and provides full speed control from near zero to maximum RPM, good speed regulation under load, very smooth low speed operation and is rated for devices up to 2300W. It also has soft-start to eliminate the "kick" from larger power tools and has interference suppression filtering, fuse protection and over-current protection with limiting. This kit really is the duck's guts for motor speed control. The case has the tricky cutouts pre-machined, but a little bit of extra drilling is required to complete the project.

- Overlay PCB and all components.
- Machined case included

Photo shown is our prototype only

	Cat.	Qty 1+	Qty 5+
	KC-5478	$99.95	$89.95

PC Controlled Stepping Motor Driver

KwiK Kit · 45 mins · DETAILED

This kit will enable you to control the supplied stepper motor manually, or via your computer's parallel port with the software provided. You can accurately control the motors direction, speed and number of rotations. This kit has many uses and is only limited by your imagination. Use it to experiment in robotics, for camera panning, a radio antenna rotator or even to open the curtains in the morning.

- Kit supplied with PCB, stepper motor, software and all electronic components.
- Computer cable required Cat WC-7502.
- Windows 95, 98 or XP recommended
- Requires direct port access

	Cat.	Qty 1+	Qty 5+
	KV-3594	$49.95	$44.90
COMPUTER CABLE TO SUIT	WC-7502	$13.95 ea	

Have you seen our new SCHOOL 40Km/hr AUTOMATIC WARNING ALERT (ref. Silicon Chip April '09)? It includes all components & silkscreen printed front panel. Check it out on Page 82.

Battery Zapper Kit Mk II

SILICON CHIP · 4+ hours · COMPLEX

NEW & IMPROVED PROJECT!

Refer Silicon Chip Magazine May 2006

We received such a great response from satisfied customers of our lead acid battery zapper kit, we've gone one further and given it a great improvement! Like the project from Silicon Chip in July 2005,this kit attacks a common cause of failure in wet lead acid cell batteries: sulphation. Sulphation leads to a early death to your batteries but this is one treatment that thousands of our customers last year gave the thumbs up to. What the circuit does is top produce short burst of high levels of energy to reverse the sulphation effect. Going a few steps further this year, we've added the new feature of battery health checking to give you an indication of the battery's level of health, added new circuit protection against badly sulphated batteries, test points for a DMM, and connection for a battery charger.

- PCB, machined case with printed lid and components.
- 6, 12 or 24VDC.

	Cat.	Qty 1+	Qty 5+
	KC-5427	$109.00	$97.95

Low Voltage Battery Warning

KwiK Kit · 45 mins · DETAILED

This circuit monitors any battery voltage between 3-15 volts once set. Whenever the voltage falls below a predetermined value a red LED lamp lights up. It does not, however, automatically disconnect the battery. Uses a tiny amount of power from the battery being monitored. Could save you embarrassment or a fortune by avoiding battery damage.

- Kwik kit PCB and components

	Cat.	Qty 1+	Qty 5+
	KG-9000	$9.95	$8.75
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

LED Battery Voltage Indicator

15 mins · SIMPLE

- Quick battery condition indication.

Refer Electronics Australia Sept 1995

This tiny circuit measures just 25mm x 25mm and will provide power indication and low voltage indication using a bi-colour LED, and can be used in just about any piece of battery operated equipment. The circuit is suitable for equipment powered from about 6-30VDC. With a simple circuit change, the bi-colour LED will produce a red glow to indicate that the voltage has EXCEEDED a preset value.

- PCB, bi-colour LED and components
- 6-10VDC 3-8mA

	Cat.	Qty 1+	Qty 5+
	KA-1778	$9.95	$8.75
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

Kit Powertool Battery Charger Controller

SILICON CHIP · 3 hours · COMPLEX

Refer Silicon Chip Magazine December 2006

Normal power tool battery chargers are just plain hopeless and do nothing to prolong the life of the batteries in the battery pack. Now you can enhance the performance of the charger supplied with your power tool with this controller. It incorporates charge timeout, min and max temperature monitoring, Delta charge detection, power and charge LED indicator, adjustable Delta V, temperature settings, and optional adjustable trickle charge. Suits both Ni-Cd and Ni-MH cells.

- PCB, case with screen printed lid and components.

	Cat.	Qty 1+	Qty 5+
	KC-5436	$39.95	$35.90

POWER KITS (CONTINUED)

IMPROVED LOW VOLTAGE REGULATOR

Refer Silicon Chip Magazine May 2008

Need to operate a CD, DVD or MP3 player from the cigarette lighter socket in your car? Or perhaps run some powered speakers from the power supply inside your PC? The odds are you'll need this little adaptor to step the voltage down. It has a push-on jumper shunt to select one of six common output voltages 3V, 5V, 6V, 9V, 12V or 15V and, when used with an appropriate input voltage and heatsink, with a thermal resistance of 1.4°C per Watt, can deliver up to four amps at the selected output voltage.

- PCB and components.
- Input voltage 3VDC above the output voltage is required.

Cat.	Qty 1+	Qty 5+
KC-5463	$17.95	$15.95

VOLTAGE REGULATOR KIT

Refer Silicon Chip Magazine May 2007

This handy voltage regulator can provide up to 1,000mA at any voltage from 1.3 to 22VDC. Ideal for experimental projects or as a mini bench power supply.

- PCB and components
- Heatsink may be required depending on input voltage and output current

	Cat.	Qty 1+	Qty 5+
	KC-5446	$16.95	$14.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

VERSATILE REGULATED VOLTAGE ADAPTOR

Refer Electronics Australia August, 1997

This kit is a low-powered DC converter for many applications, for example, where you need a reliable voltage source for a portable CD player or similar battery-powered device. Another use for this adaptor is for a peripheral computer power supply to power external Zip drives, powered speakers, modems, music/MIDI keyboards, etc. Just plug its input into your PC's internal power supply cable and get selectable regulated voltage out from 3 to 15VDC. Output current capability is around 1.5 amps (depending on the size of heatsink used - not included, available separately).

- PCB and components
- Input voltage 3VDC above output voltage

	Cat.	Qty 1+	Qty 5+
	KA-1797	$7.95	$6.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

3V TO 9V DC - DC CONVERTER

- Never buy an expensive 9V battery ever again

Refer Silicon Chip Magazine March 2004

9V batteries are a great source of portable power, but let's face it - they don't last long if you want more than a few milliamps, & they are not real cheap either. This nifty converter allows you to use regular Ni-Cd or Ni-MH 1.2V rechargeables, or Alkaline 1.5V cells for 9V applications. It can run from AA, C or D cells, deliver up to 90mA output at 9V.

- Output can be set from 4.5-20VDC.
- 12VDC plugpack input, optional trickle charge for rechargeable batteries.
- PCB and components.
- The required holder for the batteries depends on the supply cells used.

	Cat.	Qty 1+	Qty 5+
	KC-5391	$15.95	$14.35
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

FAST NI-MH BATTERY CHARGER KIT

Refer Silicon Chip Magazine September 2007

A truly versatile charger, capable of handling up to 15 of the same type of Ni-MH or Ni-Cd cells. Build it to suit any size cells or cell capacity and set your own fast or trickle charge rate. It also has overcharge protection including temperature sensing. Ideal for R/C enthusiasts who burn through a lot of batteries.

- PCB, programmed micro and components.
- Case, heatsink and battery holder not included.

Cat.	Qty 1+	Qty 5+
KC-5453	$42.95	$38.45

SUPER SIMPLE 12VDC-240VAC INVERTER

This circuit is ideal if you need 240V for a small appliance away from mains. It says it will drive up to 70W peak but we would rate it at 30W RMS max. (Not really RMS as it is square wave). In addition to this kit you will need 2 x Heatsinks such as HH-8660 (page 158) and a step-up transformer such as the MM-2015 or, better MM-2016

Tip: If you have a big 12-0-12/240 transformer in your junk box this will end up a cheap inverter.

Cat.	Qty 1+	Qty 5+
KG-9006	$19.95	$17.70

12VDC RELAY CARD

- Add a relay output to your equipment.

This kit will close a relay's contacts with as little as 5mA to trigger the circuit. Literally any kit you see on these pages that uses an LED as a trip-condition indicator, can be used with this nifty project. Use the relay to sound buzzers, switch on lights, operate solenoids, trigger alarms, etc. Operation: common ground - output LED to ground

- Kit includes Kwik Kit PCB, relay plus electronic components.
- LED cathode must be connected to ground.

Cat.	Qty 1+	Qty 5+
KG-9142	$9.95	$8.75

DC RELAY SWITCH

Refer Silicon Chip Magazine November 2006

An extremely useful and versatile kit that enables you to use a tiny trigger current - as low as 400µA at 12V to switch up to 30A at 50VDC. It has isolated input, and is suitable for a variety of triggering options, including an AC or oscillating signal. It also has a relay-on LED indicator.

- PCB, relay and components.
- 12VDC 150mA.
- Recommended enclosure HB-6015

	Cat.	Qty 1+	Qty 5+
	KC-5434	$16.95	$14.95
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	

4 CHANNEL RELAY BOARD WITH IP CONTROL

This 4 input switcher can be given an IP address which allows you to switch up to 4 devices. Turn on your security cameras and view what's going on at your premises by switching your cameras on from anywhere in the world! TVs, air conditioners, lighting, or other home appliances can be switched on or off by utilising this module.

- Polarity protection
- 4 Inputs
- Supports; HTTP,IP,DDNS, and DHCP
- Embedded Web Server
- Password protection

Cat.	Qty 1+	Qty 5+
KV-3595	$99.95	$89.95

UNIVERSAL +/-15V POWER SUPPLY

Refer Silicon Chip Magazine August 1988

This small kit enables you to obtain +15V, -15V or ±15V DC from a number of different transformer and rectifier combinations.

- Obtain ±15V rails from 12V AC plugpack, or 2851 transformer (MM-2006)
- Obtain ±15V rails from 30V AC centre tapped (MM-2007) transformer
- Revised PCB and components to build one of the options listed above.

	Cat.	Qty 1+	Qty 5+
	KC-5038	$14.95	$13.35
TRANSFORMER TO SUIT	MM-2006	$7.95 ea	
RECOMMENDED BOX - UB5	HB-6015	$2.95 ea	
CT TRANSFORMER TO SUIT	MM-2007	$8.95 ea	

TEST EQUIPMENT

Testing equipment need not cost you an arm and a leg. We have a nifty range of test kits for the hobby enthusiasts, learning tradesperson or simply for those looking for a cost-effective solution for simple testing instruments.

SERIAL PROGRAMMER KIT FOR DSPIC30F SERIES PIC MICROCONTROLLERS

Refer Silicon Chip Magazine May 2008

The dsPIC30F series of micro-controllers are extremely useful but some older PIC programmers cannot program them due to pinning incompatibilities and other issues. This very affordable programmer kit can handle all the dsPIC30F family and almost all of the regular PICs available in a DIP package. It uses freely available software for PCs and is easy to build. It will work with any RS232 serial port and most USB-RS232 converters. If you have ever wanted to experiment with DSPs, the dsPIC30F series is a good starting point. Microchip offers a lot of free documentation and source code on their website, so getting started should be a breeze.

new

- Double sided plated through hole PCBs, 2 x 40 pin tip sockets and components
- 16VDC 400mA

Cat.	Qty 1+	Qty 5+
KC-5467	$74.95	$67.50

DIGITAL MULTIMETER KIT

Learn everything there is to know about component recognition and basic electronics with this comprehensive kit. From test leads to solder, everything you need for the construction of this meter is included together with a detailed manual. With test questions and schematic supplied in the manual, the kit can be geared to an individual or class learning environment, making it an excellent choice for first year trade apprentices. Kit includes DMM case, LCD, solder, battery, test leads, PCB, comprehensive 18 page learning manual and electronic components.

All you'll need is a soldering iron.

- Meter dimensions: 67(W) x 123(H) x 25(D)mm

Cat.	Qty 1+	Qty 5+
KG-9250	$24.95	$22.35

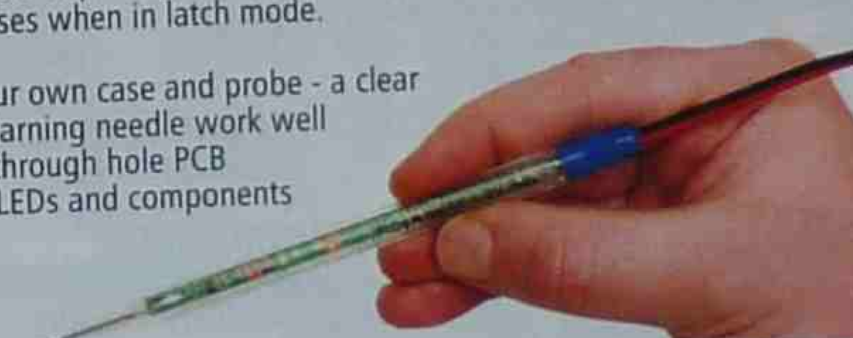
PIC LOGIC PROBE KIT

SMD

Refer Silicon Chip Magazine October 2007

Most logic probes are designed to operate on the 5V rails that have been around in logic circuits for years. This design operates on a wide voltage range down to 2.8V so it's suitable for use on the most modern circuits. It's also extremely compact with SMT devices on a PCB only 5mm wide, so it will fit inside a very slim case. It's capable of picking up a pulse only 50mS long and will also detect and hold infrequent pulses when in latch mode.

- You'll need to add your own case and probe - a clear ballpoint pen and a darning needle work well
- Double sided plated through hole PCB programmed, micro, LEDs and components
- 2.8-15VDC



Cat.	Qty 1+	Qty 5+
KC-5457	$16.95	$14.95

ELECTRONIC TEST BENCH MAGAZINE SILICON CHIP

This book contains plans and instructions regarding a selection of the best test equipment kits from the pages of Silicon Chip magazine. Includes details explaining 18 kits, from a low-cost dual tracking power supply, to a compact pink noise generator.

- Softcover, 205 x 275mm. • 128 pages.

Cat.	Qty 1+
BS-5070	$10.95

IN CIRCUIT TRANSISTOR TESTER

- Test the suspect transistor without having to desolder it.

Refer Electronics Australia September 1983

Have you ever unsoldered a suspect transistor only to find that it checks OK? Troubleshooting exercises are often hindered by this type of false alarm. You can avoid these hassles with the "In-Circuit Transistor, SCR and Diode Tester".

The kit does just that, test drives WITHOUT the need to unsolder them from the circuit! VERY HANDY!

- Revised PCB, 9V battery, components & case with screen printed panel showing truth table for device checking.
- Revised PCB by Jaycar! No more messy wiring. Easy assembly!
- 9VDC

Cat.	Qty 1+	Qty 5+
KA-1119	$27.95	$24.95

50MHZ FREQUENCY METER MK 2

Refer Silicon Chip Magazine February 2007

An invaluable service tool, this frequency meter kit features an 8-digit LCD, automatic indication of units (Hz, kHz, MHz or GHz) and prescaler.

- PCB, LCD, components and case with machined and screen printed lid.
- Input sensitivity: <20mV RMS from 1Hz to 100kHz, 50mV at 20MHz and 85mV at 50MHz.
- Input Imped: 1.1MW in parallel with about 10pF
- Frequency range: 0.1Hz to 50MHz or better
- Untrimmed accuracy: ±20ppm equivalent to 1000Hz at 50MHz
- Trimmed accuracy: ±10ppm from -20°C to 70°C
- High Resolution Mode: 0.1Hz from 0.1-150Hz; 1Hz from 150Hz-16MHz; 10Hz from 16-50MHz. Low Resolution Mode: 1Hz from 1-999Hz; 10Hz from 1kHz-50MHz
- Update time (approx.): 200ms for 10Hz resolution; 1s for 1Hz resolution; 1s for 0.1Hz resolution to 10Hz, 10s at 0.1Hz
- Display units: Hz from 0.1-999Hz; kHz from 1-999.999kHz; MHz from1-50MHz
- 7.5-12VDC 65mA

Cat.	Qty 1+	Qty 5+
KC-5440	$69.95	$62.95

SCI - FI EFFECTS

George Lucas eat your heart out. Whether you're a budding sci-fi movie producer or simply wanting to impress your infrequent intergalactic visitors, our Galactic Voice Kit and Jacob's Ladder MkII are sure to impress even the most discerning Klingon or Dalek.

GALACTIC VOICE KIT

Refer Silicon Chip Magazine September 06.

Be the envy of everyone at the next Interplanetary Conference for Evil Beings with this galactic voice simulator kit. Effect and depth controls allow you to vary the effect to simulate everything from the metallically-challenged C-3PO, to the hysterical ranting of Daleks hell-bent on exterminating anything not nailed down.

For those who really need to get out of the house a lot more. Take me to your leader....

- PCB, enclosure, panels and components.
- 9VDC (battery)

Cat.	Qty 1+	Qty 5+
KC-5431	$45.95	$41.45

JACOB'S LADDER HIGH VOLTAGE DISPLAY KIT MK2

Refer Silicon Chip Magazine April 2007

With this kit and the purchase of a 12V ignition coil (available from auto stores and parts recyclers), create an awesome rising ladder of noisy sparks that emits the distinct smell of ozone. This improved circuit is suited to modern high power ignition coils and will deliver a spectacular visual display. Inspired by the good doctor's laboratory in the Frankenstein movie, use this kit for theatre special effects or just to impress your friends.

- PCB, pre-cut wire/ladder and components.
- Requires 12VDC
- 12V automotive ignition coil (Holden VN commodore equivalent)
- 12V 7Ah battery or 12VDC >5A power suply

Warning: The Jacobs Ladder Kit uses potentially dangerous voltage.

Cat.	Qty 1+	Qty 5+
KC-5445	$42.95	$38.45

JAYCAR ELECTRONICS RETAIL STORE QUICK LOCATION REFERENCE

AUSTRALIA

ACT

BELCONNEN
54 Weedon Cl, 2617
Ph: (02) 6253 5700
Mon-Thurs: 8.30-5.30
Fri: 8.30-6.30
Sat: 8.30-5.00
Sun: 10.00-4.00

FYSHWICK
121 Wollongong St, 2609
Ph: (02) 6239 1801
Mon-Fri: 8.30-6.00
Sat: 8.30-5.00
Sun: 10.00-4.00

NSW

- opening hours
Mon-Fri: 8.30-5.30
Sat: 8.30-5.00
Sun: 10.00-4.00
* Exceptions/Late Trading see Stores.

ALBURY
565 Hume St, 2640
Ph: (02) 6021 6788
*Thurs: 8.30-7.00

ALEXANDRIA
647 Botany Rd, 2015
Ph: (02) 9699 4699
*Thurs: 8.30-7.30

BANKSTOWN
363 Hume Hwy, 2200
(CNR Meredith St)
Ph: (02) 9709 2822
*Thurs: 8.30-7.30

BLACKTOWN
102 Sunnyholt Rd, 2148
(CNR Bessemer St)
Ph: (02) 9678 9669
*Thurs: 8.30-7.30

BONDI JUNCTION
125 Bronte Rd, 2022
(CNR Birrell Rd)
Ph: (02) 9369 3899
*Thurs: 8.30-7.30

BROOKVALE
208 Harbord Rd, 2100
Ph: (02) 9905 4130
*Thurs: 8.30-7.30

CAMPBELLTOWN
Shop 1, 50 Queen St, 2560
Ph: (02) 4620 7155
*Thurs: 8.30-7.30

ERINA
1/217 The Entrance Rd, 2250
Ph: (02) 4365 3433
*Thurs: 8.30-7.30

GORE HILL
188 Pacific Hwy, 2065
Ph: (02) 9439 4799
*Thurs: 8.30-7.30

HORNSBY
130 George St, 2077
Ph: (02) 9476 6221
*Thurs: 8.30-7.30

LIVERPOOL
3/381 Macquarie St, 2170
(CNR Castlereagh St)
Ph: (02) 9821 3100
*Thurs: 8.30-7.30

NEWCASTLE
224 Maitland Rd, (CNR Dent St)
Islington, 2296
Ph: (02) 4965 3799
*Thurs: 8.30-7.30

PENRITH
20 Henry St, 2750
Ph: (02) 4721 8337
*Thurs: 8.30-7.30

RYDALMERE
320 Victoria Rd, 2116
Ph: (02) 8832 3120
*Thurs: 8.30-7.30

SYDNEY CITY
127 York St, 2000
(Opposite QVB)
Ph: (02) 9267 1614
*Mon-Fri: 8.30-6.00
*Thurs: 8.30-8.00

TAREN POINT
49A Captain Cook Dr, 2229
Ph: (02) 9531 7033
*Thurs: 8.30-7.30

WOLLONGONG
357 Keira St, 2500
(CNR Swan St)
Ph: (02) 4226 7089
*Thurs: 8.30-7.30

NT

DARWIN
297 Bagot Rd, Coconut Grove, 0810
Ph: (08) 8948 4043
Mon-Thu: 8.30-5.30
Fri: 8.30-7.30
Sat: 8.30-5.00
Sun: 10.00-4.00

QLD

- opening hours
Mon-Fri: 8.30-5.30
Sat 8.30-5.00
Sun 10.00-4.00
* Exceptions/Late Trading see stores

ASPLEY
1322 Gympie Rd, 4034
Ph: (07) 3863 0099
*Thurs: 8.30-7.30

CAIRNS
194 Mulgrave Rd, 4870
Ph: (07) 4041 6747
*Thurs: 8.30-7.30

GOLD COAST
2474 Gold Coast Hwy, Mermaid Beach, 4218
Ph: (07) 5526 6722
*Thurs: 8.30-7.30

IPSWICH
Shop 2, 160 Brisbane Rd, Booval, 4304
(OPP Booval Fair)
Ph: (07) 3282 5800
*Thurs: 8.30-7.30

MACKAY
88 Sydney Street, 4740
(CNR of Alfred Street)
Ph: (07) 4953 0611
*Thurs: 8.30-7.30

MAROOCHYDORE
100 Aerodrome Rd, 4558
Ph: (07) 5479 3511
*Thurs: 8.30-7.30

TOWNSVILLE
177 Ingham Rd, (CNR Edwards St)
West End, 4810
Ph: (07) 4772 5022
*Thurs: 8.30-6.30

TWEED HEADS SOUTH
Unit 2, No.1 CNR Greenway & Machinery Dr, 2486
Ph: (07) 5524 6566
*Thurs: 8.30-7.30

UNDERWOOD
Unit 2/3255 Logan Rd, 4119
Ph: (07) 3841 4888
*Thurs: 8.30-7.30

WOOLLOONGABBA
65 Ipswich Rd, 4102
(CNR Wilton St)
Ph: (07) 3393 0777
*Thurs: 8.30-7.30

SA

- opening hours
Mon-Fri: 8.30-5.30
Sat 8.30-5.00
Sun 10.00-4.00
* Exceptions/Late Trading see Stores

ADELAIDE
221-223 Morphett St, 5000
Ph: (08) 8231 7355
*Fri: 8.30-7.30

CLOVELLY PARK
1168A, South Rd, 5042
Ph: (08) 8276 6901
*Thurs 8.30-7.30

GEPPS CROSS
Shop 3B, 580 Main Nth Rd, 5094
Ph: (08) 8262 3200
*Thurs: 8.30-7.30

TAS

HOBART
245 Main Rd, Derwent Park, 7009
Ph: (03) 6272 9955
Mon-Thurs: 8.30-5.30
Fri: 8.30-6.30
Sat: 8.30-5.00
Sun: 10.00-4.00

LAUNCESTON
167 Brisbane St, 7250
Ph: (03) 6334 2777
Mon-Thurs: 8.30-5.30
Fri: 8.30-6.30
Sat: 8.30-5.00
Sun: 10.00-4.00

VIC

- opening hours
Mon-Thurs: 8.30-5.30
Sat: 8.30-5.00
Sun 10.00-4.00
* Exceptions/Late Trading see stores

CHELTENHAM
315 Warrigal Road, 3192
Ph: (03) 9585 5011
*Fri: 8.30-7.30

COBURG
266 Sydney Rd, 3058
Ph: (03) 9384 1811
*Fri: 8.30-7.30

FRANKSTON
80 Dandenong Rd West, 3199
Ph: (03) 9781 4100
*Fri: 8.30-7.30

GEELONG
180 Moorabool St, 3220
(CNR Little Ryrie St)
Ph: (03) 5221 5800
*Fri: 8.30-7.00

HALLAM
1/261 Princes Highway
Tel: (03) 9796 4577
*Fri: 8.30-7.30

MELBOURNE CITY
45 A'Beckett St, 3000
Ph: (03) 9663 2030
*Fri: 8.30-7.00

RINGWOOD
94 Maroondah Hwy, 3134
Ph: (03) 9870 9053
*Fri: 8.30-7.30

SPRINGVALE
CNR Springvale Rd & Dandenong Rd, Mulgrave, 3170
Ph: (03) 9547 1022
*Fri: 8.30-8.30

SUNSHINE
204 McIntyre Rd, 3020
Ph: (03) 9310 8066
*Fri: 8.30-7.30

THOMASTOWN
225B Settlement Rd, 3074
Ph: (03) 9465 3333
*Fri: 8.30-7.30

WERRIBEE
CNR of Synnot & Wedge St, 3030
Tel 1800 022 888
*Fri: 8.30-7.30

WA

- opening hours
Mon-Fri: 8.30-5.30
Sat: 8.30-5.00
Sun CLOSED
* Exceptions/Late Trading see Stores

MADDINGTON
1A 1808 Albany Hwy, Kenwick, 6107
Ph: (08) 9493 4300
*Thurs: 8.30-8.00

MIDLAND
Unit 1, 203 Gt Eastern Highway, 6056
Ph: (08) 9250 8200
*Thurs: 8.30-7.30

PERTH
326 Newcastle St, Northbridge, 6003
Ph: (08) 9328 8252
*Thurs: 8.30-7.30
*Sun: 12.00-5.00

ROCKINGHAM
Shop 1, 61 Dixon Rd, 6168
Ph: (08) 9592 8000
Thurs: 8.30-7.30

NEW ZEALAND

- opening times
Mon-Fri: 8.30-5.30
Sat 9.00-5.30
Sun 10.00-4.00
* Exceptions/Late Trading see Stores

AUCKLAND - GLENFIELD
135 Wairau Rd, 0627
(CNR Tristram Ave)
Ph: (09) 444 4628
*Fri: 8.30-6.30

AUCKLAND - MANUKAU
12 Earl Richardson Ave, 2104
Ph: (09) 263 6241

AUCKLAND - NEWMARKET
231 Khyber Pass Rd, 1023 (CNR Boston Rd)
Ph: (09) 377 6421
*Thurs: 8.30-7.00
*Sat: 8.30-5.30

CHRISTCHURCH
409 Colombo St, 8023
(CNR Wordsworth St)
Ph: (03) 379 1662

DUNEDIN
1 Lees St, Kensington, 9016 (CNR Princes St)
Ph: (03) 471 7934
Fri: 8.30-6.30

HAMILTON
92 Commerce St, Frankton, 3204
Ph: (07) 846 0177
*Thurs: 8.30-6.30

HASTINGS
819 Heretaunga St West, 4120
Ph: (06) 876 0239

PALMERSTON NORTH
2-4 Ada St, 4410
Ph: (06) 353 8246
Fri: 8.30-6.30

WELLINGTON
264-272 Taranaki St, 6011
Ph: (04) 801 9005

NEW LOCATION | NEW STORE

JAYCAR AUTHORISED STOCKISTS

If you operate or plan to open a professional electronics retail business and would like to increase your income, we would like to hear from you. Call 1300 738 555 or email dealers@jaycar.com.au

For the latest and most up-to-date list of authorised Jaycar Stockists visit our website.

Jaycar Electronics AUTHORISED STOCKISTS

PREMIUM STOCKISTS
▲ Stockists highlighted are classed as our Premium Stockists. They carry a wide range of Jaycar products at any time.

STOCKISTS
▲ Stockists shown that are highlighted are located throughout regional Australia and carry the standard Stockist core range of products.

AGENTS
▲ Agents that are not highlighted are located in smaller regional areas and carry a small range of Jaycar core range.

STOCKISTS

AUSTRALIA

NSW

ARMIDALE
Amac Digital Products
Ph: (02) 6771 2266
Fax: (02) 6772 1169

BALLINA
Leading Edge Electronics
Ph: (02) 6686 3610
ballina@bigpond.com

BATHURST
Leading Edge Electronics
Ph: (02) 6332 3189
Fax: (02) 6332 4501

BEGA
Pyramid Power House
Ph: (02) 6492 3355
Fax: (02) 6494 3096

BOWRAL
Leading Edge Electronics
Ph: (02) 4861 3200
Fax: (02) 4861 6400

BRAIDWOOD
RJ Nomchong Electrics
Ph: (02) 4842 2233
rj.nomchong@bettastores.com.au

BROKEN HILL
Leading Edge Computers & Electronics
Ph: (08) 8087 8633
Fax: (08) 8088 7956

CASINO
CS & B Hardware
Ph: (02) 6662 3022
Fax: (02) 6662 2619

COROWA
Reconnect Communications
Ph: (02) 6033 2211
corowa@reconnect.net.au

DENILIQUIN
Wired Entertainment
Ph: (03) 5881 3555
Fax: (03) 5881 3164

DUBBO
IT West
Ph: (02) 6884 3188
sales@itwest.biz

FORSTER
Powerpoint Electronics
Ph: (02) 6554 5006
ppoint@midcoast.com.au

GLEN INNES
Home Living Glen Innes
Ph: (02) 6732 3316
admin@homelivingonline.com.au

GLOUCESTER
Autolec
Ph: (02) 6558 1600
autolecg@bigpond.net.au

GRAFTON
Downes Electronics
Ph: (02) 6642 1911
Fax: (02) 6643 3691

GRIFFITH
Miatronics
Ph: (02) 6962 4534
gthdse@bigpond.net.au

ILUKA
Crystal Solutions
Ph: (02) 6646 5448
Fax: (02) 6646 5448

KEMPSEY
Powertronics Australia
Ph: (02) 6562 8222
kempsey@powertronics.com.au

INVERELL
BK Oliver Home Hardware
Ph: (02) 6722 1306
Fax: (02) 6722 5341

LEETON
Ezone Electronics & Entertainment
Ph: (02) 6953 2800
sales@ezone.net.au

LIGHTNING RIDGE
T F Lapidary
Ph: (02) 6829 1184
Fax: (02) 6829 1184

LISMORE
Lismore Phones & Car Audio
Ph: (02) 6622 2232
lpaca@bigpond.com

LITHGOW
Leading Edge Electronics
Ph: (02) 6352 3333
Fax: (02) 6352 2233

MOREE
Scott Murray Security Services
Ph: (02) 6752 6555
morecom@bigpond.com

MURWILLUMBAH
Leading Edge Electronics
Ph: (02) 6672 3684
Fax: (02) 6672 6053

MUSWELLBROOK
Leading Edge Electronics
Ph: (02) 6543 1904
info@headoverheels.com.au

NAROOMA
Narooma Picture Framing & Photographic
Ph: (02) 4476 1755
Fax: (02) 4476 1755

NARRABRI
Busy Bee Auto Electrics
Ph: (02) 6792 1295
Parts@busybee-elec.com.au

OBERON
Oberon Video And Disc
Ph: (02) 6336 0199
Fax: (02) 6336 0199

PARKES
Griffins Leading Edge
Ph: (02) 6862 2593
Fax: (02) 6862 5773

RICHMOND
Hawkesbury Electronics
Ph: (02) 4578 4006
jaycarsales@dt.com.au

SALAMANDER BAY
Pacific Satellite Pty Ltd
Ph: (02) 4982 0922
Fax: (02) 4982 0922

SINGLETON
Singleton Hi-Fi
Ph: (02) 6572 2477
terry@singletonhifi.com.au

TAREE
Noeldge Systems
Ph: (02) 6551 3622
sales@noeldge.com.au

TEMORA
Reconnect Communications
Ph: (02) 6977 4455
temora@reconnect.net.au

TENTERFIELD
Tenterfield Hardware & Gardens
Ph: (02) 6736 1341
Fax: (02) 6736 3493

TUMBARUMBA
Fixsell Computers
Ph: (02) 6948 2883
sales@fixsell.com.au

TUMUT
Reconnect Communications
Ph: (02) 6947 6699
tumut@reconnect.net.au

ULLADULLA
Leading Edge Electronics
Ph: (02) 4454 5588
leeulladulla@scoastnet.com.au

WAGGA WAGGA
Leading Edge Electronics
Ph: (02) 6931 9333
Fax: (02) 6971 8778

WEST WYALONG
Reconnect Communications
Ph: (02) 6972 2227
westwyalong@reconnect.net.au

YASS
Yass Computers
Ph: (02) 6226 5555
sales@yasscomputers.com.au

NT

ALICE SPRINGS
Alicetronics
Ph: (08) 8952 3411
info@alicetronics.com

KATHERINE
Johnny's Electronics
Ph: (08) 8972 1102
Fax: (08) 8972 1142

TENNANT CREEK
Leading Edge Computers Tennant Creek
Ph: (08) 8962 3907
Fax: (08) 8962 3127

QLD

BRIBIE ISLAND
Banksia IT Solutions
Ph: (07) 3408 9525
info@banksiasolutions.com.au

BUNDABERG
Tri-Dek Lighting & Electrical
Ph: (07) 4153 1423
Fax: (07) 4152 4067

CALOUNDRA
Leading Edge Electronics
Ph: (07) 5491 8533
Fax: (07) 5491 8511

CHARTERS TOWERS
Hi-Tech Solutions
Ph: (07) 4787 1122
Fax: (07) 4787 2016

COOKTOWN
Everything Electrical
Ph: (07) 4069 5860
Fax: (07) 4069 5765

COOROY
Cooroy Inks & Electronics
Ph: (07) 5447 6365
Fax: (07) 5447 7898

EMERALD
Central Western Communications
Ph: (07) 4987 5520
Fax: (07) 4987 5520

GAYNDAH
Gayndah Retravision
Ph: (07) 4161 1355
gayndah@retravision.com.au

GLADSTONE
Leading Edge Electronics Gladstone Hi-Fi
Ph: (07) 4972 6660
leegladstone@bigpond.com

GOONDIWINDI
Tobacco Station Goondiwindi
Ph: (07) 4671 4600
Fax: (07) 4671 0027

GYMPIE
Leading Edge Electronics
Ph: (07) 5483 6677
info@cooloolaelectronics.com.au

KINGAROY
South Burnett Computers
Ph: (07) 4160 4800
sales@sbcomputerservices.com.au

MAREEBA
Outback Solar & Wind
Ph: (07) 4092 1659
OutSolar@bigpond.net.au

MONTO
Monto Retravision
Ph: (07) 4166 1436
chalmers@retravision.com.au

MORANBAH
G & D Communications
Ph: (07) 4941 6611
admin@gdcomms.com.au

MOSSMAN
Powerhouse Appliance Centre
Ph: (07) 4098 3659
Fax: (07) 4098 3659

ROMA
Roma Computer Centre
Ph: (07) 4622 5300
rccsales@westnet.com.au

ST GEORGE
Morecom Computers
Ph: (07) 4625 1236
Fax: (07) 4625 3653

STANTHORPE
Topbeat Entertainment & Electronics
Ph: (07) 4681 0681
Fax: (07) 4681 0682

TOOWOOMBA
Vidcam
Ph: (07) 4639 1100
sales@vidcam.com.au

WARWICK
Topbeat Entertainment & Electronics
Ph: (07) 4661 2690
Fax: (07) 4667 1764

YEPPOON
Leading Edge Electronics
Ph: (07) 4939 1555
Fax: (07) 4939 1988

SA

BALAKLAVA
Balaklava Newsagency
Ph: (08) 8862 1725
Fax: (08) 8862 1725

BARMERA
Barmera Electronics
Ph: (08) 8588 2381
Fax: (08) 8588 2382

GAWLER
Contact Communications
Ph: (08) 8522 5558
Fax: (08) 8523 0510

KADINA
Kadina Computers & Electronics
Ph: (08) 8821 1299
kadcom@westnet.com.au

KINGSCOTE
Ingrams Home & Timber Hardware
Ph: (08) 8553 2822
Fax: (08) 8553 2690

LOXTON
Scrafe's Electronics
Ph: (08) 8584 6700
Fax: (08) 8584 4700

MILLICENT
Roger Smith Auto
Ph: (08) 8733 3444
Fax: (08) 8733 5007

MINLATON
Minlaton Discounters
Ph: (08) 8853 2262
Fax: (08) 8853 2326

MOUNT BARKER
Just Fix It Electronics & Computers
Ph: (08) 8391 5554
Fax: (08) 8391 3641

MOUNT GAMBIER
Anvil Electronics
Ph: (08) 8725 3366
Fax: (08) 8725 3066

MURRAY BRIDGE
Classic Lights Of Murray Bridge
Ph: (08) 8532 1433
classiclights@internode.on.net

NARACOORTE
RM Electronics
Ph: (08) 8762 0120
rob@rmelectronics.com.au

NURIOOTPA
Perry Trade Services
Ph: (08) 8562 2269
sales@perrytrade.com.au

STRATHALBYN
Stone Electronics
Ph: (08) 8536 3300
Fax: (08) 8536 3471

TAS

BURNIE
Electronics Plus
Ph: (03) 6432 2644
stevelec@southcom.com.au

DEVONPORT
A I Electronics & Hobbies
Ph: (03) 6424 8322
aielec@bigpond.com

SMITHTON
Taztec Computers
Ph: (03) 6452 3366
Fax: (03) 6452 3536

VIC

ALEXANDRA
Bourke Refrigeration
Ph: (03) 5772 1147
bourke@virtual.net.au

BACCHUS MARSH
Westconnect Technologies
Ph: (03) 5367 7336
Fax: (03) 5367 5846

BALLARAT
Ballarat Electronics
Ph: (03) 5333 3344
Fax: (03) 5333 3355

BENDIGO
PCB Electronics
Ph: (03) 5444 4044
Fax: (03) 5444 4045

ECHUCA
Switched On Electronics
Ph: (03) 5480 2122
swit@bigpond.net.au

HEALESVILLE
Shorelec Electrical
Ph: (03) 5962 6500
Fax: (03) 5962 6499

HORSHAM
Hobby N Tronix
Ph: (03) 5382 2319
sales@hobbyntronix.com.au

KILMORE
Leading Edge Computers
Ph: (03) 5781 0400
sales@nl.com.au

MANSFIELD
Mansfield Internet & Computers
Ph: (03) 5775 3066
sales@office.mansfield.net.au

MARYBOROUGH
Microblast Computers
Ph: (03) 5460 4006
sales@microblastcomputers.com

ORBOST
Leading Edge Video
Ph: (03) 5154 2212
leadingedgeorbost@eftel.net.au

PAKENHAM
Pakenham Photo & Electronics
Ph: (03) 5941 1562
dspargo@spargophoto.com.au

ROSEBUD
Wild World
Ph: (03) 5982 2225
Fax: (03) 5986 1538

SALE
The Electronic Barn
Ph: (03) 5144 7700
elecbarn@westnet.com.au

STAWELL
David O Jones Mitre 10
Ph: (03) 5358 1205
Fax: (03) 5358 4062

SUNBURY
Sunbury City Electrical & Electronics
Ph: (03) 9744 4144
sunbury@ssc.net.au

SWAN HILL
Electronics Doctor
Ph: (03) 5032 4716
Fax: (03) 5032 3623

TRARALGON
The Electronics Hive
Ph: (03) 5174 2266
electronicshive@gmail.com

WARRACKNABEAL
PA & FE Johns
Ph: (03) 5394 1444
Fax: (03) 5398 1333

WARRAGUL
LE Computers Warragul
Ph: (03) 5622 2999
jaycar@lecw.com.au

WONTHAGGI
Raneye Systems
Ph: (03) 5672 4774
sales@raneye.com.au

WA

AUGUSTA
Leeuwin Networks
Ph: (08) 9758 1444
leeuwinnet@bigpond.com

BUNBURY
Leading Edge Telecoms & Hi Fi
Ph: (08) 9721 6222
Fax: (08) 9721 9686

CARNARVON
Electronics Plus
Ph: (08) 9941 3690
Fax: (08) 9941 3660

ESPERANCE
Esperance Communications
Ph: (08) 9071 3344
sales@esperancecomms.com.au

GERALDTON
ML Communications
Ph: (08) 9965 7555
sales@mlcomms.com.au

KARRATHA
NW Communications & I.T. Specialists
Ph: (08) 9143 1134
Fax: (08) 9143 1135

MOUNT BARKER
Have A Browse
Ph: (08) 9851 2524
Fax: (08) 9851 2524

NARROGIN
Great Southern Communications
Ph: (08) 9881 4881
Fax: (08) 9881 4991

YORK
Aussie Alternative Energies
Ph: (08) 9641 2133
Fax: (08) 9641 2033

NEW ZEALAND

DANNEVIRKE
Brownings Electronic Services
Ph: (06) 374 6322
browningselectronics@xtra.co.nz

FEILDING
Dr Electronics
Ph: (06) 323 7921
info@drelectronics.co.nz

GISBORNE
A&E Appliances & Electronics
Ph: (06) 863 3099
Fax: (06) 863 3990

INVERCARGILL
Gracetek Communications Ltd
Ph: (03) 211 0046
sales@gracetek.co.nz

KERIKERI
Retravision
Ph: (09) 407 8514
Fax: (09) 407 9672

MASTERTON
Electric Blue Electronics
Ph: (06) 377 1870
electricblue@wizbiz.net.nz

NEW PLYMOUTH
Plug N Play Electronics
Ph: (06) 758 0050
sales@plugnplay.co.nz

OAMARU
Oamaru Computer Services
Ph: (03) 434 2084
sales@oamarucomputers.co.nz

TAUPO
Copy Solutions Taupo
Ph: (07) 378 6278
Fax: (07) 377 2347

TAURANGA
Boonen Electronics
Ph: (07) 579 4861
boonen.tga@xtra.co.nz

TIMARU
KM Electronics
Ph: (03) 688 1580
chas@kmelectronics.co.nz

TOKOROA
Adam Ants Electrical
Ph: (07) 886 0536
shop@adamants.co.nz

WHANGAREI
Compass Computers
Ph: (09) 438 8776
info@compasscomputers.co.nz

G069+ G070 Test (1)

(1) How do you understand Project management?

(2) What are the steps in development of Project lifecycle?

(3) What are included in technical meaning of Project management?

(4) Sketch the overview diagram of Project management.

(5) Explain the strategy in Project management with outline diagram

(6) What are the indicators of success & failure of Project?

(7) How will you begin the Project planning?

(8) Sketch the overview diagram of Project Planning.

(9) If there are many steps & activities in the Project, how will you manage to handle?

E071 Test (1)

(1) What are the aspects of electrical contracting?

(2) Write down any ten questions related to sale and marketing aspect before proceeding to electrical contract.

(3) As an electrical contractor, you are required to provide the specification for 11 kV, 2700 crossarm structure on pole with pin insulators.

You need to refer the appropriate technical documents and perform the followings

(a) Sketch the design diagram

(b) Indicate the dimensions

(c) Prepare the material list

(d) Provide the specifications.

(e) Indicate appropriate Australian standard,

(f) Include safety aspect.

(4) What are the steps for purchasing?

(5) How will you manage to keep the tools in your work place?

E001+ E002+ E005+ E007+E008+ E033

Theory Test (1)

E001

(1) what is occupational health & safety?

(2) what is the responsibilitys of employer for work place safety of employees?

(3) what is the responsibility of employee for work place safety?

(4) what is the responsibilitys of government authorities for work place safety?

(5) can electric fire be extinguished with water?

(6) If you have to lift a heavy weight, what will you do?

(7) write down the dangers given by industrial chemicals.

(8) Is possession of electrician licence alone enough for working underground?

(9) How can excessive heat, noise and vibration give disadvantage to human body and how can you protect them?

(10) what will you check before using electrical equipments in work place?

(11) what is danger tagging?

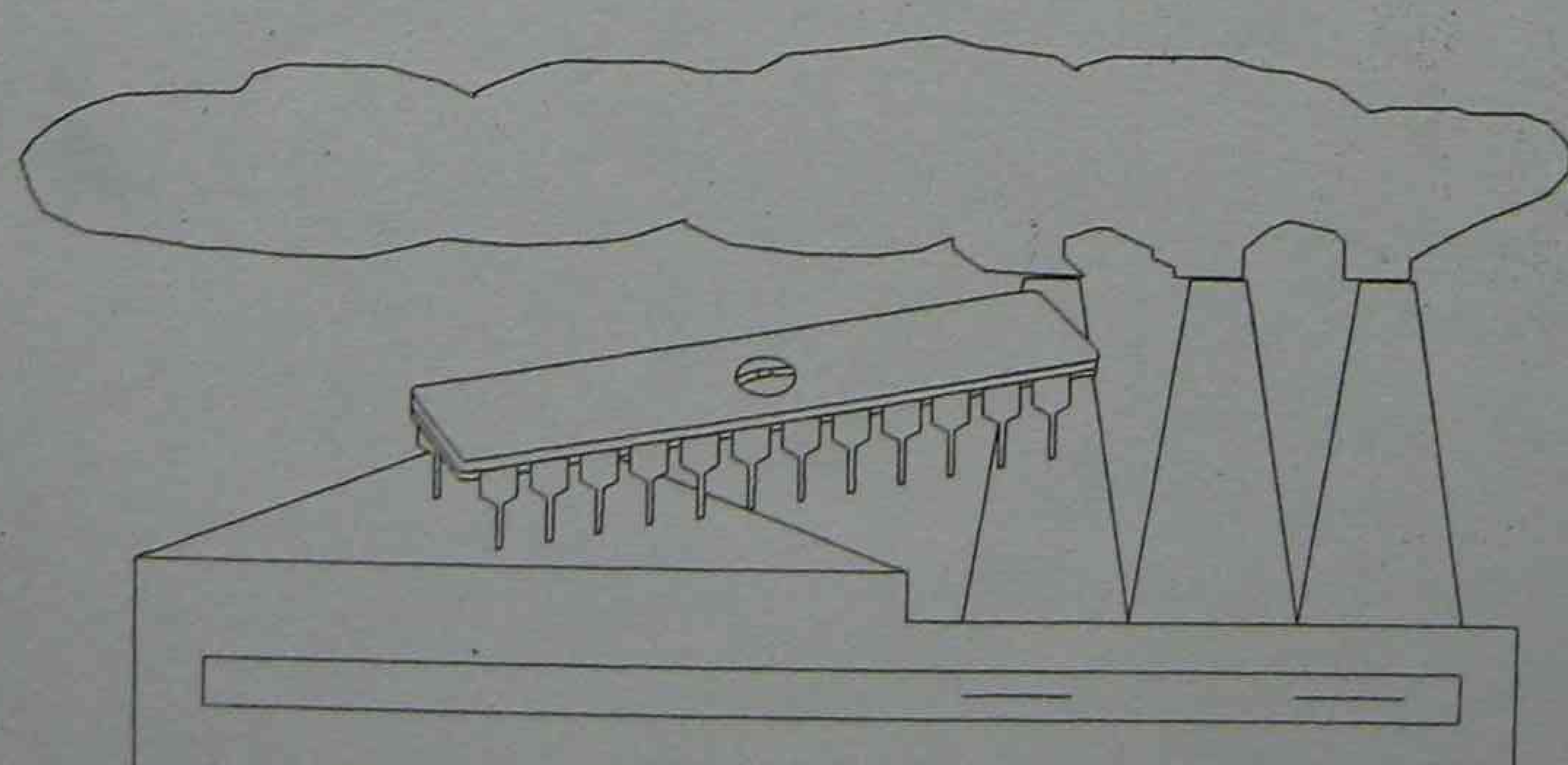
(12) what is D R A B C ?

ADVANCED CERTIFICATE
IN APPLIED INDUSTRIAL ELECTRONICS

OPERATIONAL AMPLIFIERS

SUBJECT NO. 6016C

THEORY-TUTORIAL MANUAL



ADVANCED CERTIFICATE
IN APPLIED INDUSTRIAL ELECTRONICS 6016

OPERATIONAL AMPLIFIERS 6016C

THEORY/TUTORIAL MANUAL

by

Graeme Odgers and Peter Phillips

ACKNOWLEDGEMENTS

This manual is the combined effort of Graeme Odgers and Peter Phillips. Graeme is responsible for most of the content and Peter for editing and producing the document. While the content is original, we gratefully acknowledge the work of George Marges, Peter Harle, Vic Ciscato and others too numerous to mention.

TO THE STUDENT

This manual is a summary and should be read in conjunction with a textbook. While other texts may be suitable, reference is made in these notes to the book Operational Amplifiers and Linear ICs by Coughlin and Driscoll (Prentice Hall). 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 theory component for the subject Operational Amplifiers. The emphasis may vary between teaching centres and locally relevant material might need to be included. Each theory lesson should occupy 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. Manual produced 1992.

CONTENTS

ooooOOOOoooo

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

THEORY LESSON 1

REVISION AND THE DIFFERENTIAL AMPLIFIER

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

(a) ***List an advantage of a direct coupled amplifier.***
(b) ***Calculate the DC voltages of a differential amplifier.***
(c) ***Calculate the differential gain of a differential amplifier for both the unbalanced and balanced output configurations with either an unbalanced or balanced input.***

INTRODUCTION

Many industrial electronic applications require an amplifier that has characteristics as close to the ideal as possible. The ideal characteristics of an amplifier are:

* Infinite gain.
* Infinitely high input impedance.
* Zero output impedance.
* Infinite bandwidth.
* High noise rejection.
* High stability over the range of operation.

A device that has characteristics approaching those of the ideal amplifier is the operational amplifier, or op-amp. To be able to understand the characteristics of an op amp, its internal circuit needs to be examined. The basic building block of an op amp is the ***differential amplifier*** (diff amp), a circuit able to amplify signals from DC to MHz as all connecting stages are ***direct coupled***. The diff amp can also provide high noise rejection (of unwanted signals) and high stability over the range of operation. These notes describe the operation of the differential amplifier in terms of its DC and AC characteristics.

1. DC CONDITIONS OF THE DIFFERENTIAL AMPLIFIER

The differential amplifier is a DC coupled amplifier that amplifies the *difference* between the two input voltages, V_{in1} and V_{in2}.

Fig.1 shows that the circuit of a differential amplifier comprises two common emitter amplifiers, both sharing a single emitter resistor, R_{EE}. The circuit has two outputs, V_{C1} and V_{C2}.

As there are no coupling capacitors this circuit will amplify DC voltages as well as AC voltages.

NOTE: For the purposes of calculating the DC conditions for a differential amplifier, the following approximations are used.

1. For a silicon transistor, $V_{BE} = 0.6$ Volt.
2. Because the base current of a small signal transistor is generally very small, a voltage drop of zero will be assumed across any resistor in series with the base terminal.

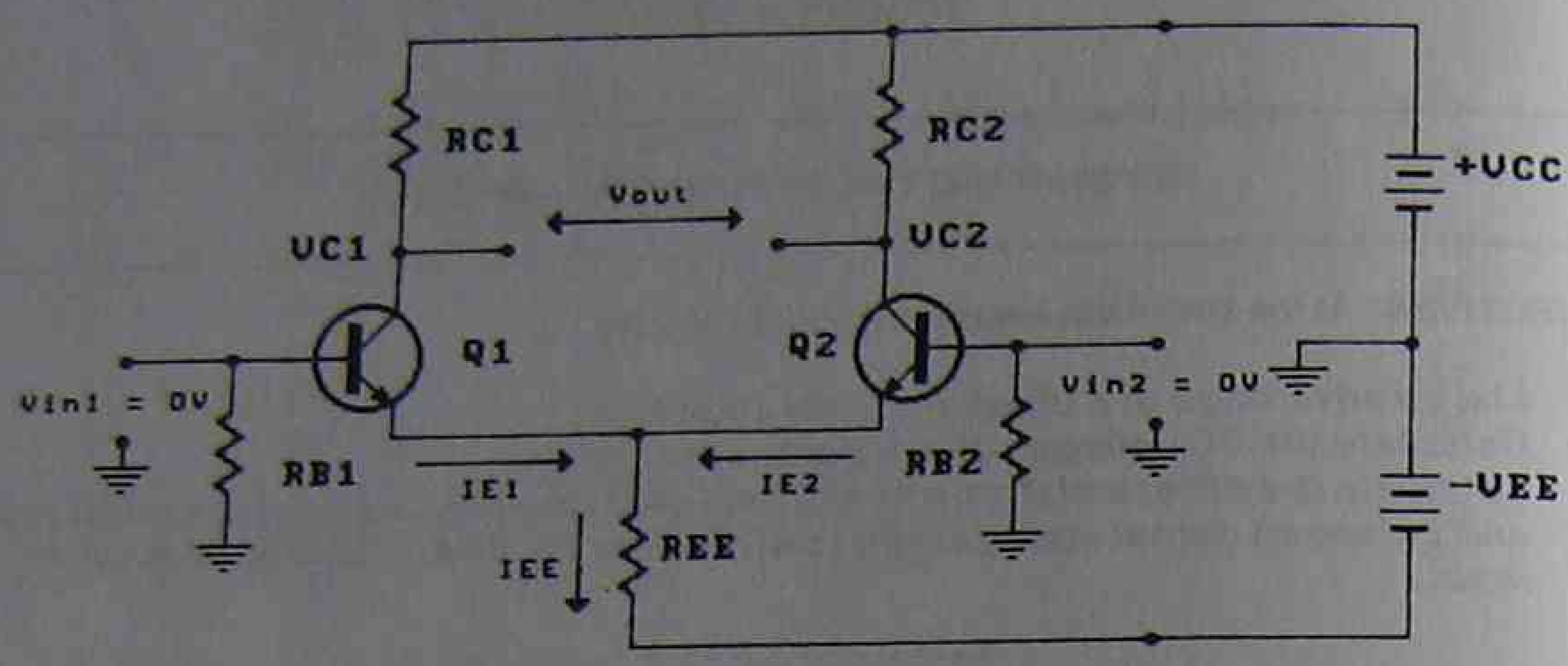


Fig.1: *The Differential Amplifier*

The power supply of Fig.1 has three lines; +VCC, -VEE and ground and is referred to as a dual polarity power supply (generally around +/- 12 to +/- 20 volts).

The DC conditions for Fig.1 are calculated as shown.

1. Assume V_B = 0V for both transistors.

Because the base terminal of the transistors must be +0.6V higher than the emitter terminal, the emitter voltage, with respect to the common (0V) line, is -0.6V.

2. Then V_E = -0.6 V for both transistors

3. The voltage across R_{EE}: $V_{REE} = V_{EE} - 0.6V$ and;

4. $I_{EE} = \frac{V_{REE}}{R_{EE}}$

The emitter currents are approximately equal to the collector currents, as the base currents can be ignored. Since both transistors are matched and have the same DC bias conditions, then:

5. $I_{C1} = I_{C2} = \frac{I_{EE}}{2}$

This allows V_{C1} and V_{C2} to be calculated using:

6. $V_C = V_{CC} - (I_C \times R_C)$

To operate successfully both transistors in a differential amplifier should have identical internal parameters (Beta and V_{BE} should be the same) and both should be operating at the same temperature. If this is not true then a difference will occur at the transistor inputs and this difference will be amplified, varying both V_{C1} and V_{C2}.

The internal parameters of the transistors can be *matched* by manufacturing them as an integrated circuit, such as the LM394 supermatched pair, the specifications of which are included in the practical notes. A feature of this type of device is that Beta is high, (typically 500) resulting in a very small base current and minimal DC loading on a preceding stage.

2. AC OPERATION OF A SINGLE INPUT, UNBALANCED OUTPUT DIFF AMP

This method of circuit connection involves amplifying a signal connected between one input and common, as shown in Fig.2. The circuit still amplifies the difference between the two inputs, V_{in1} and V_{in2}, but V_{in2} is connected to ground (0V).

The difference voltage, V_d (the voltage actually amplified by the circuit) is determined by -

1. $V_d = V_{in1} - V_{in2}$, which gives $V_{in1} - 0$. **Thus:** $V_d = V_{in1}$

The output voltage will be the amplified value of V_{in1}. However, the circuit has two outputs, both V_{C1} and V_{C2}, either of which can be used.

2.1 THE DIFFERENTIAL VOLTAGE GAIN (Ad)

The differential voltage gain (A_d) of a single input, unbalanced output diff amp is found by:

2. $A_d = \frac{R_C}{2r_e}$ (NOTE: $r_e = \frac{30mV}{I_E}$)

The term "unbalanced output" means that each output is referenced to the common line of the circuit. The waveforms associated with this circuit are shown in Fig.2.

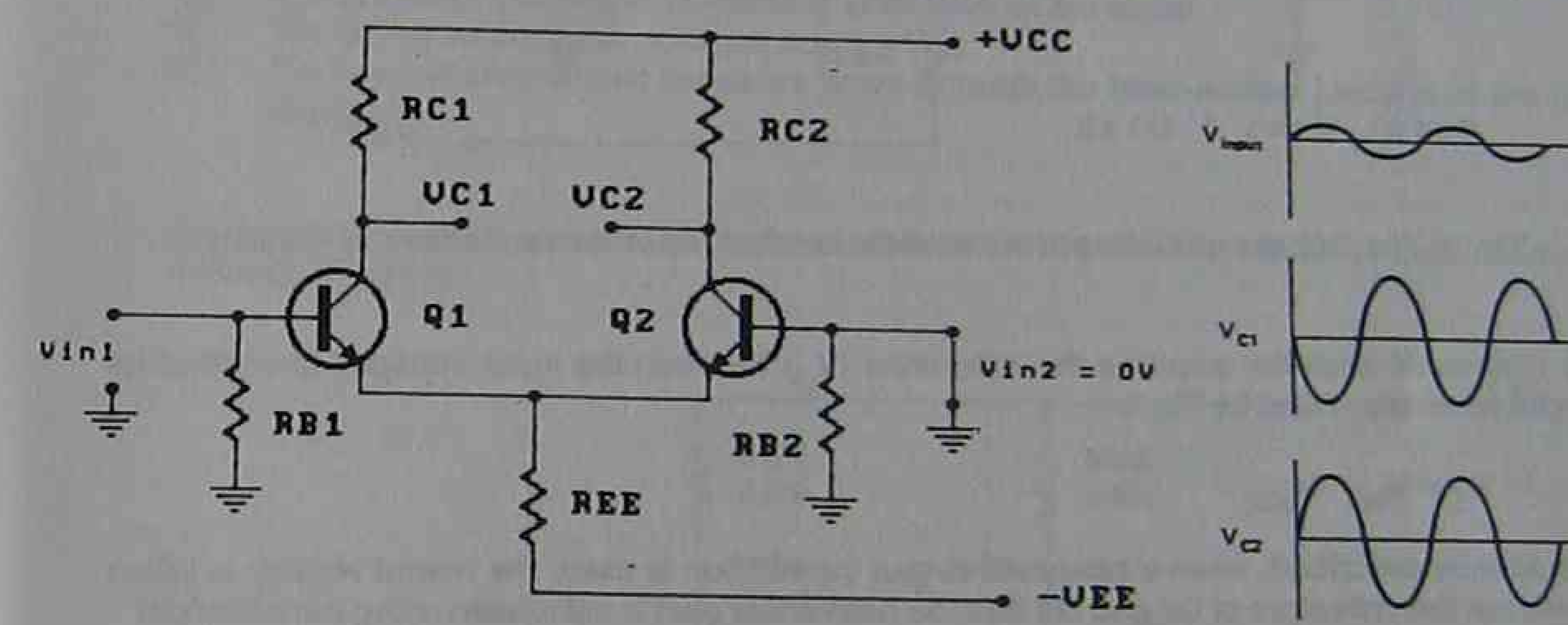


Fig.2: *The Differential Amplifier, Single Input*

2.2 RESPONSE TO A POSITIVE GOING INPUT SIGNAL

* As V_{in1} rises, I_{C1} and the voltage drop across R_{C1} increase causing V_{C1} to fall.
* The rise in I_{C1} also causes V_E to rise making V_{BE2} drop below 0.6V.
* The fall in V_{BE2} reduces I_{C2} by an amount equal to the rise of I_{C1}, keeping I_{EE} constant.
* When I_{C2} falls, a smaller voltage drop occurs across R_{C2}, making V_{C2} rise towards V_{CC}.

2.3 RESPONSE TO A NEGATIVE GOING INPUT SIGNAL

* As V_{in1} falls, I_{C1} decreases causing V_{C1} to rise.
* As I_{EE} is constant, I_{C2} will increase by the same amount that I_{C1} fell, causing V_{C2} to fall.

Note that in both cases V_{C1} is inverted and V_{C2} is in phase when compared to V_{in1}.

The maximum voltage swing at each output equals $+V_{CC}$ to $-V_{EE}$.

3. VOLTAGE GAIN FOR A SINGLE INPUT AND BALANCED OUTPUT DIFF AMP

A differential amplifier with a balanced output has the output voltage taken *between* the two collectors. As already described, these voltages swing in opposite directions, producing a total voltage swing twice that of a single output. The differential voltage gain for a balanced output differential amplifier is:

$$A_d = \frac{R_C}{r_e}$$

4. VOLTAGE GAIN FOR THE BALANCED INPUT, BALANCED OUTPUT DIFF AMP

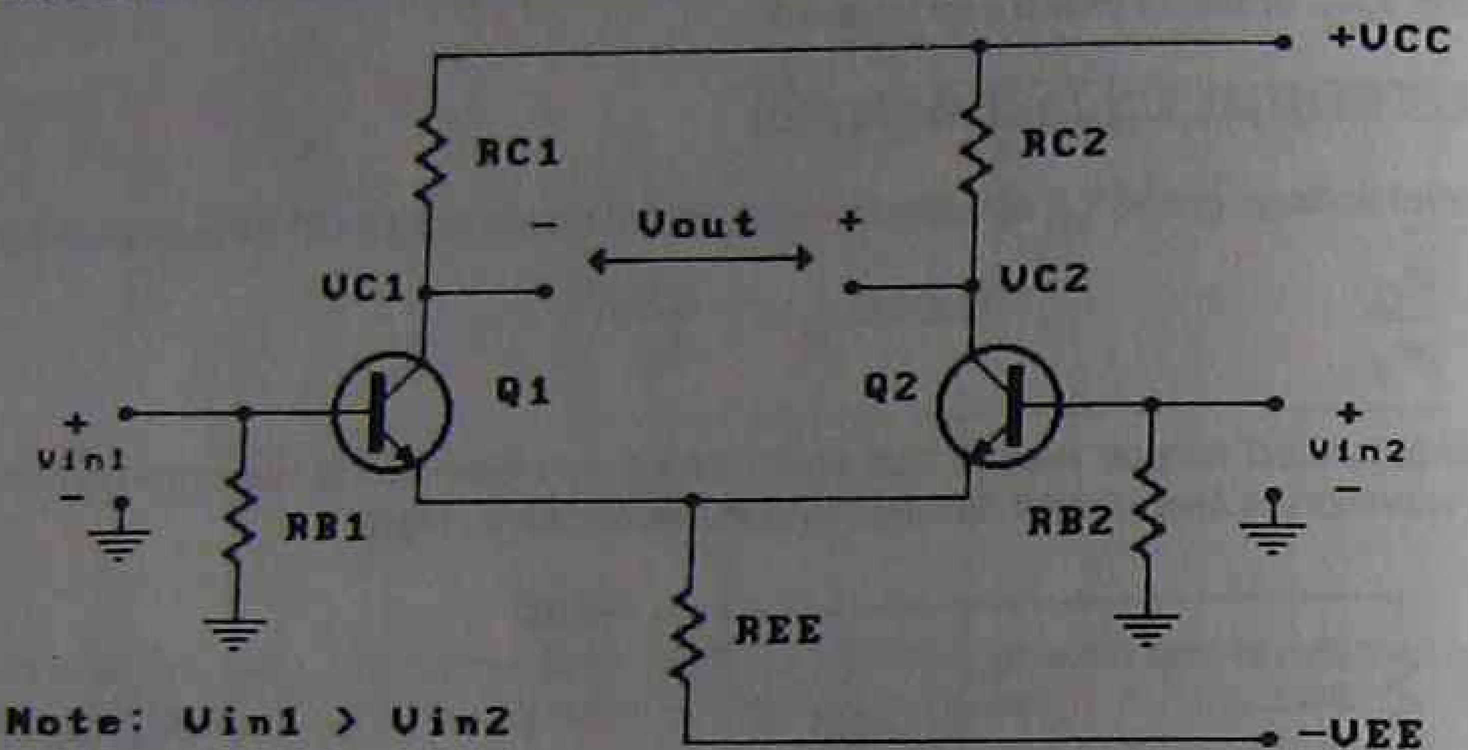


Fig.3. *The Differential Amplifier with Balanced Input and a Balanced Output*

A differential amplifier amplifies the difference (V_d) between the input voltages present at its input terminals. Thus, for Fig.3:

1. $V_d = V_{in1} - V_{in2}$

As already described, when a balanced output connection is used, the output voltage is taken between the collectors of Q1 and Q2 and the differential gain is calculated using the equation:

2. $A_d = \frac{R_C}{r_e}$ *(same equation as in Part 3)*

The output voltage equals $V_d \times A_d$.

Note the signal polarities shown in Fig.3, which shows that the differential input voltage is inverted at V_{C1} and in phase at V_{C2}.

5. INPUT AND OUTPUT IMPEDANCES OF A DIFF AMP:

The input resistance any differential amplifier is calculated using the equation:

$$r_{in} = 2\beta r_e$$

The output resistance (r_o) of the unbalanced circuit is R_C.

Neither of these impedances equal the ideal, but the input impedance could be increased substantially by replacing the two BJT devices with either JFETs or MOSFETs.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

TUTORIAL WEEK 1

1. A differential amplifier with a balanced-input, balanced output has:

   (a) Two inputs and one output referenced to the common rail.
   (b) One input and one output referenced to the common rail.
   (c) Two inputs and two outputs, usually not referenced to the common rail.
   (d) One input and two outputs referenced to the common rail.

2. In a discrete component BJT differential amplifier powered with a dual polarity power supply:

   (a) The base voltage at both transistors is 0.6V.
   (b) The quiescent DC emitter currents of both devices are equal.
   (c) The quiescent collector voltages equal +Vcc.
   (d) The base current of one transistor flows through the base-emitter junction of the other transistor.

3. Determine the voltages present at the base, collector and emitter terminals for both transistors in the circuit of Fig.1.

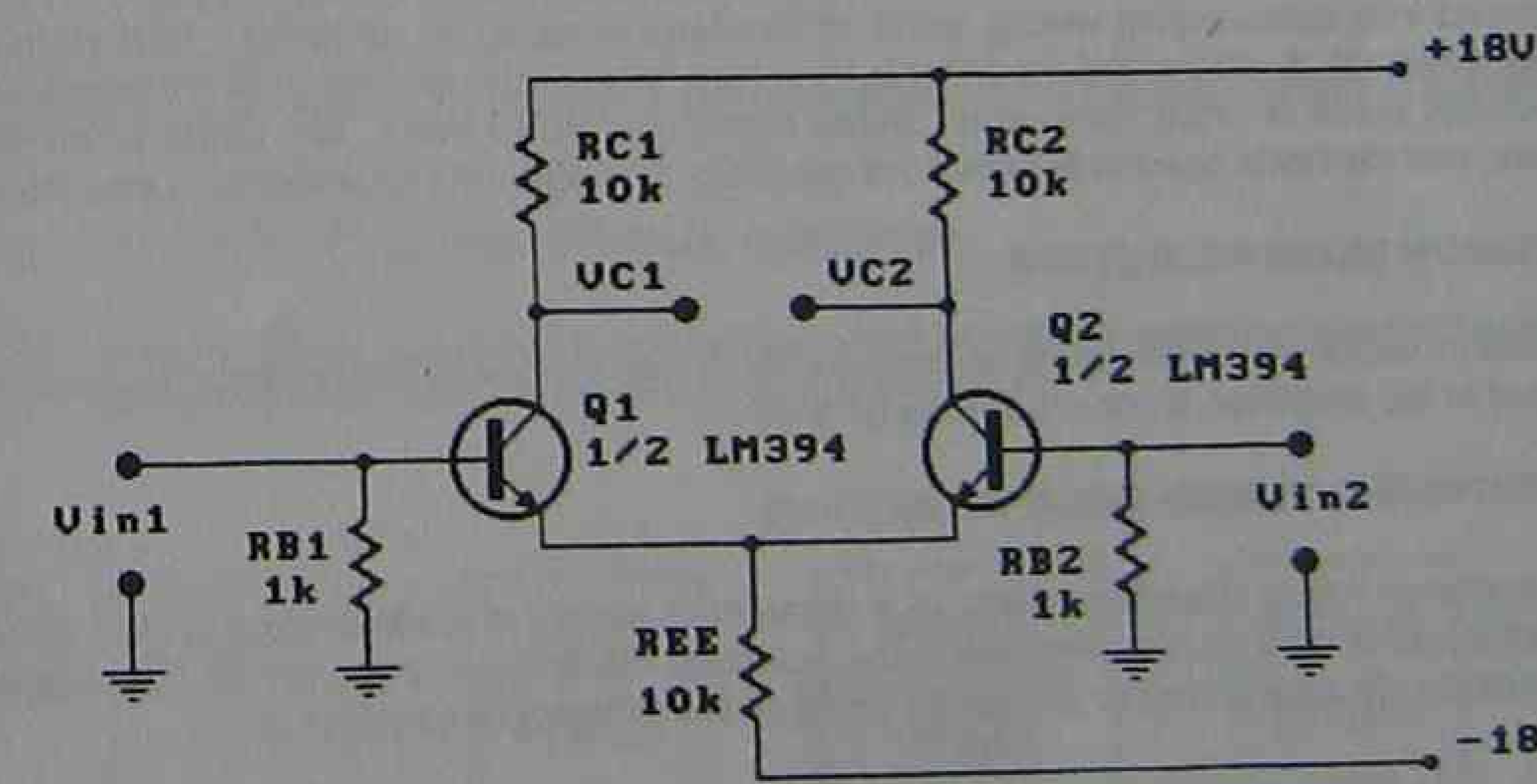


Fig.1

4. Given that r_e equals 29Ω for both transistors in Fig.1, calculate the differential voltage gain for:

   (a) An unbalanced output.
   (b) A balanced output.

****

3. VOLTAGE GAIN FOR A SINGLE INPUT AND BALANCED OUTPUT DIFF AMP

A differential amplifier with a balanced output has the output voltage taken *between* the two collectors. As already described, these voltages swing in opposite directions, producing a total voltage swing twice that of a single output. The differential voltage gain for a balanced output differential amplifier is:

$$A_d = \frac{R_C}{r_e}$$

4. VOLTAGE GAIN FOR THE BALANCED INPUT, BALANCED OUTPUT DIFF AMP

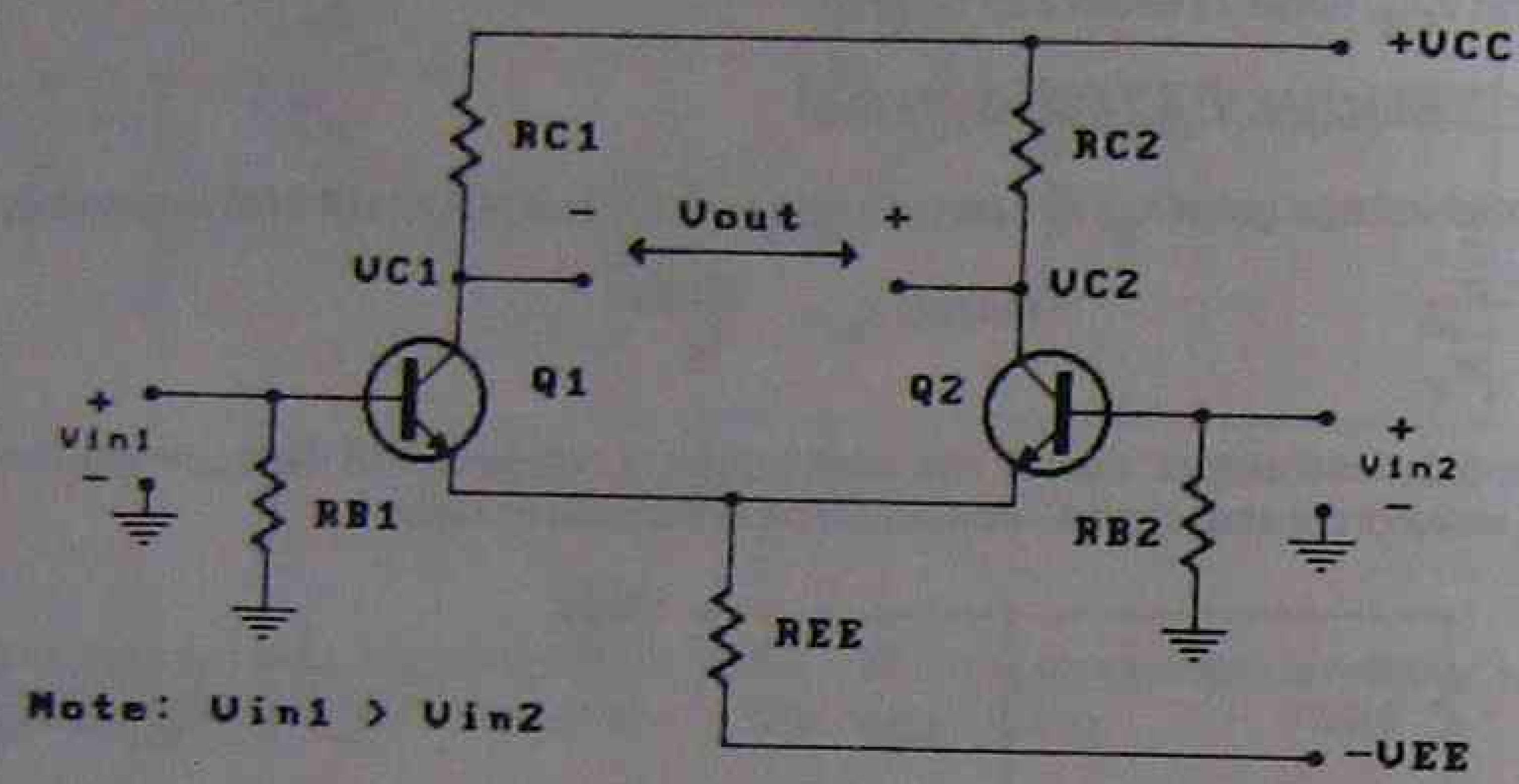


Fig.3. *The Differential Amplifier with Balanced Input and a Balanced Output*

A differential amplifier amplifies the difference (V_d) between the input voltages present at its input terminals. Thus, for Fig.3:

1. $V_d = V_{in1} - V_{in2}$

As already described, when a balanced output connection is used, the output voltage is taken between the collectors of Q1 and Q2 and the differential gain is calculated using the equation:

2. $A_d = \dfrac{R_C}{r_e}$ *(same equation as in Part 3)*

The output voltage equals $V_d \times A_d$.

Note the signal polarities shown in Fig.3, which shows that the differential input voltage is inverted at V_{C1} and in phase at V_{C2}.

5. INPUT AND OUTPUT IMPEDANCES OF A DIFF AMP:

The input resistance any differential amplifier is calculated using the equation:

$$r_{in} = 2\beta r_e$$

The output resistance (r_o) of the unbalanced circuit is R_C.

Neither of these impedances equal the ideal, but the input impedance could be increased substantially by replacing the two BJT devices with either JFETs or MOSFETs.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

TUTORIAL WEEK 1

---

1. A differential amplifier with a balanced-input, balanced output has:

(a) Two inputs and one output referenced to the common rail.
(b) One input and one output referenced to the common rail.
(c) Two inputs and two outputs, usually not referenced to the common rail.
(d) One input and two outputs referenced to the common rail.

2. In a discrete component BJT differential amplifier powered with a dual polarity power supply:

(a) The base voltage at both transistors is 0.6V.
(b) The quiescent DC emitter currents of both devices are equal.
(c) The quiescent collector voltages equal +Vcc.
(d) The base current of one transistor flows through the base-emitter junction of the other transistor.

3. Determine the voltages present at the base, collector and emitter terminals for both transistors in the circuit of Fig.1.

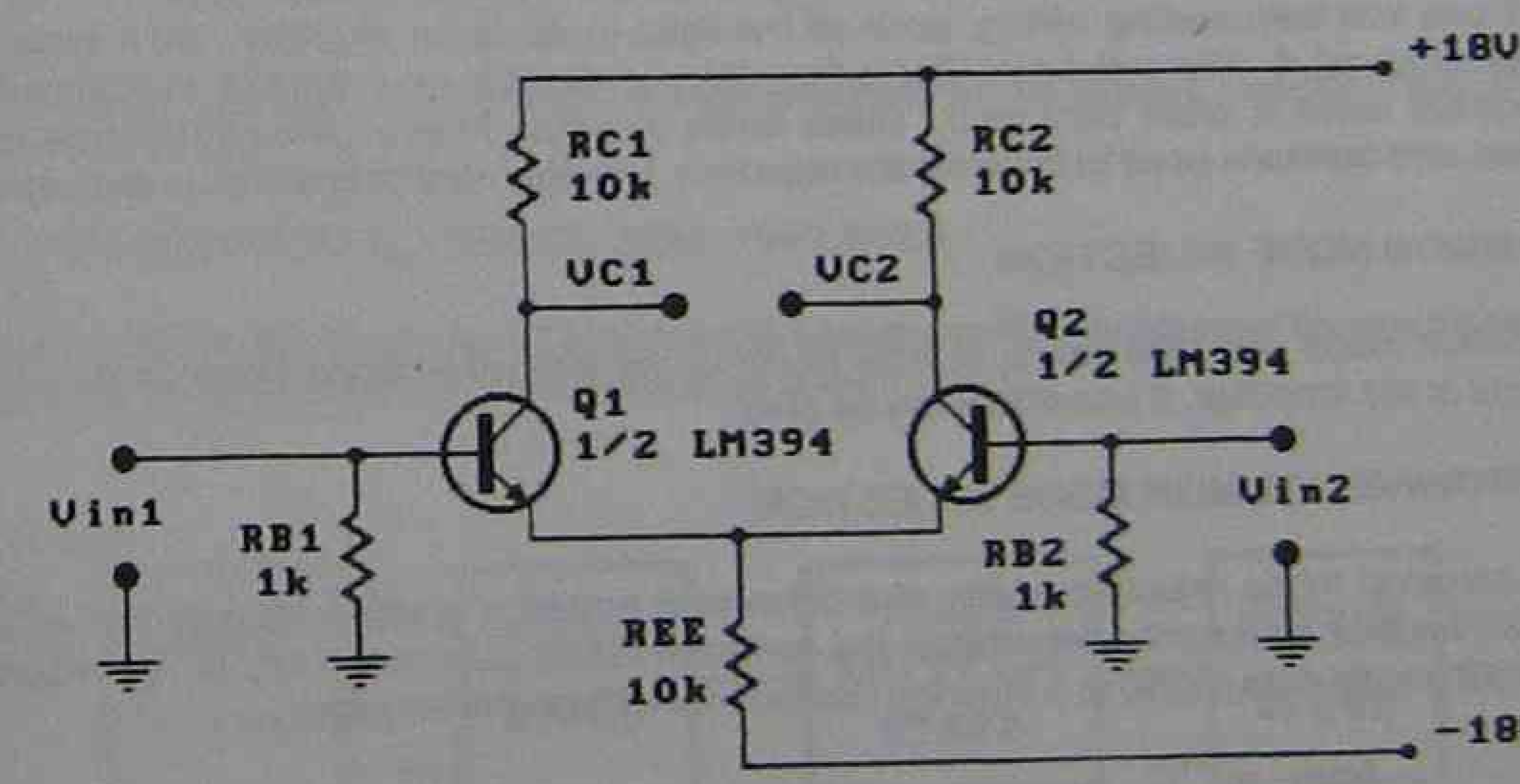


Fig.1

4. Given that r_e equals 29Ω for both transistors in Fig.1, calculate the differential voltage gain for:

(a) An unbalanced output.
(b) A balanced output.

****

THEORY LESSON 2

NOISE REJECTION AND INTRODUCTION TO THE OPERATIONAL AMPLIFIER

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

(a) *Define the term 'common mode rejection ratio' as applied to a differential amplifier, and calculate its value given the differential gain and the common mode gain.*
(b) *Identify the role of a constant current source in a differential amplifier circuit, and calculate the DC current flowing in a typical discrete component, BJT constant current source.*
(c) *Draw the symbol and internal block diagram of an operational amplifier.*
(d) *Describe the nature of the output signal of an operational amplifier that is operating from a dual polarity power supply.*

INTRODUCTION

The differential amplifier described previously has the significant advantage compared to conventional amplifiers in that it can reject noise signals that are common to both inputs. These noise signals are referred to as a *common mode input signal.* Noise signals occur as a result of electrical switching, and are produced, for example, by fluorescent lights, radio transmitters, welding machines, oscillator circuits and vehicle ignition systems. The noise is induced into surrounding wiring, such as the input leads to an amplifier, often swamping the original signal. A differential amplifier is therefore essential in an industrial environment, where electrical noise is often very high. These notes describe how a differential amplifier rejects noise, and methods used to improve the rejection. The operational amplifier is also introduced.

1. COMMON MODE REJECTION

Common mode rejection refers to how much of the common mode signal is present in the output of the amplifier. It should ideally be zero.

2. IMPROVING COMMON MODE REJECTION

The common mode rejection ability of a differential amplifier is dependent on the value of the emitter resistor as it partly determines the common mode gain of the circuit. The equation for common mode gain (Acm) in a discrete component differential amplifier is:

$$Acm = \frac{R_C}{2R_{EE}}$$

The equation shows that Acm depends on the value of the emitter resistor REE. The larger this resistance to changes in the emitter potential then the smaller the common mode gain, which ideally should be zero. A small common mode gain causes the common mode rejection ratio (CMRR) to be large (ideally it should be infinitely high). The equation to determine CMRR is:

$$CMRR = \frac{Ad}{Acm}$$

Because a large value of emitter resistor is impracticable, a *constant current source* is often used in its place, as a high resistance with the required DC current flow is obtained. This gives a low value of Acm, with a correspondingly high value of CMRR.

3. THE DISCRETE COMPONENT BJT CONSTANT CURRENT SOURCE

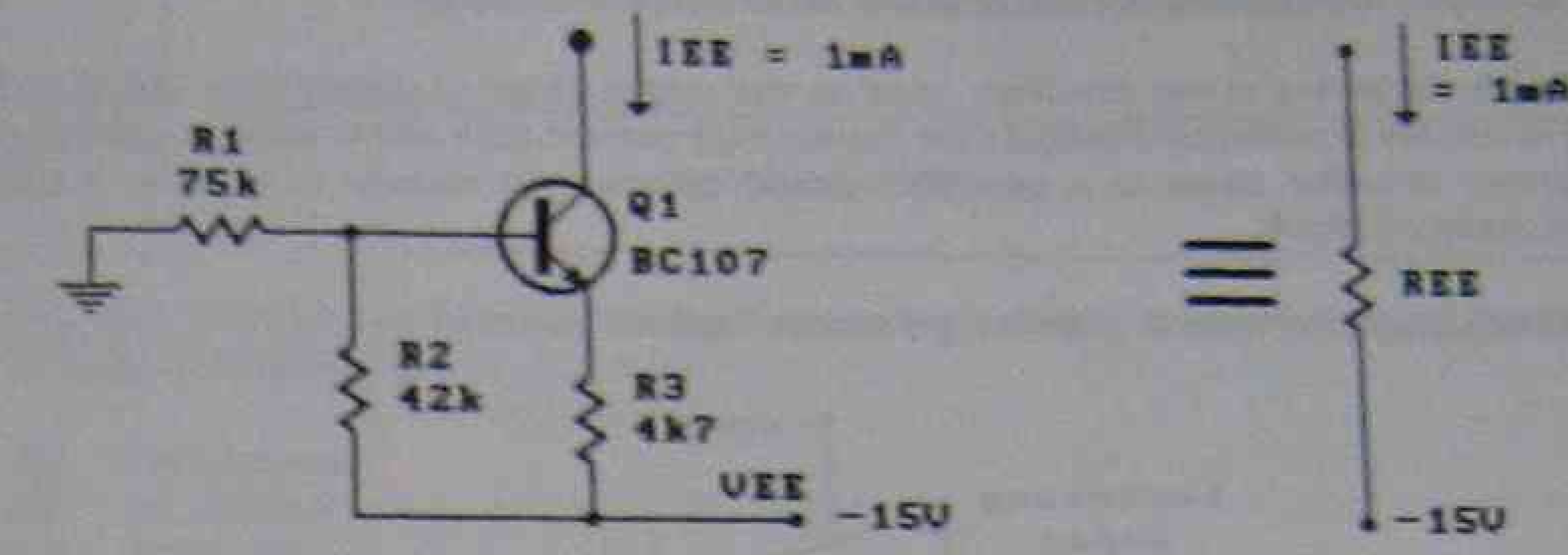


Fig.1: *The basic discrete component BJT constant current source*

The current considered as being constant in the differential amplifier is the emitter current, and a constant current source therefore passes this current when used as a replacement for the emitter resistor in a diff amp. The circuit of Fig.1, shows a voltage stabilised BJT constant current circuit. The base voltage fixes the emitter voltage which in turn establishes the current flowing in the circuit. The DC equations for this circuit are:

1. $V_{R2} = \left(\frac{R2}{R1 + R2}\right) \times V_{EE} = 5.38V$ (where $V_{EE} = -15V$)
2. $V_{R3} = V_{R2} - 0.6V = 4.78V$
3. $I_C = I_{EE}$, which equals $\frac{V_{R3}}{R_3} = 1mA$

Resistor R2 can be replaced by diodes or zeners to improve the thermal stability of the circuit.

4. INTRODUCTION TO THE OPERATIONAL AMPLIFIER

An operational amplifier, (op amp) is a direct-coupled amplifier containing approximately four stages, all contained in an integrated circuit, as shown in Fig.2.

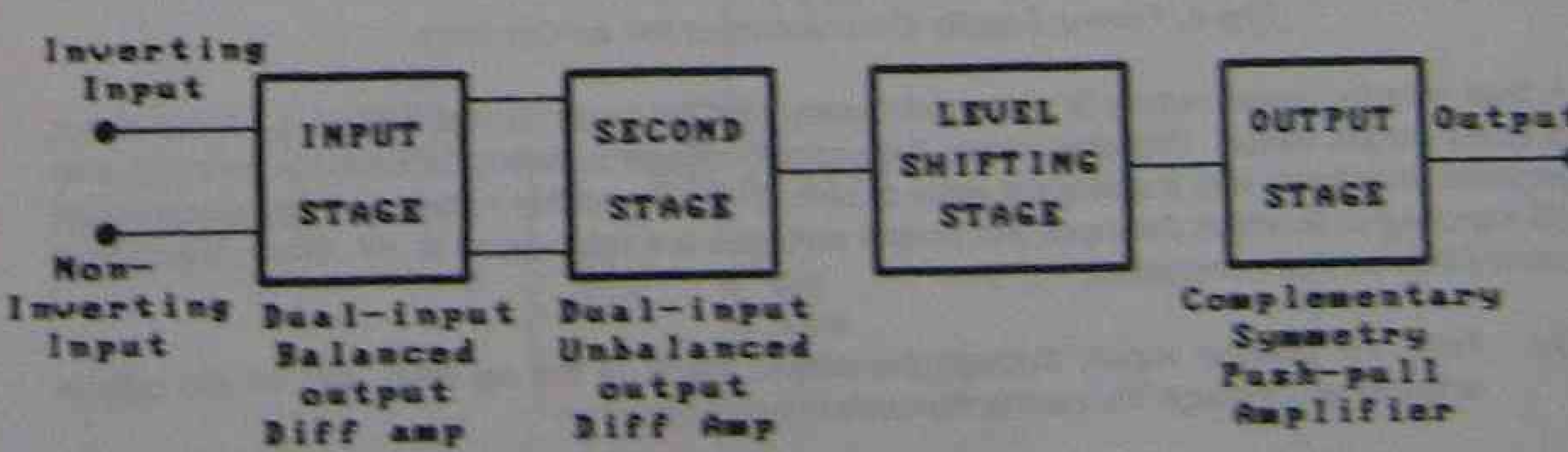


Fig.2: *Block diagram of an operational amplifier.*

As shown in Fig.2, the first stage of an op amp is a differential amplifier with a balanced output, giving a high common mode rejection. The second stage is also a diff amp, but with an unbalanced output. These two stages provide good noise rejection and a high voltage gain.

The third stage is a DC level shifter and is used to buffer the high DC output level of the second differential amplifier and change it to a lower DC level. Biasing of the DC level shifter is used to set the output voltage level half way between the two supply voltages.

The fourth stage is a power amplifier, used as the output stage to supply load current. This stage is usually a push-pull configuration having high current gain and a voltage gain of one. The power amplifier stage of a general purpose op-amp can usually only drive loads of approximately 500mW.

The symbol for an operational amplifier is shown in Fig.3.

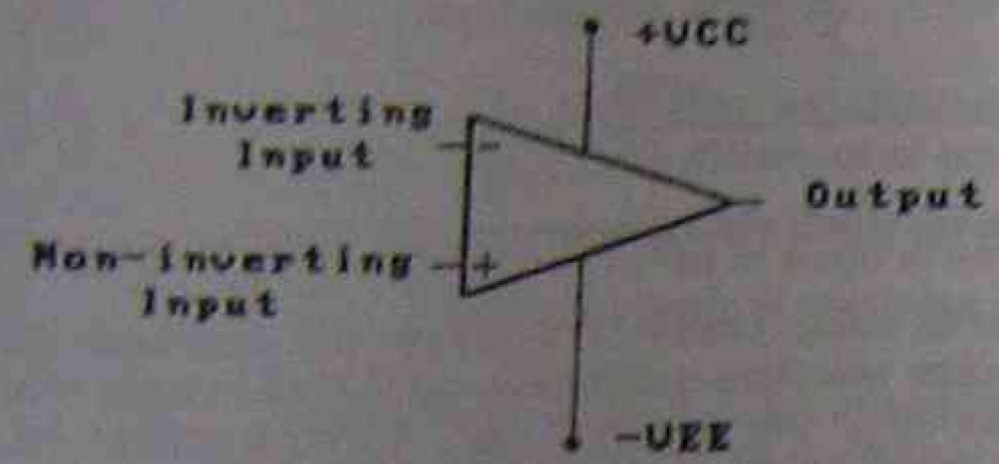


Fig.3: *The symbol for an Operational Amplifier.*

The op amp symbol shows two inputs, labelled as the *inverting* input and the *non-inverting* input. The power output stage connects to the output terminal. The symbol A_{Vol} is the open-loop voltage gain, generally in the range of 200,000 and higher.

5. POWER SUPPLY CONNECTIONS FOR AN OP AMP

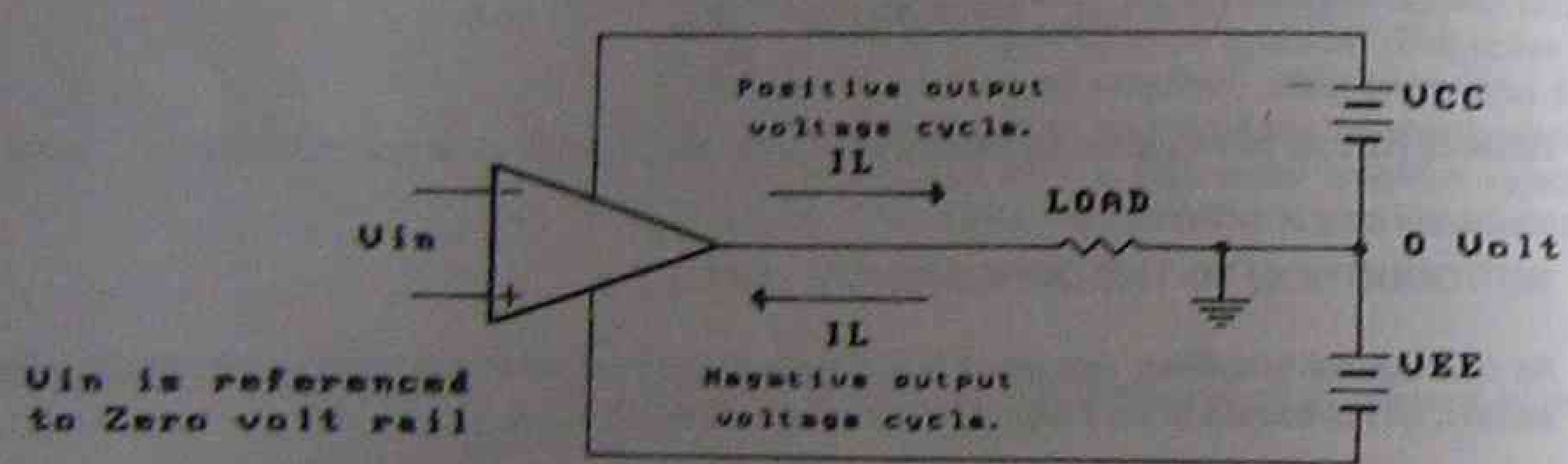


Fig.4. *Power Supply Connections for an Op Amp.*

A dual polarity power supply is commonly used with op amps to make biasing of the direct coupled stages easier. This also allows the output voltage to swing in both the positive and negative directions, with a quiescent value of zero. Fig.4 shows the output voltage swing for an AC input signal in which the input and output voltages are referenced to 0V. Thus, the output current flow can take two paths:

(a) From the positive supply through the output stage of the op amp, out of the output terminal and through the load to the common rail (0V).

(b) From the common rail, through the load, into the output of the op amp, then through the power amplifier to the negative DC supply.

A dual polarity power supply allows an op amp to supply a load voltage of virtually 2Vcc, swinging equally around a potential of zero volts.

****

TUTORIAL WEEK 2

1. The phase difference between the two output signals of a single input (unbalanced input) differential amplifier is:

(a) 180 degrees
(b) 90 degrees.
(c) 0 degrees.

2. The phase difference between the 'common-mode' signals present at both inputs of a balanced input differential amplifier is:

(a) 180 degrees.
(b) 90 degrees.
(c) 0 degrees.

3. A constant current source in the emitter circuit of a differential amplifier is used to:

(a) Increase differential voltage gain.
(b) Increase the frequency response.
(c) Decrease the common mode gain.
(d) Decrease the common mode rejection ratio.

4. For the circuit of Fig.1, calculate:

(a) The voltage across R1.
(b) The voltage across R3.
(c) The current I_{EE}
(d) The current that would flow in each transistor of a differential amplifier if Fig.1 was connected as a constant current source in the emitter circuit of a differential amplifier.

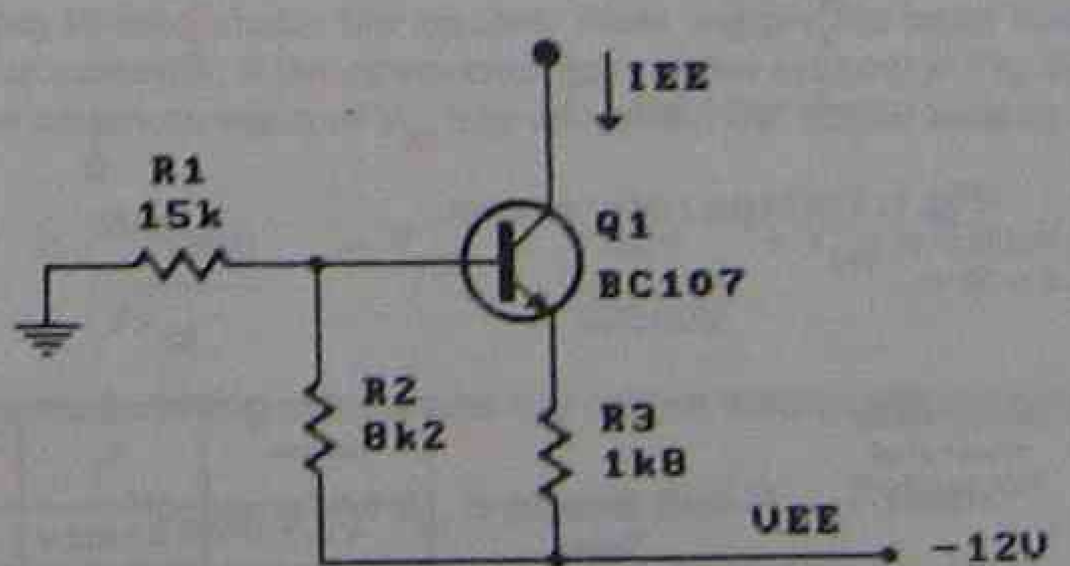


Fig.1.

5. Draw the internal block diagram of an operational amplifier, and show the current paths that can flow in the load, assuming the amplifier is powered with a dual polarity power supply.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS, 6016C

THEORY LESSON 3

COMPARATORS

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

(a) *Identify the need for a reference voltage with a voltage comparator circuit.*
(b) *Sketch the output waveform of an inverting and a non-inverting comparator.*
(c) *Calculate the threshold voltages and sketch the output waveform of a comparator with voltage hysteresis.*
(d) *Describe the operation of a window comparator.*

INTRODUCTION

In industry it is often necessary to compare an unknown quantity to a reference. Once a comparison is made a control circuit can indicate whether a measured quantity is higher, equal to or less than the prescribed reference. An example would be set-point control, whereby an oven is kept within a set temperature band, say 350°C ± 5°C. The measured temperature is converted to a voltage and compared to a voltage reference. Op amp comparators and a reference voltage of some sort form the building blocks of such measurement.

1. DERIVING REFERENCE VOLTAGES

If a comparator is being used to compare an input signal to a set point value, a stable reference voltage is required to represent the set point value. Such a reference can be produced either with voltage dividers, as shown in Fig.1(a) and (b), or with an IC designed for the purpose, as shown in Fig.1(c). The device shown is an Analog Devices, programmable precision positive voltage reference IC, type AD584. This IC is capable of maintaining a very stable output voltage, which can be programmed to +2.500V, +5.000V, +7.500V or 10.000V.

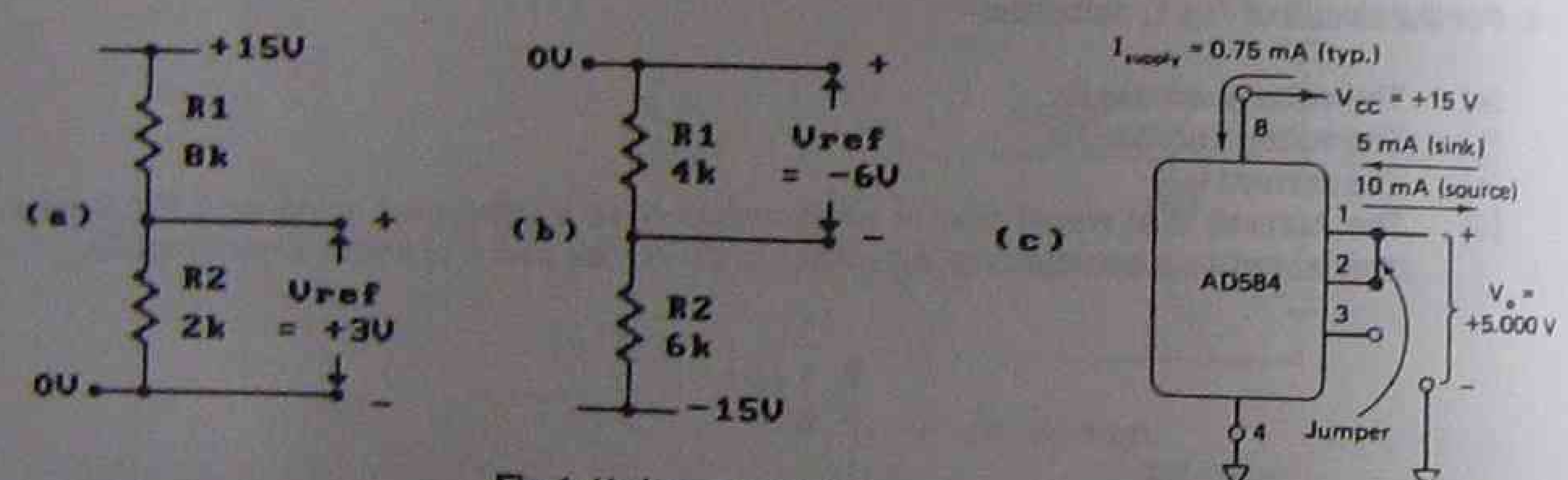
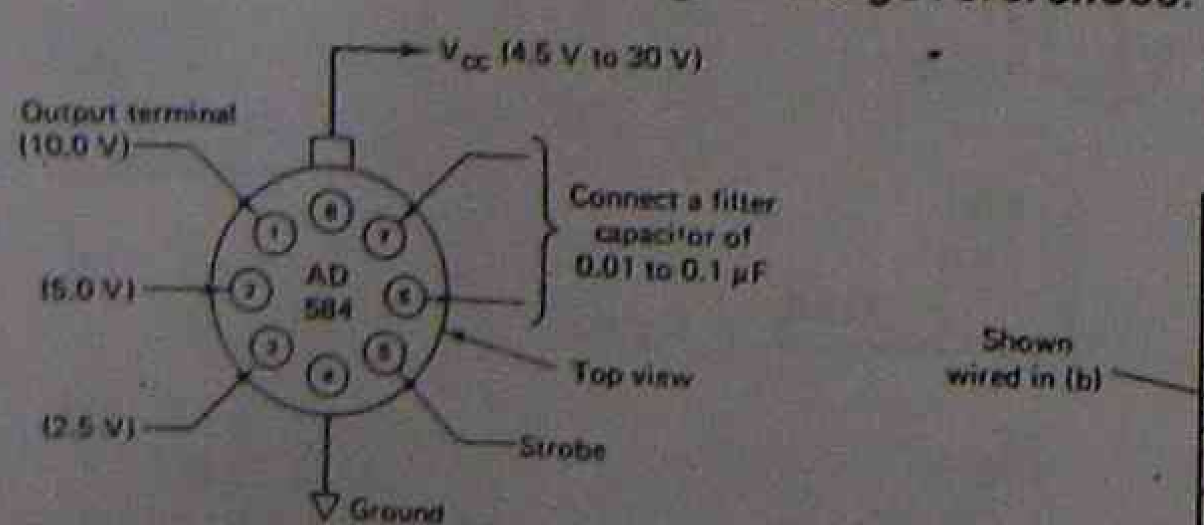


Fig.1: *Voltage references.*



Jumper	V_o	Output voltage tolerance
1 to 3	+2.500 V	±7.5 mV
1 to 2	+5.000 V	±15 mV
2 to 3	+7.500 V	±20 mV
No connection	+10.000 V	±30 mV

Fig.2: *The AD584 pin programmable voltage reference.*

Fig.2 shows the manufacturers' information for programming the reference voltage of the AD584. Linking the pins as per the table sets the output voltage level.

2. THE BASIC COMPARATOR

An op amp on its own can be wired as a comparator. A non-inverting comparator has the inverting input connected to a voltage reference and the non-inverting input as the sensing input, as shown in Fig.3(a). The output voltage will depend on whether the input voltage is above or below the reference voltage. The inverting comparator has the opposite connections. A comparator operates in *OPEN LOOP* and the voltage gain of the circuit is the open-loop gain of the op amp. Typically an op amp has an open loop gain of over 100,000 giving:

$$V_{out} = Av_{ol} \times V_{in}$$

Because an op amp comparator has such a high open-loop gain, an input voltage will cause the output voltage to be either maximum positive ($+Vo_{sat}$) or maximum negative ($-Vo_{sat}$) as summarised below:

NON-INVERTING COMPARATOR

$+Vo_{(sat)}$ when V_{in} exceeds V_{ref}

and $-Vo_{(sat)}$ when V_{in} is less than V_{ref}

INVERTING COMPARATOR

$-Vo_{(sat)}$ when V_{in} exceeds V_{ref}

$+Vo_{(sat)}$ when V_{in} is less than V_{ref}

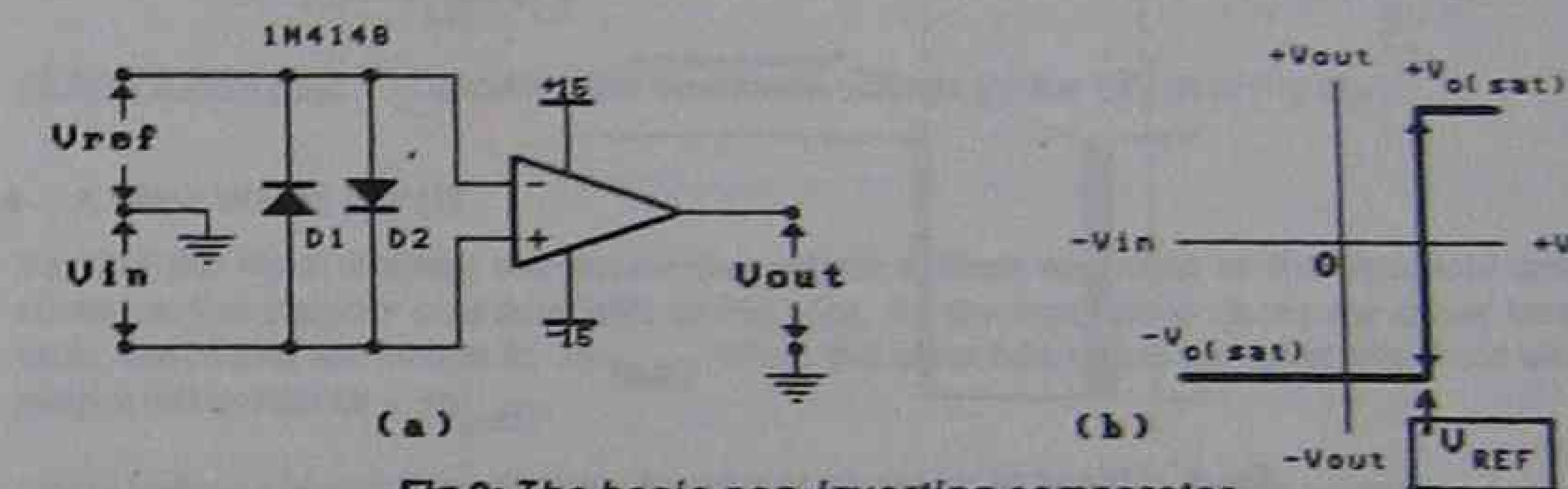


Fig.3: *The basic non-inverting comparator.*

The diodes in Fig.3(a) protect the input against overload due to a high differential input voltage by limiting the differential input voltage to ±0.6V.

2.1 OPEN LOOP GAIN AND Vin

For switching to take place, the op amp must amplify the input voltage to cause the output to saturate. For example, if the open-loop gain of the op amp in Fig.3(a) is 200,000 and $\pm Vo_{(sat)} = \pm 15V$, the minimum value of V_{in} that will cause the output level to change is:

$$V_{in} = V_{ref} \pm \frac{2Vo_{(sat)}}{Av_{ol}} = V_{ref} \pm \frac{30V}{200000} = V_{ref} \pm 150\mu V$$

Thus for the noninverting comparator the output voltage will change to:

$+Vo_{(sat)}$ when V_{in} is greater than $V_{ref} + 150\mu V$

and $-Vo_{(sat)}$ when V_{in} is less than $V_{ref} - 150\mu V$

2.2 CIRCUIT TRANSFER CHARACTERISTICS

Fig.3(b) shows the transfer curve for the changes that occur at the output of a non-inverting comparator as the input varies around the reference voltage, (or switching threshold level). As V_{in} rises above V_{ref}, the output voltage goes to $+Vo_{(sat)}$.

Similarly, if the input falls below the reference voltage, the output swings to $+Vo_{(sat)}$. Thus the output of a comparator is always either at $+Vo_{(sat)}$ or $-Vo_{(sat)}$.

2.3 EFFECT OF SLEW RATE

Slew rate refers to how fast the output of an op amp can change and is measured in volts change per unit time, usually expressed as V/μs. In the example the output voltage change is 30 volts (from -15V to +15V). An op amp with a slew rate of 0.5V/μs will take (30/0.5)μs (= 60μs) for the output to swing from one level to another. For best performance, an op amp designed to perform the function of a comparator should be used, such as the LM311 device.

2.4 LIMITATIONS OF THE BASIC COMPARATOR CIRCUIT

The basic limitation of a comparator such as that shown in Fig.3(a) is that small variations of the input voltage (around ±150μV) can lead to false switching when the input voltage is close to the reference voltage. Input noise voltages can easily exceed this level, giving an output as shown in Fig.4. This problem can be overcome by using a *Schmitt trigger* circuit, which uses *voltage hysteresis* to give reliable switching.

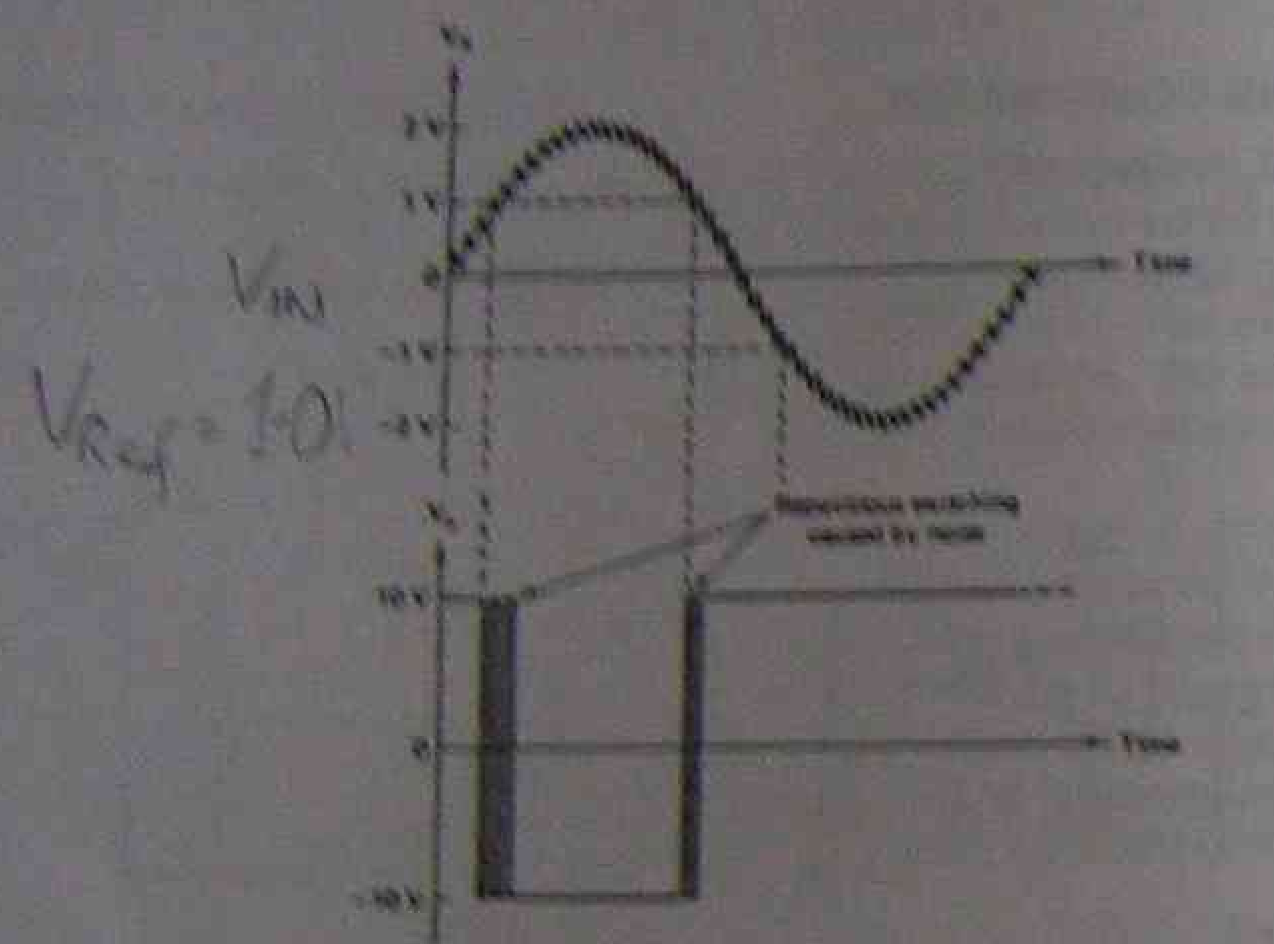


Fig.4: *Effect of noise in the input waveform.*

3. THE DIFFERENTIAL COMPARATOR WITH VOLTAGE HYSTERESIS

The circuit of Fig.5(a) is an inverting comparator with *POSITIVE FEEDBACK* applied by taking a fraction of the output voltage and feeding it back to the non-inverting terminal. This will speed up the switching transition by creating two switching threshold voltages, shown on the transfer curve of Fig.5(b), labelled as V_{LT} (lower threshold) and V_{UT} (upper threshold).

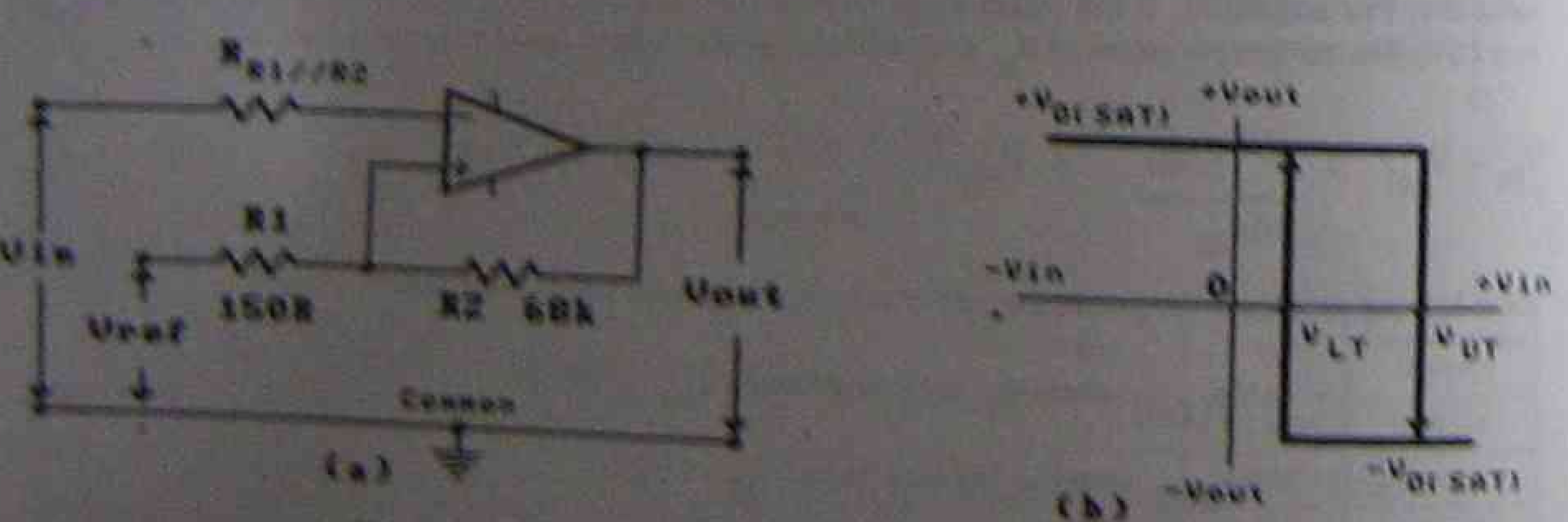


Fig.5: *Inverting comparator with hysteresis.*

The threshold voltages are those values at which switching occurs, and are different because the reference voltage at the non-inverting terminal varies, depending on whether the output voltage is either maximum positive or maximum negative. The output voltage is reduced by R1 and R2, and is added to the reference voltage to create the two threshold voltages. Circuits that use positive feedback of this type are referred to as *Schmitt triggers*.

3.1 THE UPPER THRESHOLD VOLTAGE

The upper threshold voltage (V_{UT}) for Fig.5(a) is calculated thus:

$$V_{UT} = V_{ref} + \left[[+Vo_{(sat)} - V_{ref}] \times \frac{R1}{R1 + R2} \right]$$

CLASS EXERCISE: Calculate the upper threshold voltage for the circuit of Fig.5(a).

3.2 THE LOWER THRESHOLD VOLTAGE

The lower threshold voltage (V_{LT}) for Fig.5(a) is calculated thus:

$$V_{LT} = V_{ref} + \left[[-Vo_{(sat)} - V_{ref}] \times \frac{R1}{R1 + R2} \right]$$

CLASS EXERCISE: Calculate the lower threshold voltage for the circuit of Fig.5(a).

3.3 THE HYSTERESIS VOLTAGE

The hysteresis voltage (V_H) is the difference between the two threshold voltages.

$$V_H = V_{UT} - V_{LT}$$

CLASS EXERCISE: Calculate the hysteresis voltage for the circuit of Fig.5(a).

3.4 A SINE WAVE INPUT

Varying the input voltage will cause the output voltage to switch at the threshold levels as shown in the transfer characteristic of Fig.5(b). As the input rises above the upper threshold level, the output will switch to $-Vo_{(sat)}$. When the input falls below the lower threshold level, the output will switch to $+Vo_{(sat)}$.

Noise voltages that fall between the two threshold levels will not be large enough to cause false switching of the output voltage, as illustrated in Fig.6.

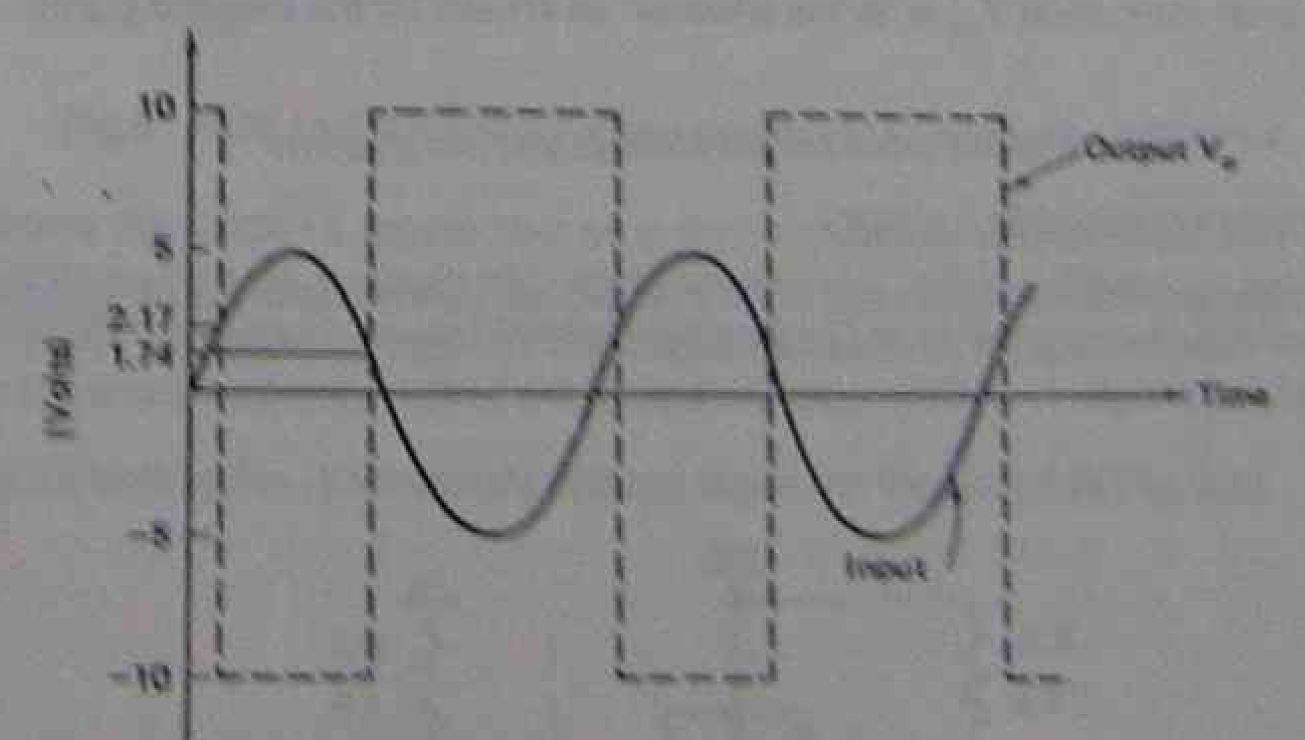


Fig.6: *Output waveform of a Schmitt comparator*

4. THE WINDOW COMPARATOR

A window comparator will detect when an input voltage is between two set levels, or within a *window*. As shown in Fig.7, two reference voltages are required to set the limits for the width of the voltage window. In Fig.7, if the input voltage is between the limits set by V_{UT} and V_{LT}, a LED is turned on by the action of Q1 turning off.

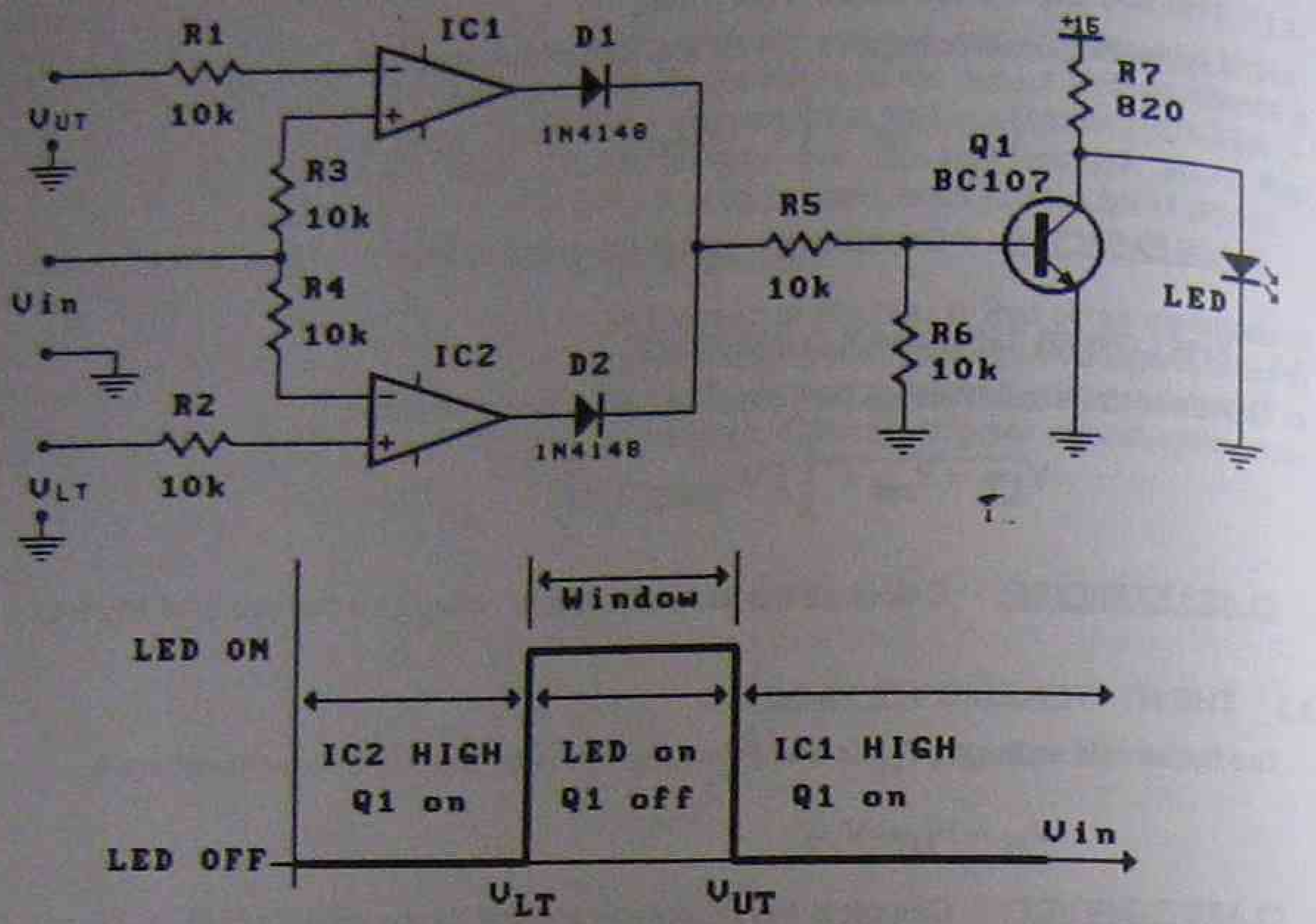


Fig.7: *The window comparator*

For Fig.7:

(a) When V_{in} is greater than V_{UT}, the output of IC1 is high, Q1 is on, and the LED is off.

(b) When V_{in} is less than V_{LT}, the output of IC2 is high, Q1 is on and the LED is off.

(c) When V_{in} is greater than V_{LT} and less than V_{UT}, the output of both IC1 and IC2 will be low, causing Q1 to turn off, allowing the LED to operate.

Thus, the LED is on only when V_{in} is 'in the window' as shown by the transfer curve.

APPENDIX (*Not for examination purposes*)

A.1. INTERFACING TO DIGITAL LOGIC

The LM311 comparator can be operated from a single rail power supply and, because it has an *open collector* output stage, it can be interfaced to TTL logic as shown in Fig.8. The pull up resistor R3 provides a logic 1, and the output stage of the LM311 provides the logic 0.

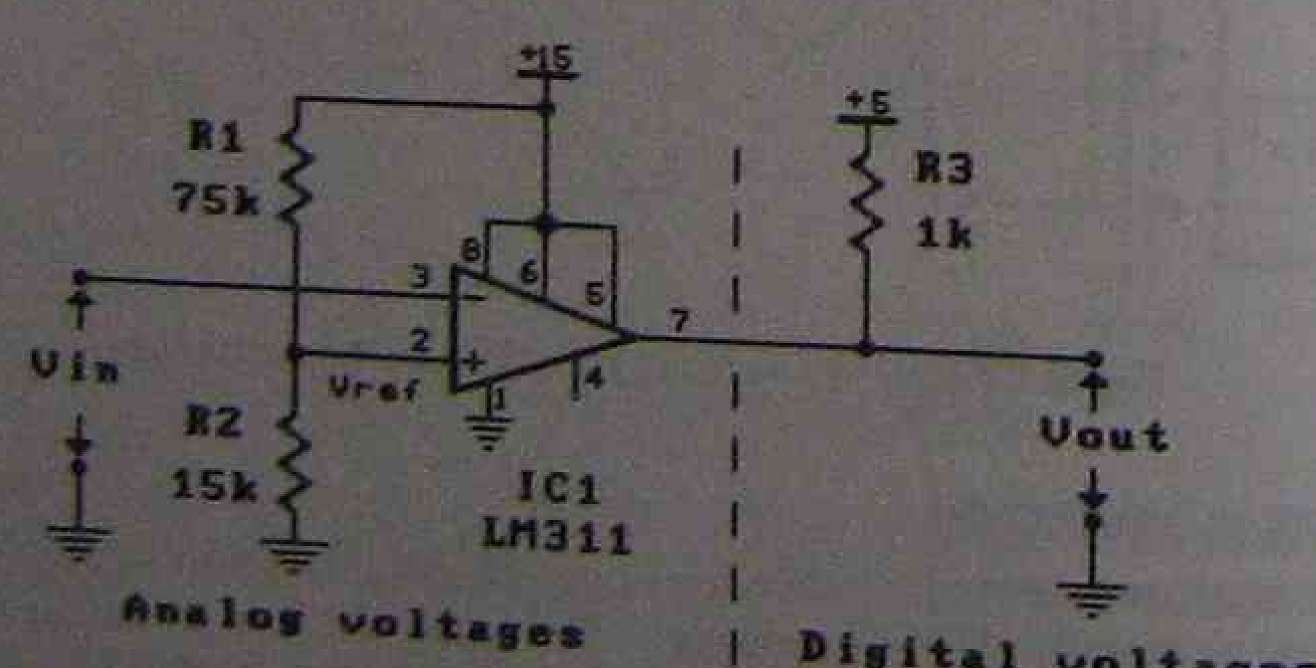


Fig.8: *The LM311 interfaced to TTL*

A.2. A LED BAR GRAPH DRIVER

Fig.9(a) shows the internal circuit of the National LM3914 LED bar graph driver IC. This IC uses ten comparators in a *staircase* arrangement, creating a set of ten threshold voltages.

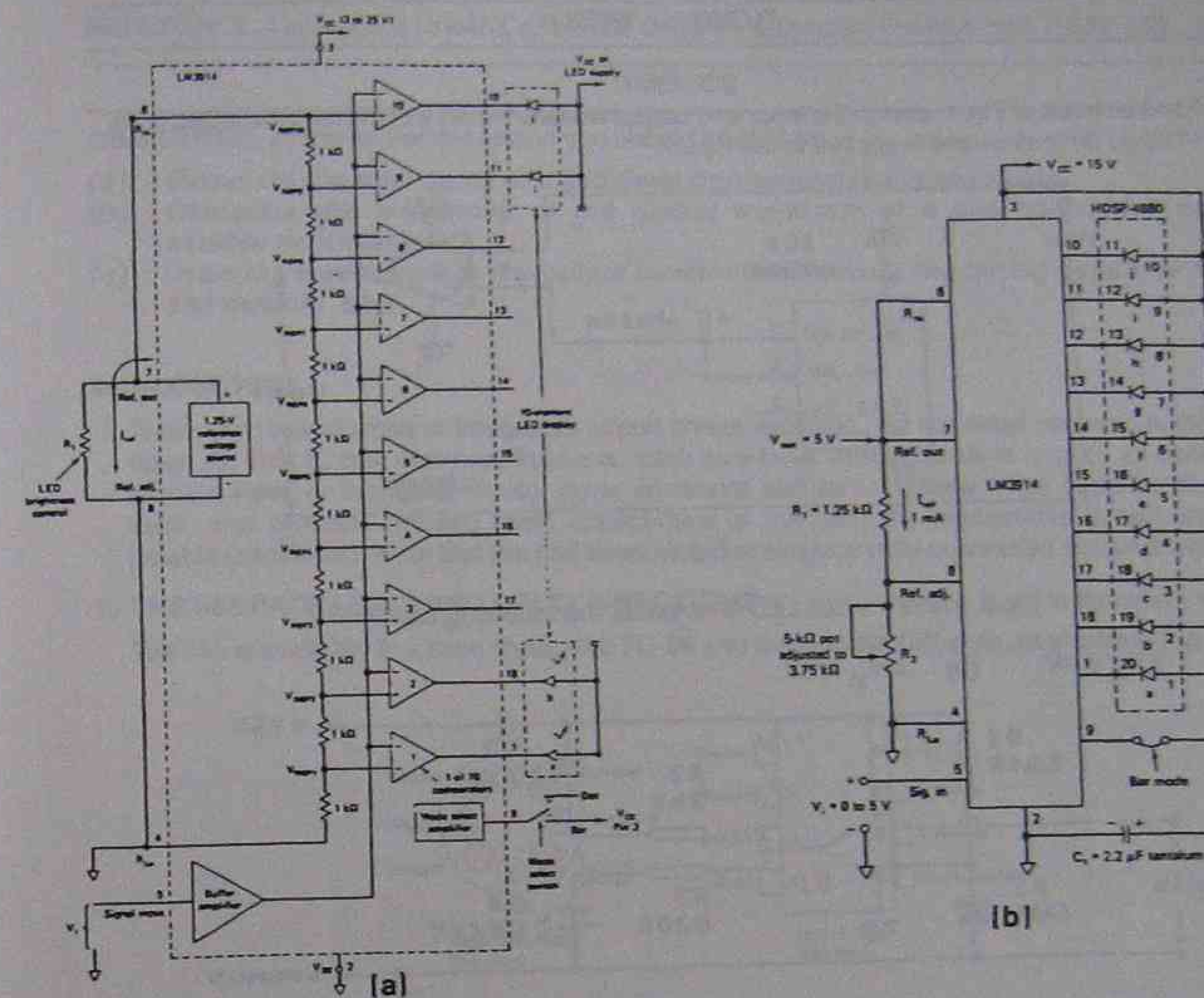


Fig.9: *A 0V to 5V light-column voltmeter using the LM3914.*

Fig.9(b) shows the LM3914 connected as a light-column voltmeter. The IC's internal (1.25V) voltage reference creates a constant current through R1. R2 is adjusted to set pin 6 to 5 volts. As the input voltage rises above each internal threshold level the related LED lights.

EXERCISE: Calculate the 10 threshold voltage levels for the circuit of Fig.9(b).

****

TUTORIAL WEEK 3

1. For the circuit of Fig.1, sketch the input and output waveforms on a common time scale, if an 11Vp-p, 1kHz sinewave is applied to the input.

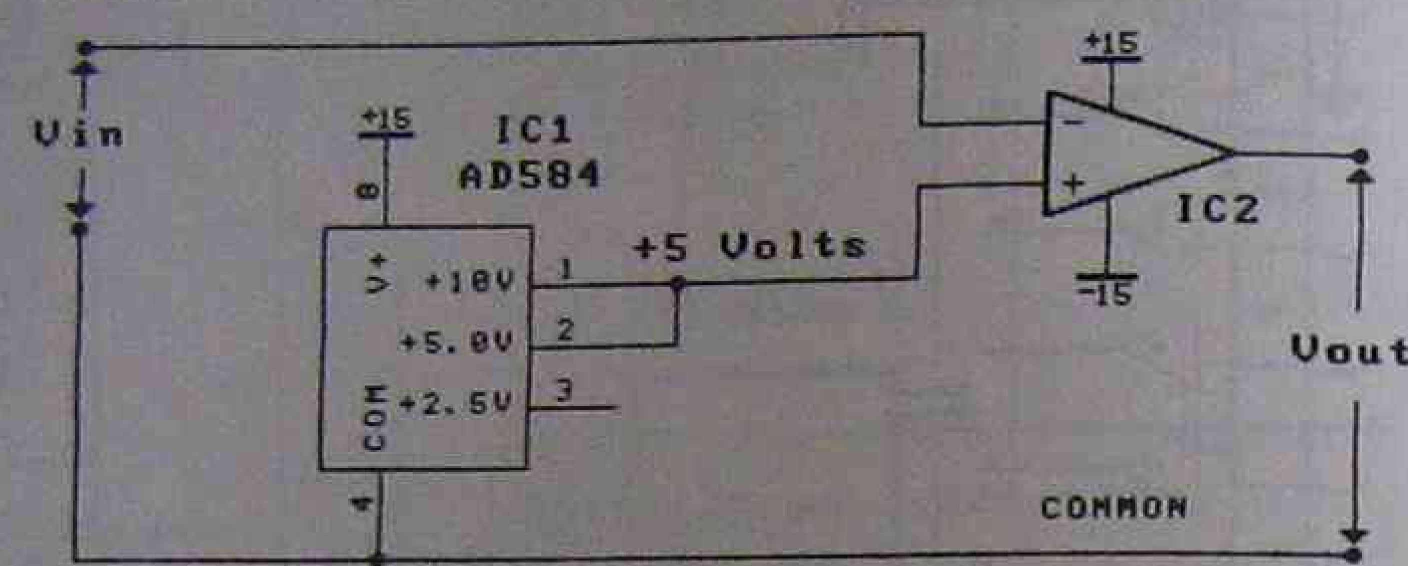


Fig.1.

2. For the circuit of Fig.2, identify which LED is on under the following conditions :

   (a) If V_{in} is 4V. (b) If V_{in} is 8V.

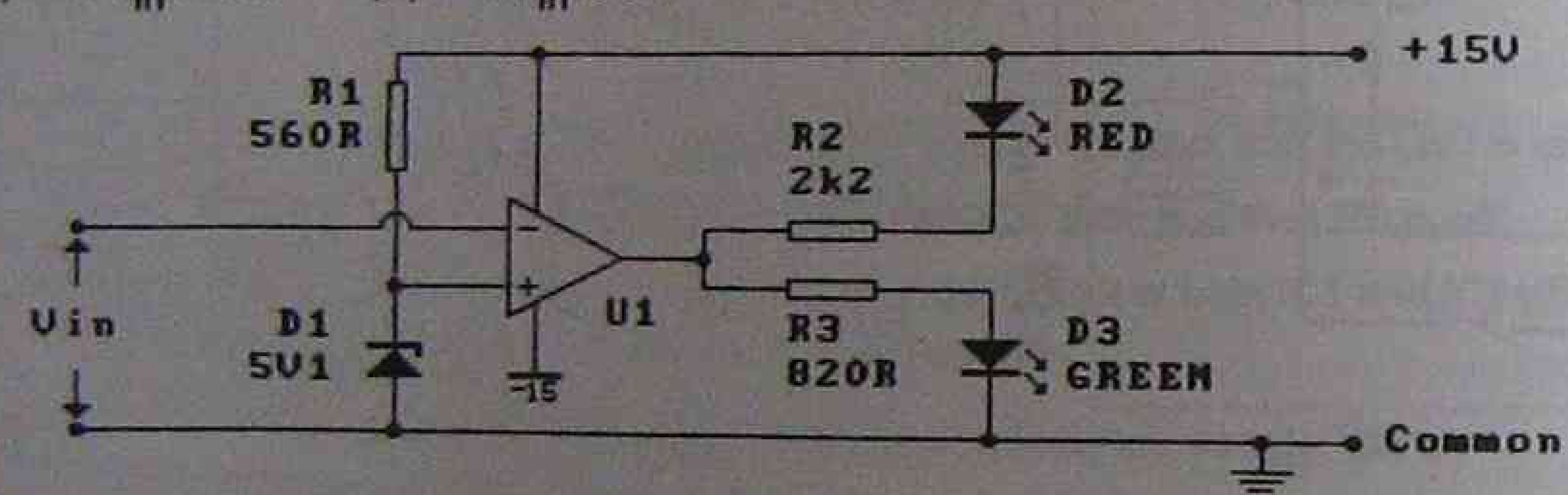


Fig.2.

3. Sketch the transfer curve for the circuit of Fig.2.

4. The circuit of Fig.3 is a Schmitt trigger, with Vref set to 0V.
   (a) Calculate the upper and lower threshold voltages at which switching will occur.
   (b) Sketch the input and output waveforms on a common time scale if V_{IN} is a 1Vp-p, 0.1Hz triangular wave.

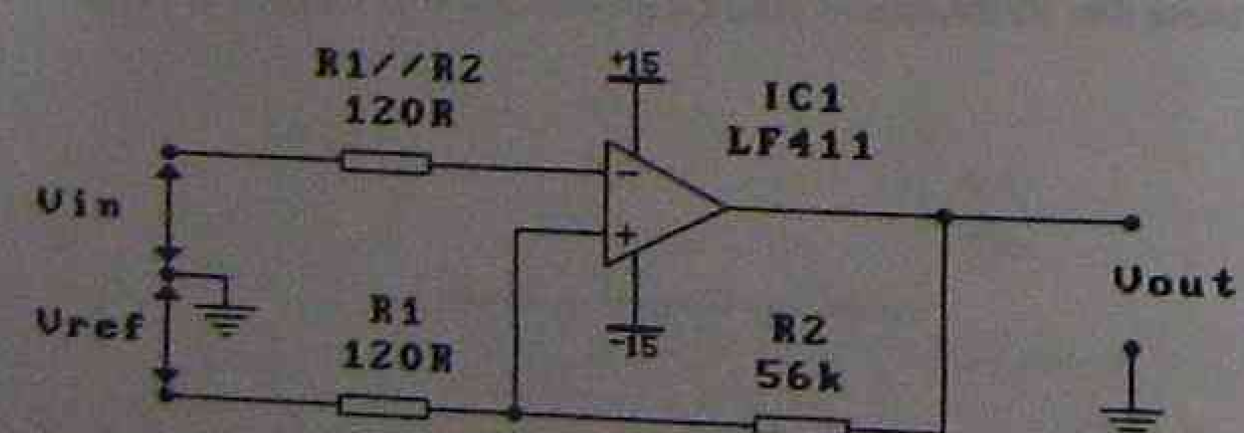


Fig.3.

****

THEORY LESSON 4

REFERENCE: Op Amps & Linear ICs, 3rd Ed, Coughlin & Driscoll, Prentice-Hall. P 330-363.

TIMER ICS

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

(a) *Calculate the time delay of a 555 timer connected as a monostable.*
(b) *Calculate the frequency of the output waveform of a 555 connected as an astable multivibrator.*
(c) *Draw the waveforms at the output terminal and across the timing capacitor for a 555 astable.*

INTRODUCTION

There are several types of integrated circuit timers available, but the most common is the 555 timer IC. This IC has many applications, such as a timer (milliseconds to hours), an oscillator (square wave or triangular wave), pulse generator and so on. These notes describe the 555 itself, and examine the two basic applications of the 555; the monostable (timer) and the astable (oscillator). Note that the 555 timer includes two internally connected comparators.

1. THE 555 PACKAGING AND PIN CONNECTIONS:

The 555 is available in 2 case styles, the TO-99 and the popular DIP style, as shown in Fig.1.

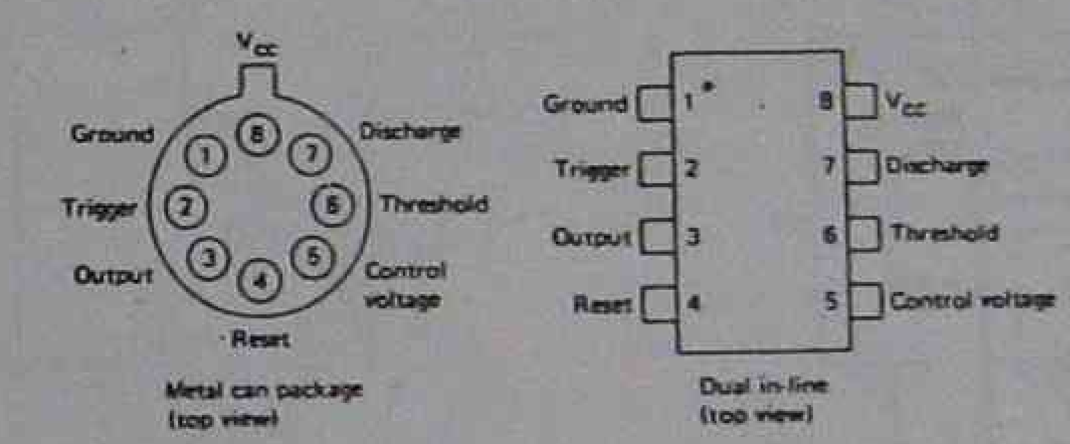


Fig.1: *555 pin connections*

The supply voltage to the 555 is within the range of +5V to +18V allowing it to be used in digital circuits, with linear 15V IC circuitry and 12V automobile circuits. Power is applied with the positive supply connected to pin 8 and the negative (ground) supply connected to pin 1.

1.1 INTERNAL CIRCUIT OF THE 555

The internal circuit of the 555 is shown in Fig.2. The internal circuit of the 555 consists of two comparators with the inputs connected to two preset voltages derived from an internal voltage divider, an SR flipflop triggered by the output of the comparators, an output stage and an internal transistor. The following describes the function of each pin for the IC which will show how the 555 works.

1.2 THE RESET TERMINAL (PIN 4)

The reset pin (pin 4) is usually connected to the +ve supply rail, but when bought to a logic 0 (ground) will reset the output of the 555 to a low. If the reset pin is held low, the output will also be held low, as it disables the internal flipflop.

1.3 CONTROL VOLTAGE TERMINAL (PIN 5)

This pin is used in special applications only and is usually connected to ground with a 10nF filter capacitor to remove any noise. This pin is used to modify the switching voltages of the comparators by allowing an external voltage to be applied.

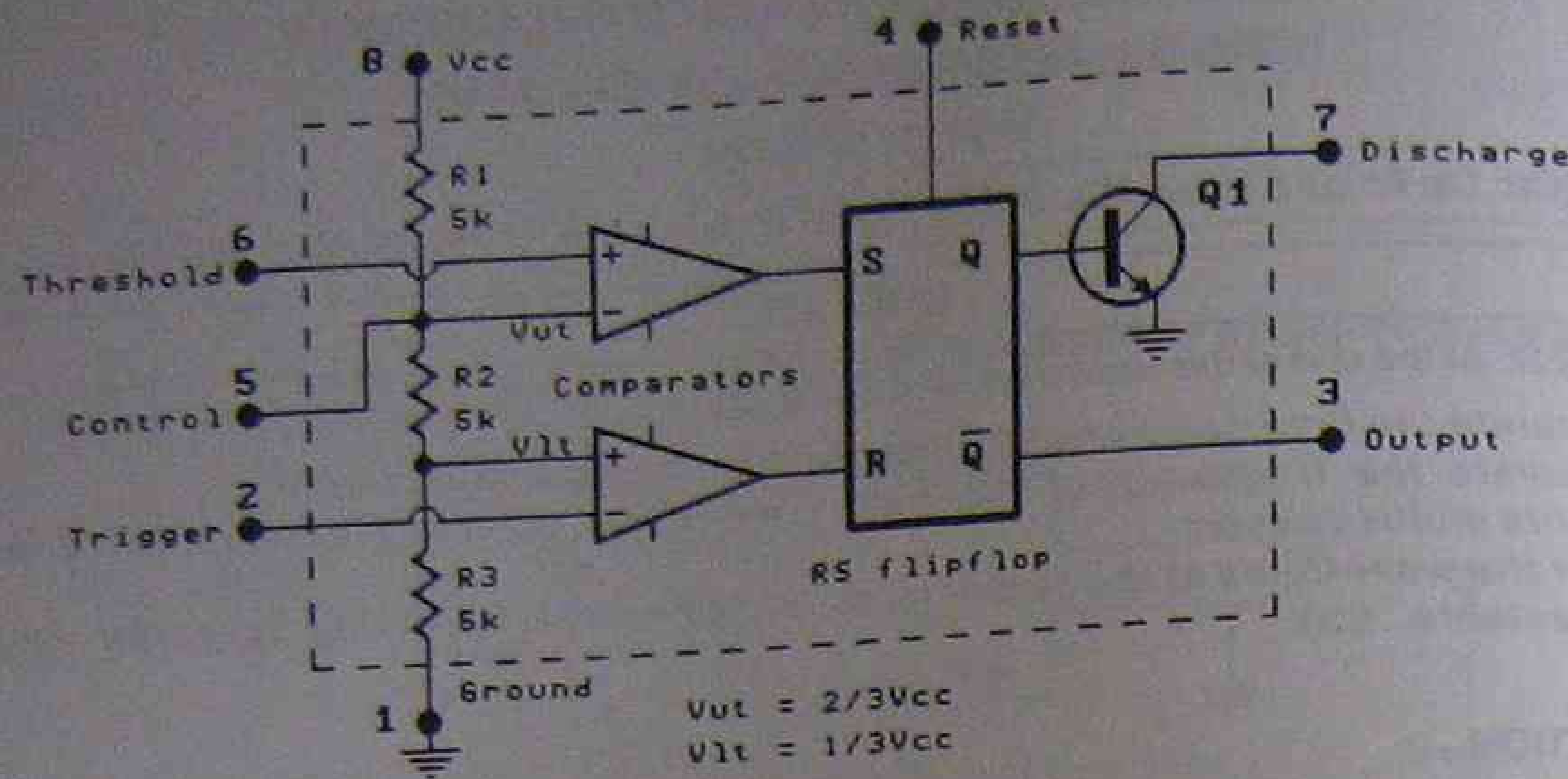
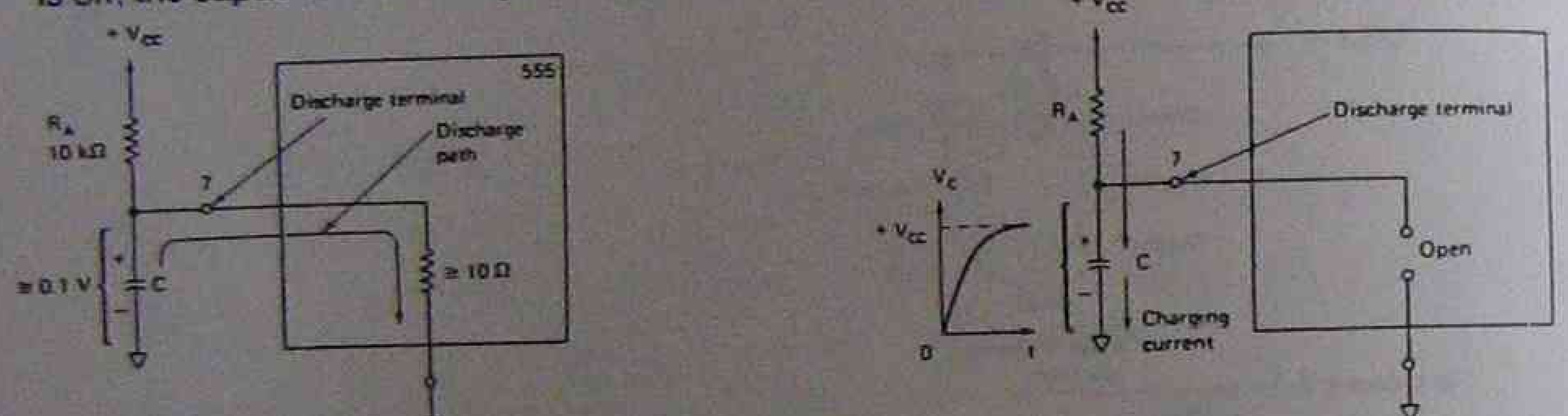


Fig.2: *Internal circuit of the 555*

1.4 THE DISCHARGE TERMINAL (PIN 7)

This terminal connects to the collector of an internal NPN transistor whose emitter is connected to ground. An external timing capacitor is usually connected between this pin and ground, and when the output of the 555 is high, the internal transistor is off. As shown in Fig.3, when the transistor is on, an externally connected capacitor will be discharged, and when the transistor is off, the capacitor can charge through a resistor connected to the +ve supply.



(a) Model of the discharge terminal when the output is low, and capacitor is discharging

(b) Model of the discharge terminal when the output is high, and capacitor is charging

Fig.3: *Operation of the discharge terminal, pin 7*

1.5 THE OUTPUT TERMINAL (PIN 3)

The output of the 555 has only two states: low or high, and can source or sink currents of 100mA or so. Depending on the load current, the output ~~will~~ voltage will be around 0.2V when low, and within 1V of the supply voltage when high. The load connections are shown in Fig.4.

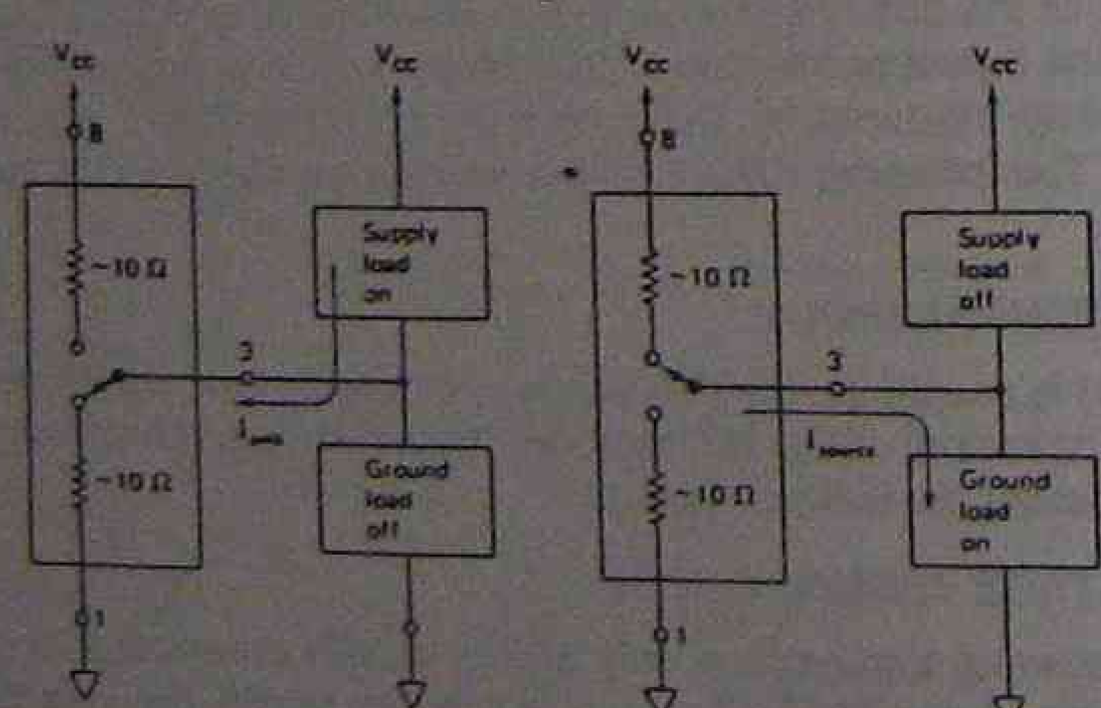


Fig.4: *Output load connections*

1.6 TRIGGER (PIN 2) AND THRESHOLD (PIN 6) TERMINALS

These pins connect directly to the internal comparators. As shown in Fig.2, the (-) terminal of the top comparator is held at 2/3Vcc, and the (+) terminal of the bottom comparator is held at 1/3Vcc by the internal voltage divider. The output of both comparators will therefore depend on the voltage applied to their other terminal. Thus, with two inputs (pins 2 and 6), the output states of the two comparators can produce four possible operating states as listed in the table of Fig.5. The operation of these terminals is examined in the next section.

Operating States of the 555 Timer; $V_{UT} = 2/3\ V_{CC}$, $V_{LT} = 1/3\ V_{CC}$, high = V_{CC}, low or ground = 0 V

Operating state	Trigger pin 2	Threshold pin 6	Output Terminal's State: Output 3	Discharge 7
H	Below V_{LT}	Below V_{UT}	High	Open
H	Below V_{LT}	Above V_{UT}	High	Open
M	Above V_{LT}	Below V_{UT}	Remembers last state	
L	Above V_{LT}	Above V_{UT}	Low	Ground

Fig.5: *The four input conditions of the 555*

2. THE ONE-SHOT OR MONOSTABLE

The basic function of the 555 is that of a timer, in which a negative going trigger pulse causes the output to go high for a period determined by the external timing components. After this time period, the output will return low ready for another trigger pulse. The timer application is also referred to as a *one shot* or a *monostable*. The 555 monostable circuit is shown in Fig.6.

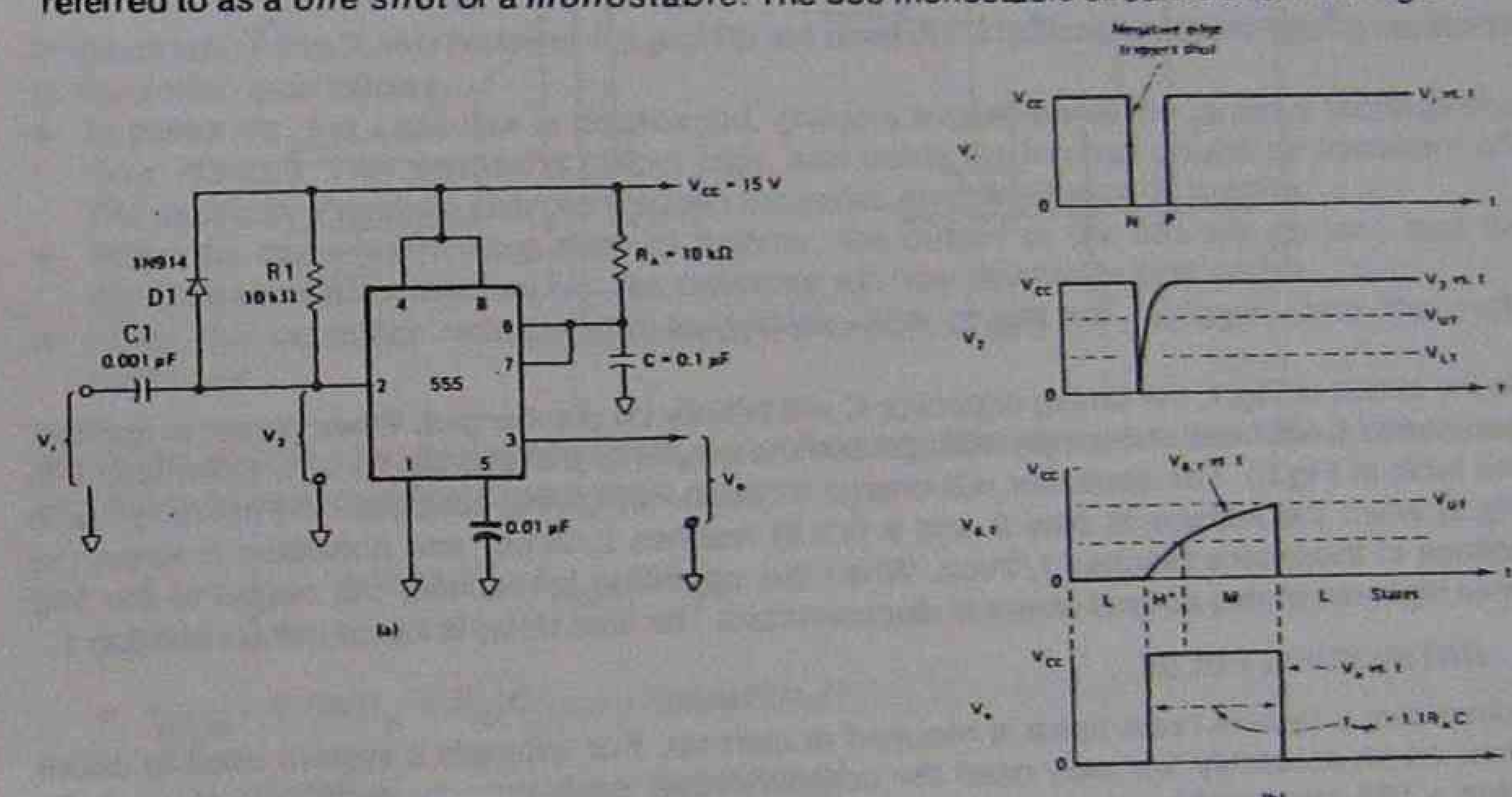


Fig.6: *555 monostable connection*

Operation of the circuit of Fig.6 is as follows:

- In the normal or *stable* state the output is low. Thus, the discharge terminal (pin 7) is also low, creating a short circuit across the timing capacitor C.
- If the voltage at the trigger input (pin 2) falls below 1/3Vcc, the internal comparator connected to pin 2 will reset the flipflop so that the output (pin 3) goes high. This will cause the discharge transistor to be turned off and the timing capacitor will charge towards Vcc through the timing resistor Ra.
- When the capacitor voltage reaches 2/3Vcc, the internal comparator connected to pin 6 (threshold) will set the internal flipflop causing the output to go low, (stable state). This will turn on the internal transistor, shorting the timing capacitor. The cycle will repeat if another negative going trigger pulse is applied.
- The time the output is high is found using the equation:

$$T = 1.1\, R_A C \quad \text{..... (equation 1)}$$

To ensure the trigger pulse is low for a period less than the time delay of the monostable, the differentiating circuit of R1 and C1 is used. The time constant for R1 and C1 should be about one-tenth that of C and Ra. Diode D1 prevents the input trigger pulse from raising the potential of pin 2 above that of Vcc.

3. APPLICATIONS OF THE MONOSTABLE

The monostable is used to produce pulses of a predetermined length. In some applications, a monostable is used as a *pulse stretcher* in which the output pulse length exceeds that of the input trigger pulse. If the output pulse is shorter in duration than the input trigger pulse, the circuit is a *pulse shrinker*). The following show applications of the 555 monostable.

3.1 POWER-ON TIME DELAY

In some electronic systems it is necessary to apply power immediately to one section, and delay power to another section. For example, in a motor control circuit, an alarm might be connected to sound if the motor was below a preset speed. At power-on, it will take some time before the motor reaches full speed, and a timer can be used to delay the application of power to the alarm section to allow the motor to run up to speed.

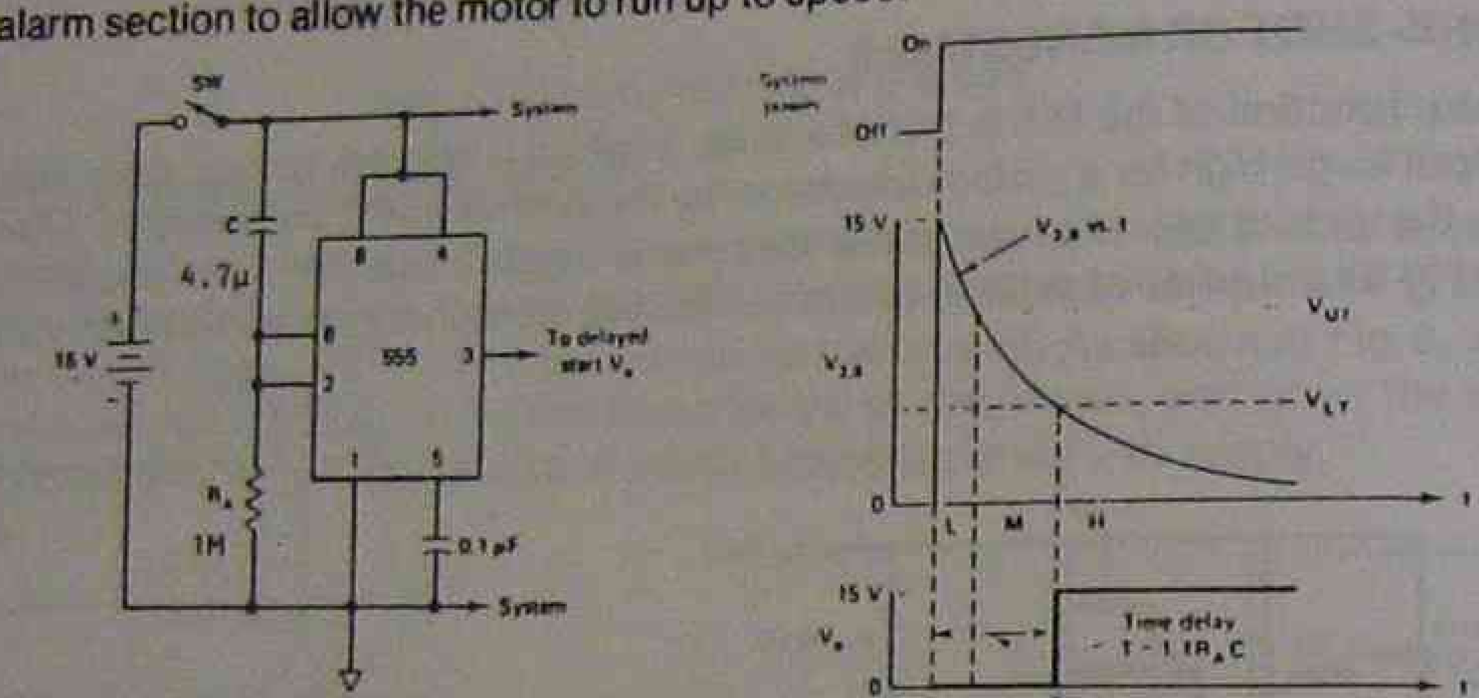


Fig.7: *Power-on time delay*

In the circuit of Fig.7, the timing capacitor C will initially be discharged. When power is applied, pins 2 and 6 will be at the supply voltage and the output of the 555 will be low, (condition L in the table of Fig.5). The capacitor will charge through Ra, giving condition M ('memory' as in Fig.5) when the voltage at pins 2 and 6 (V2.6) reaches 2/3Vcc, and condition H when the voltage at these pins reaches 1/3Vcc. When this condition is reached, the output of the 555 goes high, remaining so until power is disconnected. The time delay is found using equation 1.

3.2 INITIALISING PULSE

Sometimes a system reset pulse is required at start-up. For example a system used to count items on an assembly line may need the counters reset each time the system is started. By using a 555 monostable connected as shown in Fig.8, a reset pulse will be generated every time power is applied.

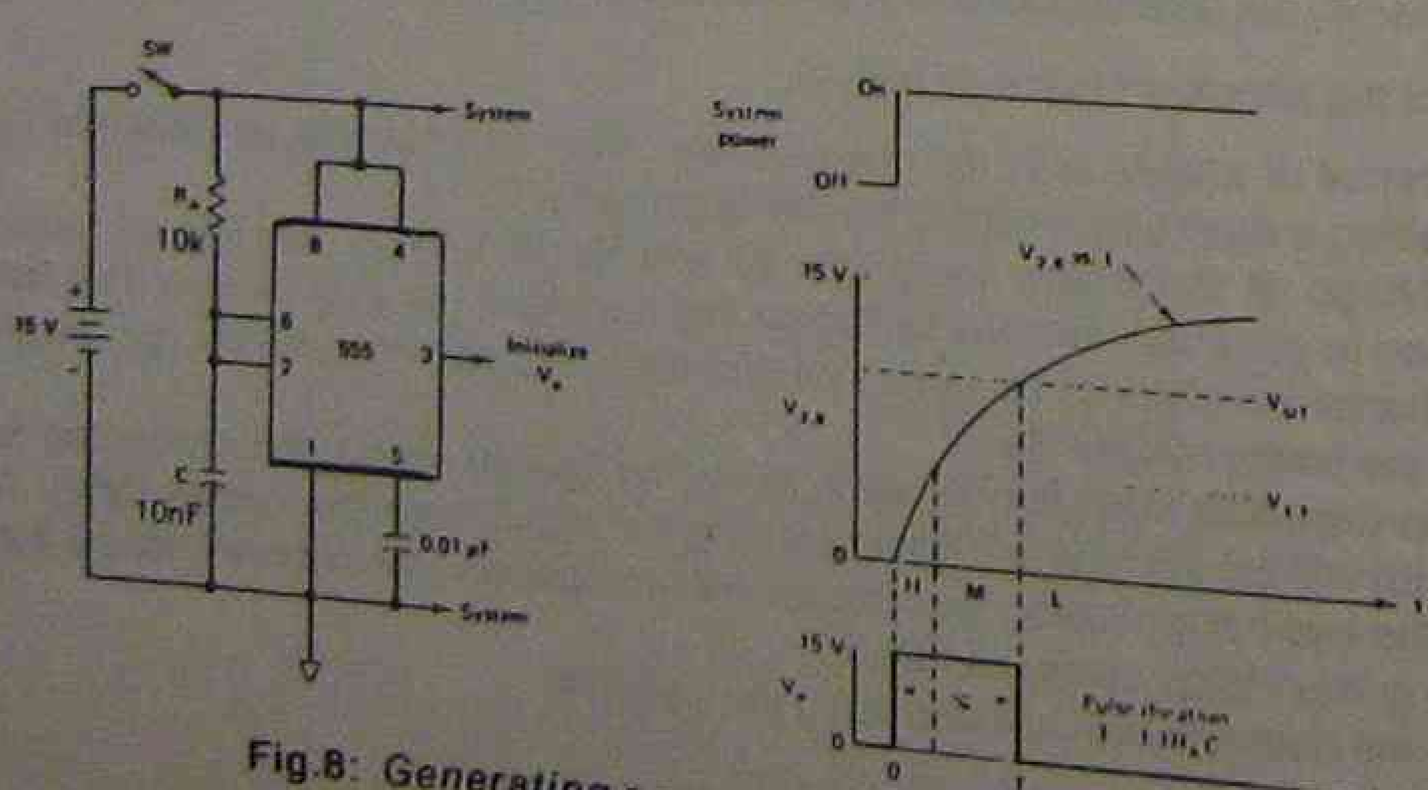


Fig.8: *Generating a reset pulse at power-on*

In Fig.8, the capacitor will initially be discharged when power is first applied, holding pins 2 and 6 at zero volts. The output will therefore be high, as shown in the table of Fig.5., (condition H). As the capacitor charges, condition M will occur (Vc = 1/3Vcc, no change in output) until eventually condition L is reached when Vc = 2/3Vcc. This will send the output low, where it remains until a power-off, power-on condition occurs. The width of the reset pulse can be calculated using equation 1.

4. THE 555 ASTABLE

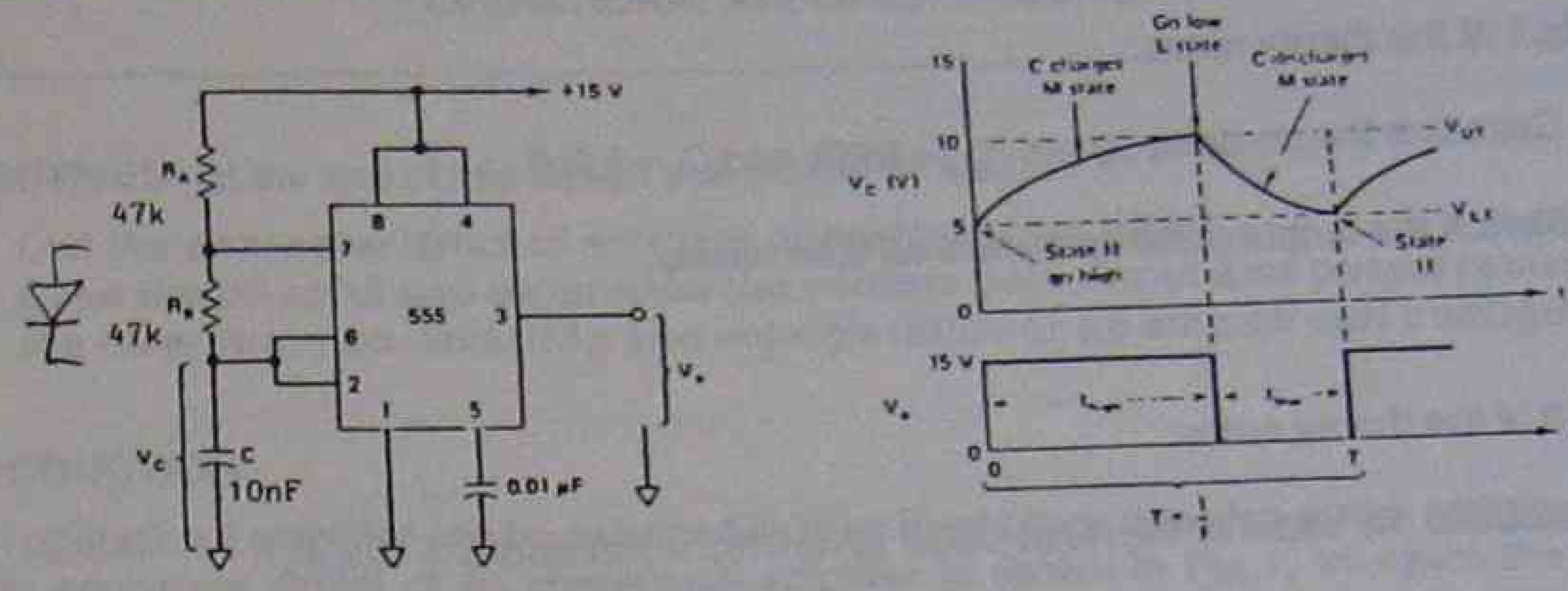


Fig.9: *The 555 Astable*

In the circuit of Fig.9, two resistors Ra and Rb are used in conjunction with the timing capacitor C. Operation is as follows:

- At power-on, the capacitor is discharged, giving a trigger condition to pin 2 (voltage less than 1/3Vcc). This sends the output high, and holds the internal discharge transistor off. The capacitor therefore charges through the series combination of Ra and Rb.
- When the capacitor voltage rises to 2/3Vcc, the output of the 555 will go low, and the discharge transistor will turn on. The capacitor will now discharge through Rb.
- When the capacitor voltage falls to 1/3Vcc, the circuit will retrigger, and the cycle continues.

Since the charge and discharge cycles occur through different resistance values, this circuit produces an asymmetrical square wave, where:

$$f = \frac{1}{T} \quad \text{where } T = t_{high} + t_{low}$$

- $t_{high} = 0.69(R_a + R_b)C$ .... (equation 2)
- $t_{low} = 0.69R_bC$ .... (equation 3)

Combining equations 2 and 3 gives:

- $T = 0.69(R_a + 2R_b)C$ .... (equation 4)

By using a value of resistance for Rb that is much higher than Ra, the output waveform can be made more symmetrical.

5. PRACTICAL ASPECTS OF THE 555

Manufacturers generally quote a minimum value of 1k for timing resistors used with the 555, and a maximum value of around 10M. The minimum value for the timing capacitor is usually specified as 500pF, although there is no limit to the maximum value. However, because the leakage current of electrolytic capacitors increases with capacitance value, practical values are generally limited to 1000uF. For reliable operation, a tantalum capacitor is generally recommended.

****

To ensure the trigger pulse is low for a period less than the time delay of the monostable, the differentiating circuit of R1 and C1 is used. The time constant for R1 and C1 should be about one-tenth that of C and Ra. Diode D1 prevents the input trigger pulse from raising the potential of pin 2 above that of Vcc.

3. APPLICATIONS OF THE MONOSTABLE

The monostable is used to produce pulses of a predetermined length. In some applications, a monostable is used as a *pulse stretcher* in which the output pulse length exceeds that of the input trigger pulse. If the output pulse is shorter in duration than the input trigger pulse, the circuit is a *pulse shrinker*). The following show applications of the 555 monostable.

3.1 POWER-ON TIME DELAY

In some electronic systems it is necessary to apply power immediately to one section, and delay power to another section. For example, in a motor control circuit, an alarm might be connected to sound if the motor was below a preset speed. At power-on, it will take some time before the motor reaches full speed, and a timer can be used to delay the application of power to the alarm section to allow the motor to run up to speed.

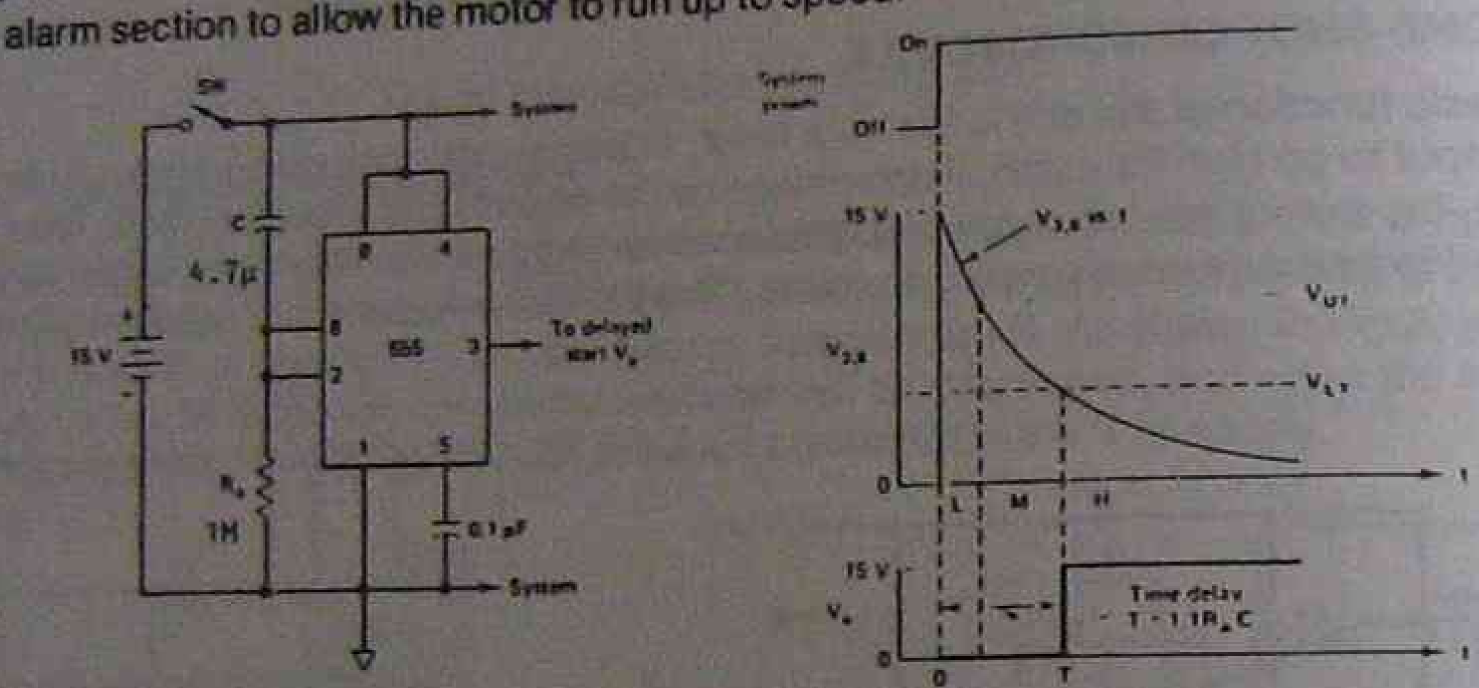


Fig.7: *Power-on time delay*

In the circuit of Fig.7, the timing capacitor C will initially be discharged. When power is applied, pins 2 and 6 will be at the supply voltage and the output of the 555 will be low, (condition L in the table of Fig.5). The capacitor will charge through Ra, giving condition M ('memory' as in Fig.5) when the voltage at pins 2 and 6 (V2.6) reaches 2/3Vcc, and condition H when the voltage at these pins reaches 1/3Vcc. When this condition is reached, the output of the 555 goes high, remaining so until power is disconnected. The time delay is found using equation 1.

3.2 INITIALISING PULSE

Sometimes a system reset pulse is required at start-up. For example a system used to count items on an assembly line may need the counters reset each time the system is started. By using a 555 monostable connected as shown in Fig.8, a reset pulse will be generated every time power is applied.

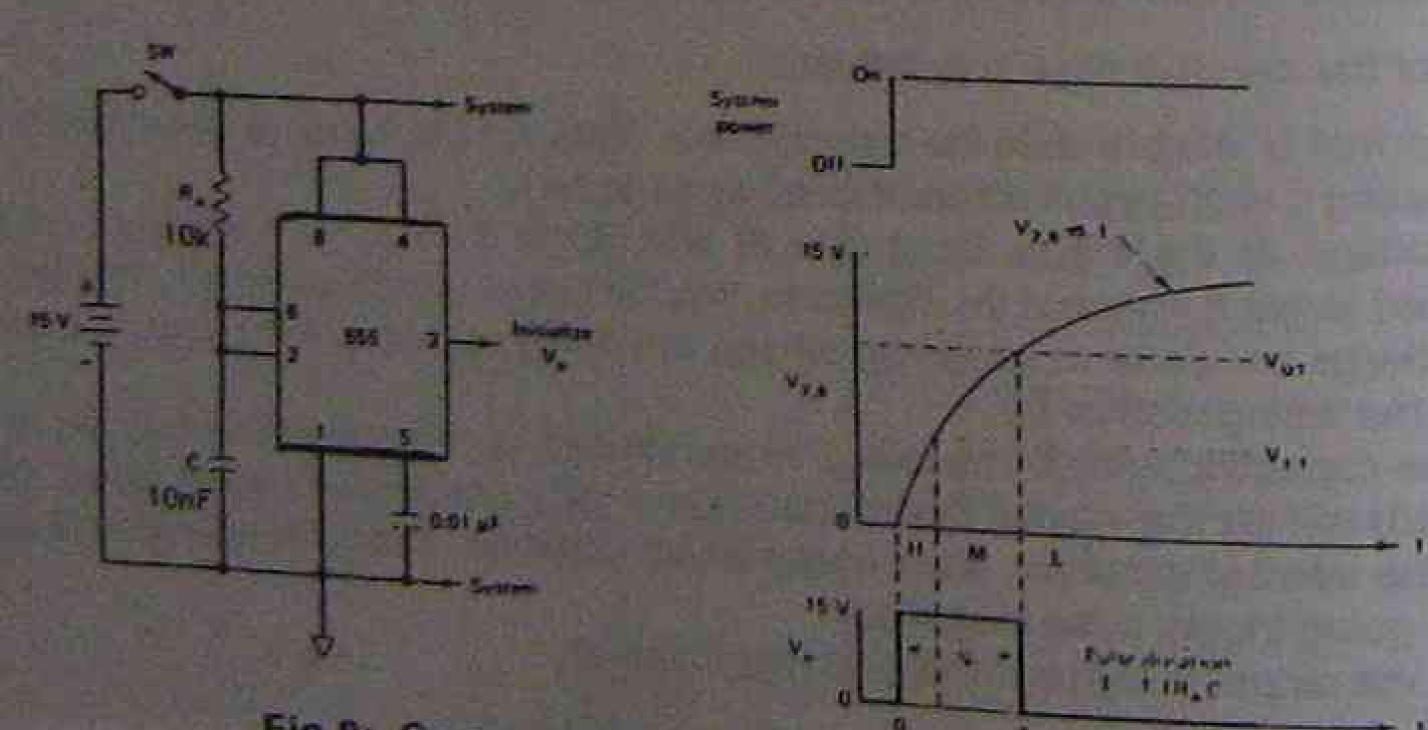


Fig.8: *Generating a reset pulse at power-on*

In Fig.8, the capacitor will initially be discharged when power is first applied, holding pins 2 and 6 at zero volts. The output will therefore be high, as shown in the table of Fig.5., (condition H). As the capacitor charges, condition M will occur (Vc = 1/3Vcc, no change in output) until eventually condition L is reached when Vc = 2/3Vcc. This will send the output low, where it remains until a power-off, power-on condition occurs. The width of the reset pulse can be calculated using equation 1.

4. THE 555 ASTABLE

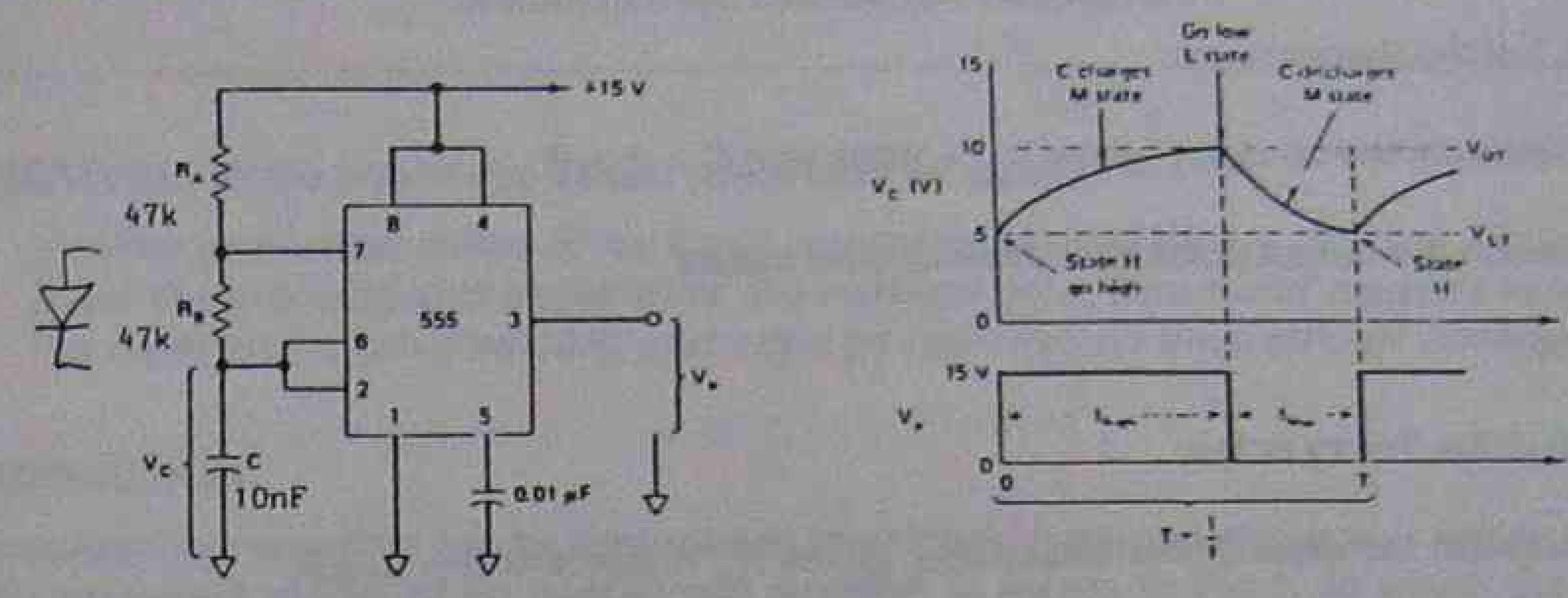


Fig.9: *The 555 Astable*

In the circuit of Fig.9, two resistors Ra and Rb are used in conjunction with the timing capacitor C. Operation is as follows:

- At power-on, the capacitor is discharged, giving a trigger condition to pin 2 (voltage less than 1/3Vcc). This sends the output high, and holds the internal discharge transistor off. The capacitor therefore charges through the series combination of Ra and Rb.
- When the capacitor voltage rises to 2/3Vcc, the output of the 555 will go low, and the discharge transistor will turn on. The capacitor will now discharge through Rb.
- When the capacitor voltage falls to 1/3Vcc, the circuit will retrigger, and the cycle continues.

Since the charge and discharge cycles occur through different resistance values, this circuit produces an asymmetrical square wave, where:

$$f = \frac{1}{T} \quad \text{where } T = t_{high} + t_{low}$$

- $t_{high} = 0.69(R_a + R_b)C$ .... (equation 2)
- $t_{low} = 0.69R_bC$ .... (equation 3)

Combining equations 2 and 3 gives:

- $T = 0.69(R_a + 2R_b)C$ .... (equation 4)

By using a value of resistance for Rb that is much higher than Ra, the output waveform can be made more symmetrical.

5. PRACTICAL ASPECTS OF THE 555

Manufacturers generally quote a minimum value of 1k for timing resistors used with the 555, and a maximum value of around 10M. The minimum value for the timing capacitor is usually specified as 500pF, although there is no limit to the maximum value. However, because the leakage current of electrolytic capacitors increases with capacitance value, practical values are generally limited to 1000uF. For reliable operation, a tantalum capacitor is generally recommended.

****

TUTORIAL WEEK 4

1. For Fig.7 of the theory notes:

   (a) Calculate the on-delay, given R_A = 820k and C = 2.2μF.

   (b) Sketch the output pulse and show all time values.

2. For Fig.8 of the theory notes:

   (a) Calculate the width of the reset pulse given R_A = 22k and C = 3.3μF.

   (b) Sketch the output pulse and show all time values.

3. For Fig.9 of the theory notes:

   (a) Calculate the period of oscillation, given R_A = 6.8k, R_B = 1.5M and C = 1nF.

   (b) Sketch the waveforms across the capacitor and the output. Include the time the output is high and time the output is low and the relative voltage values for the waveform across the capacitor.

4. For Fig.6 of the theory notes:

   (a) Calculate the width of the output pulse given R_A = 560k and C = 470μF.

   (b) Sketch the output waveform related to the input trigger pulse.

****

THEORY LESSON 5

OPERATIONAL AMPLIFIER CIRCUITS - 1

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

(a) List the characteristics of an ideal operational amplifier.
(b) Draw the circuits and determine the voltage gain, input and output resistance of the inverting, non-inverting and voltage follower op amp circuit configurations.

INTRODUCTION

An operational amplifier can be assumed to have ideal characteristics under certain conditions. The equivalent circuit of an operational amplifier is shown in Fig.1, in which the differential voltage present at the inputs (Vid) is across the internal input resistance (Ri) of the amplifier. The output voltage (Vo) equals Vid x Avol, where Avol is the open loop voltage gain (gain without feedback) of the amplifier. The output resistance of the circuit is represented by Ro.

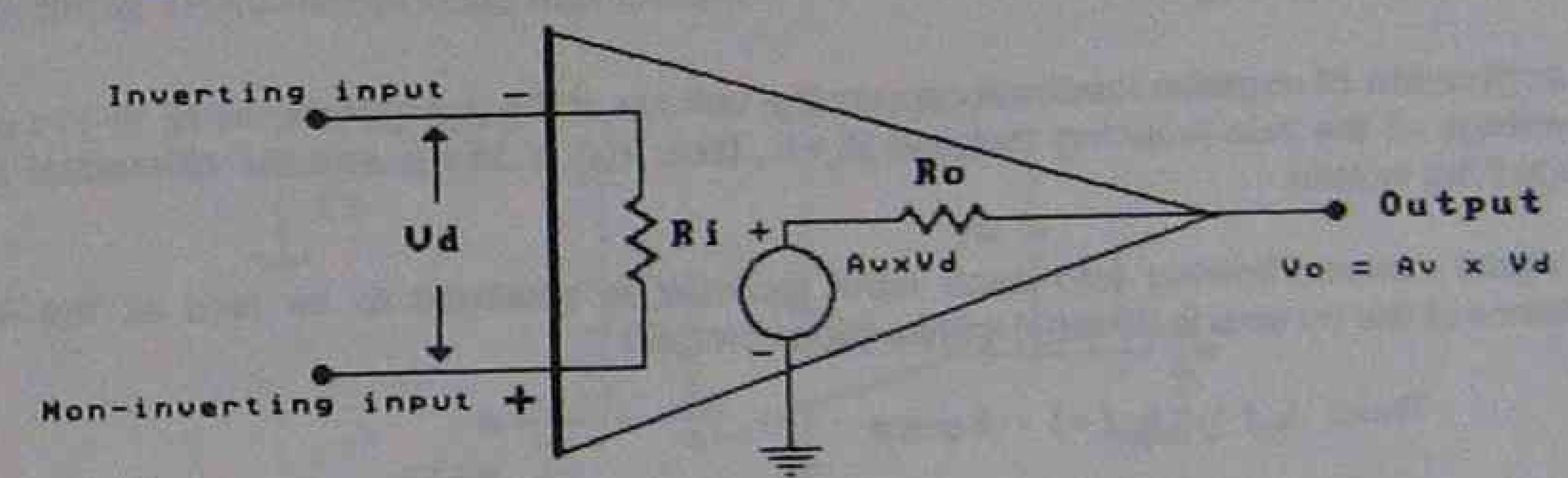


Fig.1: *The equivalent circuit of an op amp.*

The ideal characteristics of an operational amplifier are:

1. Infinite open-loop voltage gain, A_{vol}.
2. Infinite input resistance, Rin. (Equivalent to an open circuit).
3. Zero output resistance, Ro. The output can drive any load without a reduction in Vo.
4. Infinite bandwidth, (BW). The amplifier can amplify signals from DC to MHz.
5. A common mode rejection ratio of infinity, or zero gain to common mode noise signals.
6. An output voltage of 0V for an input of 0V. That is, no DC changes within the amplifier.
7. Infinite slew rate. That is, the output voltage exactly follows the input voltage.

These ideal characteristics don't occur in practical circuits, but with the application of external negative feedback these characteristics can be brought close to the ideal. In many circuits the practical limitations will be negligible and the op amp can be treated as an ideal component.

2. THE IDEAL INVERTING AMPLIFIER

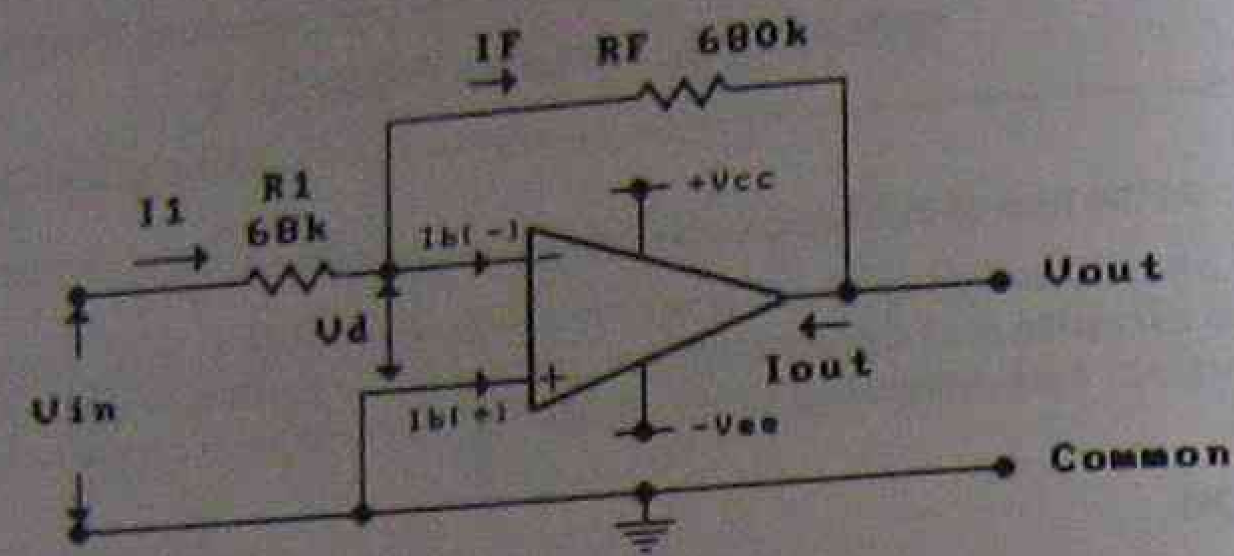


Fig.2: *The Inverting Amplifier.*

In this circuit a resistor is placed between Vout and the inverting input terminal, V(-). This results in a fraction of Vout being fed back to the input, and means the amplifier is operating in *CLOSED LOOP*, with a subsequent reduction in gain. The amount of feedback is determined by the ratio of R1 and RF, and the *feedback factor* (B) is determined by:

$$B = \frac{R1}{R1 + RF}$$

The application of negative feedback causes the voltage at the inverting terminal, V(-) to equal the voltage at the non-inverting terminal V(+). Thus V(-) = V(+), and the differential input voltage (Vid) is zero.

The signal current flowing into each input terminal is assumed to be zero as the input resistance of the op amp is (ideally) infinite (open circuit).

Thus: $I_{in}(-) = I_{in}(+) = 0$ amps

2.1 GAIN OF THE INVERTING AMPLIFIER

The non-inverting terminal is connected to ground, and as the differential voltage is zero, the inverting terminal is also at zero volts, (called a *virtual ground*). That is:

1. $V(+) = 0$ volts $= V(-)$, as $V_{id} = 0$ volts.

When Vin is applied, an input current will flow through R1, as Vin appears across R1. (Vin one end of R1, 0V at the other). Thus:

2. $I_1 = \frac{Vin}{R_1}$

As no signal current flows into the op amp's input terminals I_1 must flow around the op amp through R_F.

3. $I_1 = I_F$

This produces a voltage drop across R_F equal to:

4. $V_{RF} = I_F R_F = I_1 R_F$

As R_F has one end (left side) at virtual ground, (0V) and the other at Vout, Vout equals:

5. $Vout = (-)V_{RF} = (-)I_1R_F$ **Note:** (-) means 180 degree phase inversion.

The voltage gain with negative feedback applied is called the *CLOSED LOOP VOLTAGE GAIN*, usually referred to as A_v. Note that this is the gain of the *circuit*, not that of the op amp.

6. $A_v = \frac{Vout}{Vin}$

7. $= (-)\frac{I_1 R_F}{I_1 R_1}$ **Note:** The (-) sign indicates 180 degree phase inversion.

8. $A_v = (-)\frac{R_F}{R_1}$ **(Gain equation for the inverting amplifier)**

2.2 INPUT AND OUTPUT RESISTANCE

The input resistance (between input terminals) of the op amp is assumed to be an open circuit. However, the input voltage appears across R1, which has one end connected to a virtual earth (0V). Therefore, for the inverting amplifier:

9. $Rin = R_1$ **Note:** this is the input resistance of the *circuit*, not that of the op amp.

The output resistance of the circuit is Ro of the op amp, which is (ideally) zero.

3. THE IDEAL NON-INVERTING AMPLIFIER

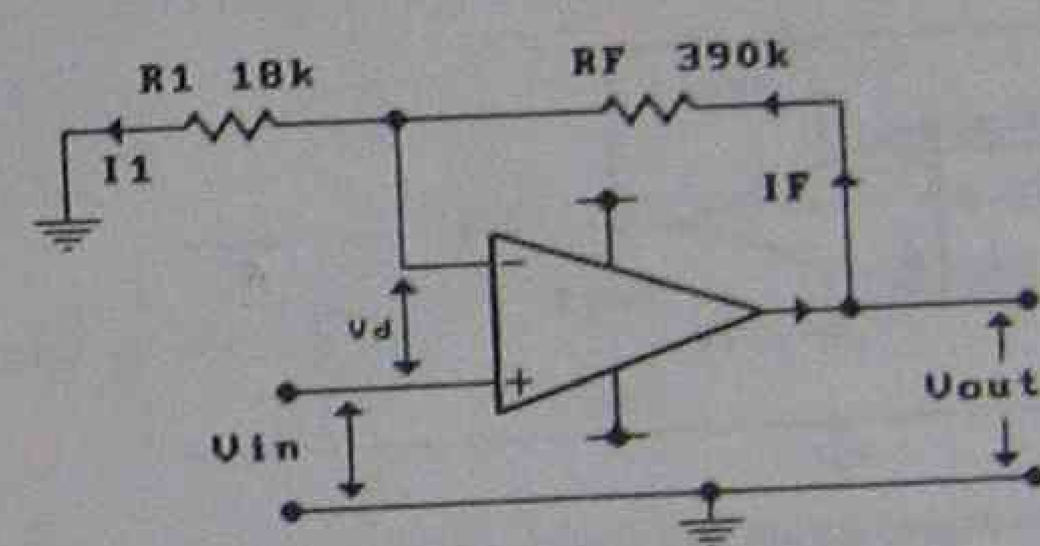


Fig.3: *The Non-inverting Amplifier.*

The circuit of Fig.3 shows a non-inverting amplifier, in which the input voltage (Vin) is applied directly to the non-inverting input. Because negative feedback is applied with R_F, the differential input voltage is zero. Thus:

1. $Vin = V(+) = V(-)$

Therefore, Vin occurs across R1 as in the inverting amplifier, causing a current (I_1) to flow. As the input resistance of the op amp is an open circuit, I_1 flows through R_F, giving:

2. $I_1 = I_F$

Vout appears between the output terminal and ground, and also across both R_1 and R_F. Thus:

3. $Vout = V_{R1} + V_{RF}$

Since these voltages are in series, the output voltage is non-inverted and larger than Vin.

3.1 VOLTAGE GAIN OF THE NON-INVERTING AMPLIFIER

The universal equation for voltage gain is $A_V = \frac{Vout}{Vin}$

Therefore, from 3. (previous page):

$$4.\quad A_V = \frac{V_{R1} + V_{RF}}{Vin}$$

$$= \frac{I_1R_1 + I_FR_F}{I_1R_1} \qquad = \frac{I1\,(R1 + RF)}{I_1R_1} \quad (\text{as } I_1 = I_F)$$

$$= \frac{R_F + R_1}{R_1}$$

$$A_V = \frac{R_F}{R_1} + 1$$ **(Gain equation for the non-inverting amplifier)**

3.2 INPUT AND OUTPUT RESISTANCE

The input resistance of this circuit is the input resistance of the op amp, which equals (ideally), infinity. This is a feature of the non-inverting amplifier and is often why this circuit is used.

The output resistance of the non-inverting amplifier is that of the op amp, and is (ideally) zero.

4. THE IDEAL VOLTAGE-FOLLOWER

The circuit of Fig.4 shows a voltage follower, also referred to as a **Buffer** or **Unity Gain** amplifier.

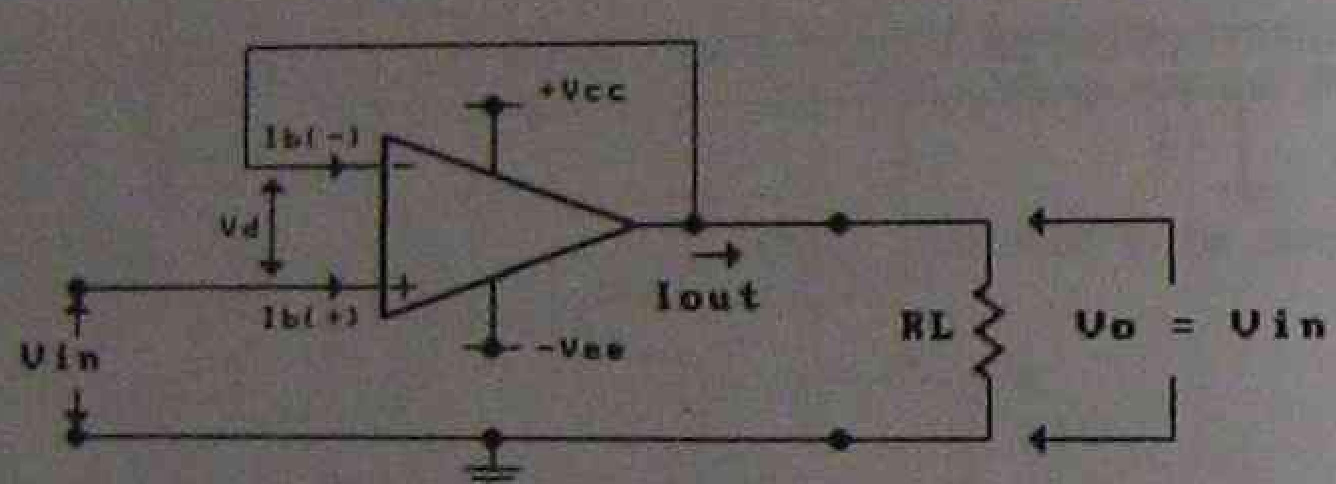


Fig.4: *The Ideal Voltage Follower*

In Fig.4, Vin is connected directly to the non-inverting input and the output is connected directly to the inverting input giving a feedback of 100%. Because the op amp has an open circuit between its input terminals, no current flows in the feedback connection, nor is there any current taken from the input signal. Because the differential input voltage is zero:

(1) Vin = V(+) = V(-) = Vout. Thus the voltage across the load (Vout) equals Vin.

4.1 AC CONDITIONS

Because the output voltage equals the input voltage, the gain (Av) of the voltage follower circuit is unity, and there is no phase difference between the input and the output.

Input and output resistance are virtually equal to the ideal values of:

(2) Rin = infinity. (3) Ro = zero. Negative feedback will keep these values close to the ideal.

****

TUTORIAL WEEK 5

1. List the ideal characteristics of an operational amplifier.
2. Assuming ideal op amps, determine the voltage gain and output voltage for each circuit of Fig.1.

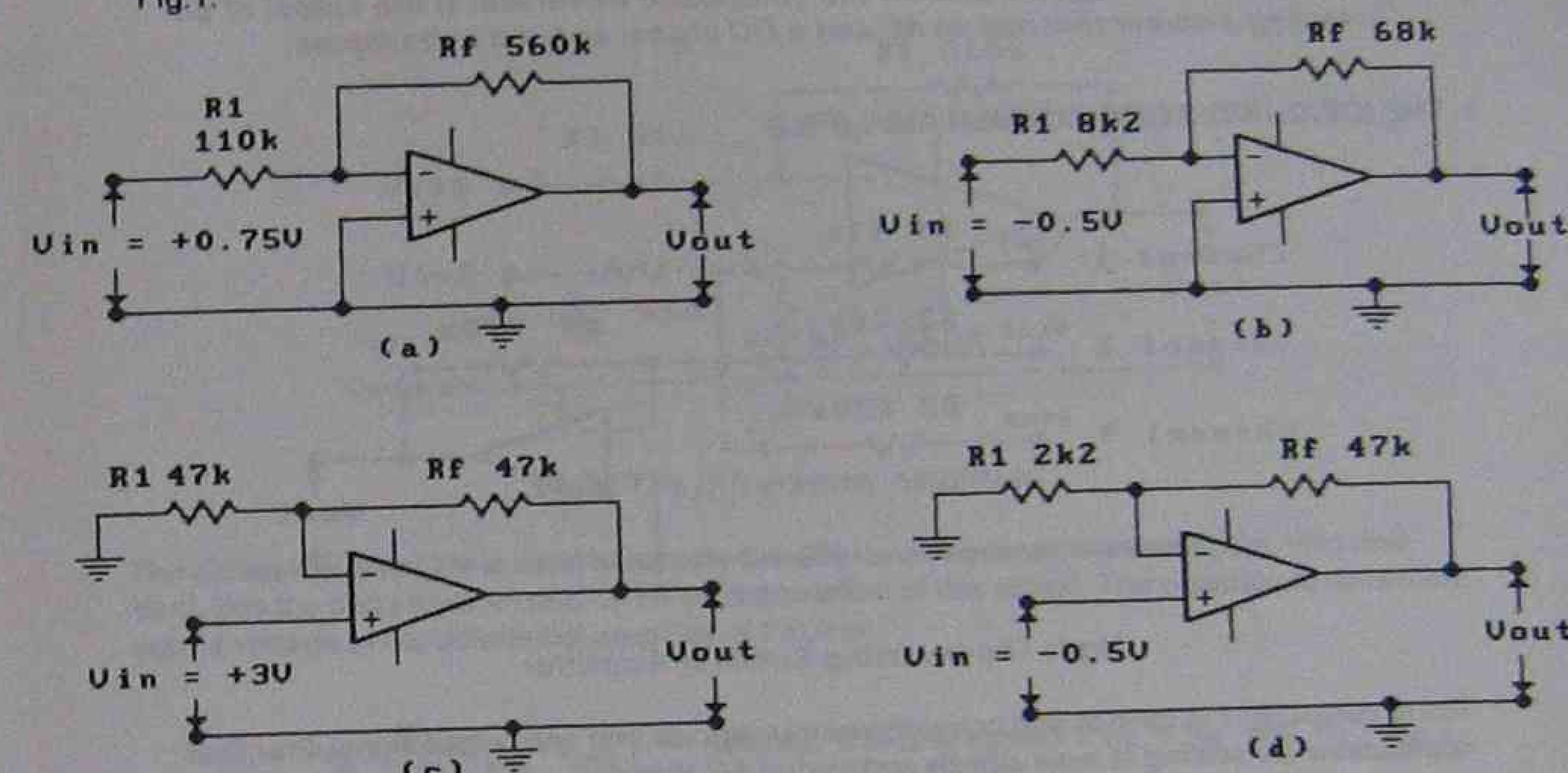


Fig.1.

3. Assuming ideal op amps, determine the value of the resistor marked (?) required to set the gain to that specified for each circuit of Fig.2.

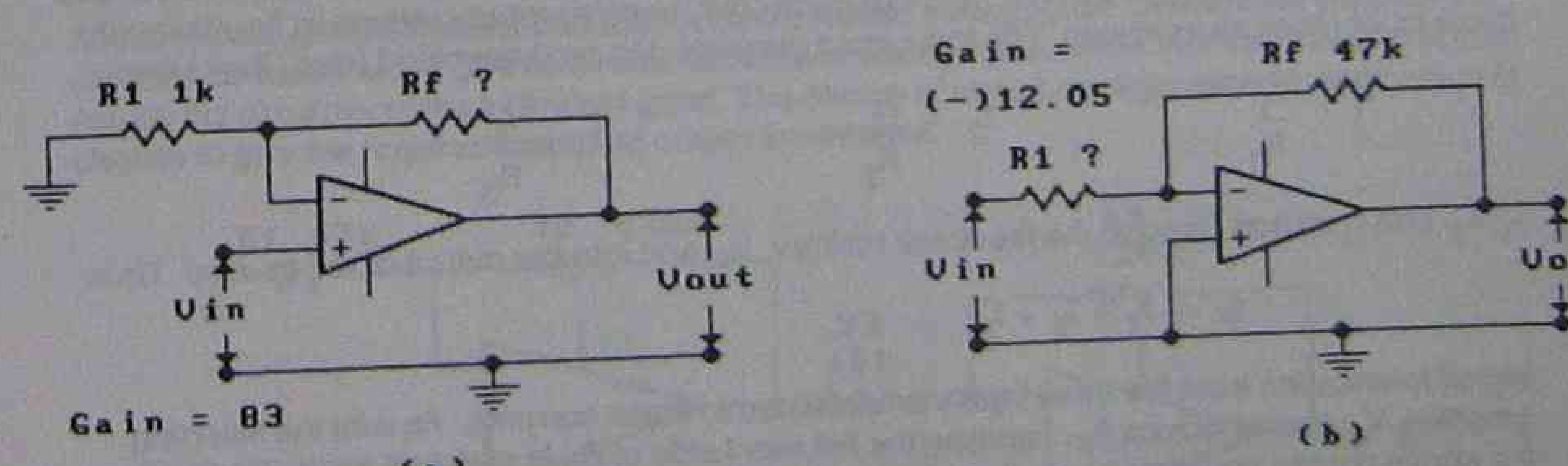


Fig.2.

****

OPERATIONAL AMPLIFIER CIRCUITS - 2

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

(a) *Draw the circuits of and calculate the output voltages for an inverting summer and a differential amplifier that use an op amp as the active device.*
(b) *Calculate the overall gain of a op amp circuit that contains cascaded stages.*
(c) *Calculate the voltages and draw the composite waveform at the output of an inverting summer that has an AC and a DC signal applied to its inputs.*

1. THE IDEAL INVERTING SUMMER AMPLIFIER

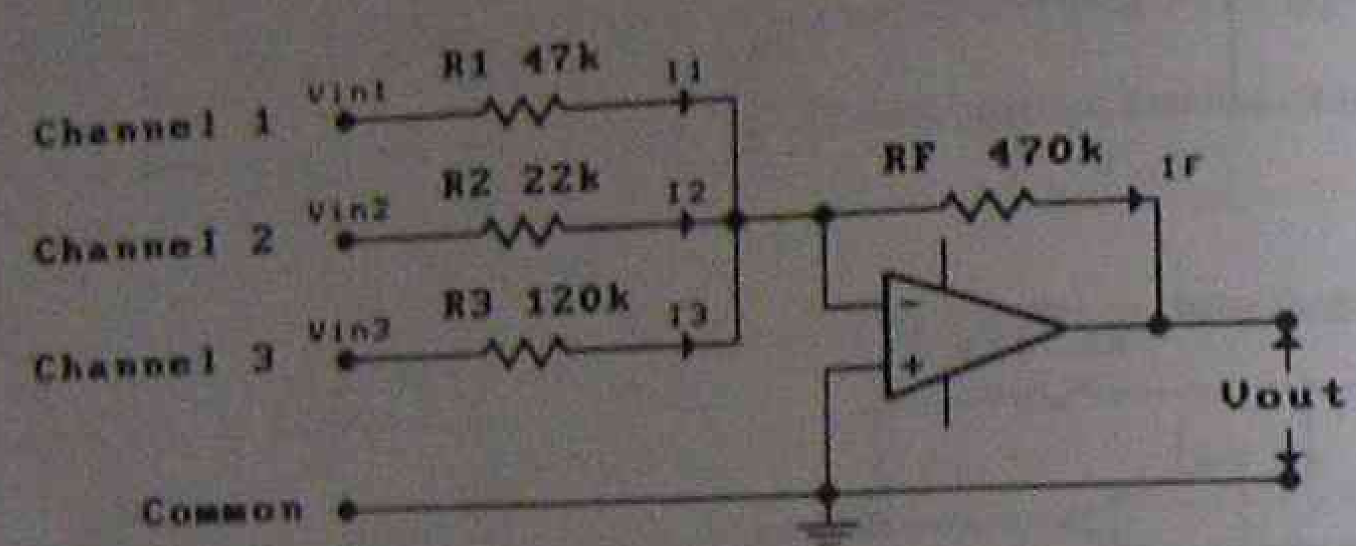


Fig.1: *The Inverting Summer Amplifier*

The circuit of Fig.1 is used to add or combine input signals into one output signal. Practical applications are scaling of input signals and mixing AC signals.

1.1 CIRCUIT ANALYSIS

In the circuit of Fig.1, three input voltages are applied, Vin1 to input one, Vin2 to input two and Vin3 to input three. The input voltages appear across the respective input resistors (R1, R2 and R3) as the inverting terminal V(-) is at virtual earth (0V), causing input currents to flow through these input resistors as shown. The three input currents can be determined from Ohm's law:

$$I_1 = \frac{V_1}{R_1} \qquad I_2 = \frac{V_2}{R_2} \qquad I_3 = \frac{V_3}{R_3}$$

I_1, I_2, and I_3 all flow through the feedback resistor, R_F and into the output of the op amp. Thus:

$$I_F = I_1 + I_2 + I_3$$

Signal information from the three inputs is combined in these currents. As with the inverting amplifier, V_o occurs across R_F, because the left hand side of R_F is at virtual earth. Converting the above current equation to a voltage based equation, gives:

$$\frac{V_o}{R_F} = -\left[\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3}\right]$$ (Note: the (-) sign indicates output is inverted)

Multiplying the equation through by R_F, results in:

$$V_o = -\left[\frac{R_F V_1}{R_1} + \frac{R_F V_2}{R_2} + \frac{R_F V_3}{R_3}\right]$$

This equation shows that each input voltage is multiplied by a gain, determined by the value of each input resistor and R_F. This applies to both DC and AC input signals. Don't forget to include the minus sign for any negative DC input voltages, and include the minus sign outside the brackets of the equation to obtain the correct polarity output for DC values.

The input resistance of each input channel equals the individual resistors; R1 for channel 1, R2 for channel 2 and R3 for input channel 3.

2. THE IDEAL DIFFERENTIAL AMPLIFIER

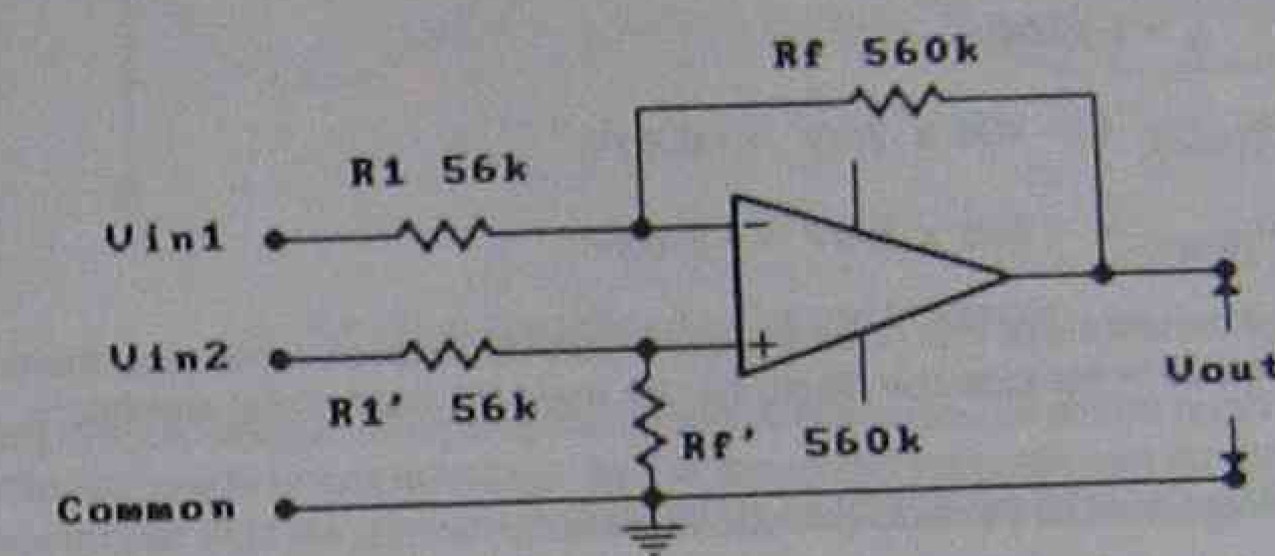


Fig.2: *The Differential Amplifier*

The differential amplifier is used to amplify the difference between the two inputs, Vin1 and Vin2. See the text pages 44 and 45 for an explanation of this circuit. The equation to determine output voltage of the differential amplifier of Fig.2 is:

$$V_o = \frac{R_F}{R_1}(V_{in2} - V_{in1})$$ (Note: This only applies if $R_F = R_F'$ and $R_1 = R_1'$)

If two in-phase (common mode) voltages of the same amplitude are applied to the circuit, the output voltage will be zero. That is, common mode signals are rejected by this circuit.

3. CASCADING OPERATIONAL AMPLIFIER CIRCUITS

Amplifiers are generally cascaded to provide increased voltage gain. Operational amplifiers can also be cascaded to give an overall increase in voltage gain. Note that the total circuit gain equals the *product* of the individual gains. The choice of circuit configuration for each stage is chosen to give the required input and output impedance.

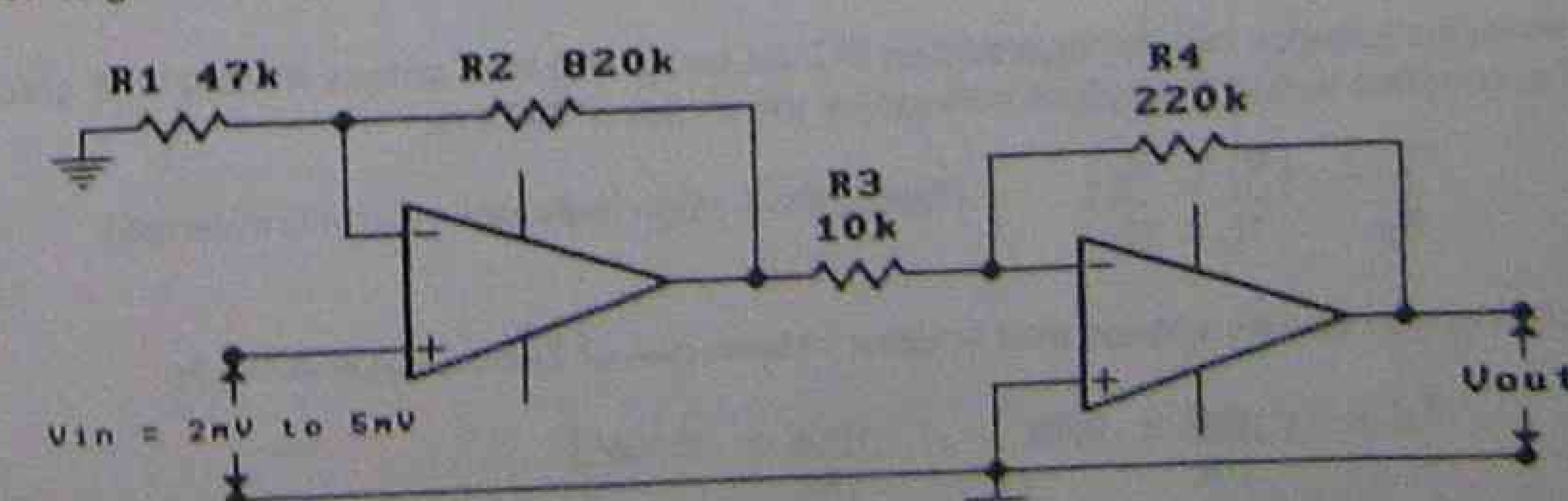


Fig.3: *A Two Stage Op Amp Circuit*

Fig.3 shows a two stage op amp circuit that uses a high input impedance amplifier coupled to a transducer with an output signal that varies over a 24 hour period of between 2 to 5mV. The second stage is an inverting amplifier that increases the overall circuit voltage gain. The high impedance of the first stage (ideally equals infinity) is chosen to give minimum loading to the input transducer.

The input resistance of the second stage is R3, (10k), and its output resistance is (ideally) zero. Because the first stage has virtually zero output resistance, it can drive the the second stage without loss of signal. The total circuit voltage gain can be determined from:

$A_{vtot} = A_{v1} \times (-) A_{v2}$ **Note:** the (-) sign indicates the phase inversion of stage 2.

Thus: $A_{vtot} = [\frac{820k}{47k} + 1] \times (-)\frac{220k}{10k}$

$= (-)\,406$

Thus: $Vo_{(min)} = -406 \times 2mV = -0.81V$

$Vo_{(max)} = -406 \times 5mV = -2.03V$

Slow DC voltage variations of this nature are common in industrial measurements, and small values such as those in the example illustrate the need for multi-stage amplifiers.

4. OUTPUT SIGNAL OF THE INVERTING SUMMER

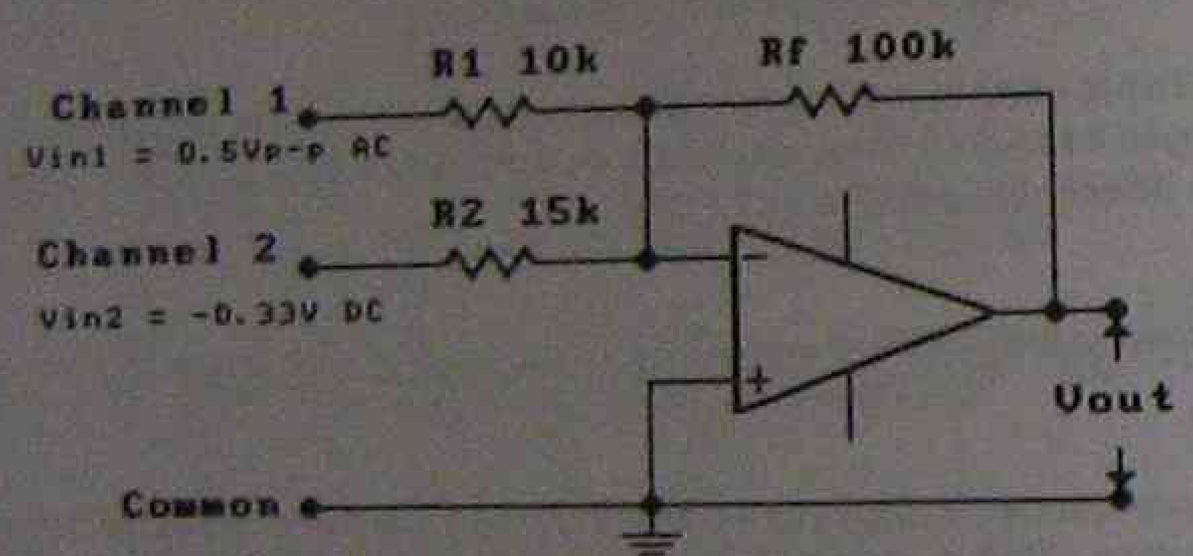


Fig.4: *Combining a DC and an AC signal*

In Fig.4, an AC voltage is applied to channel 1 and a DC voltage is applied to channel 2. The summing circuit will amplify and combine these two voltages to produce a composite output signal that contains a DC and an AC component.

4.1 CIRCUIT ANALYSIS

The two input voltages can be regarded as DC values, giving Vin1 across R1 and Vin2 across R2. The equation to find the output voltage for the circuit of Fig.4 is:

$$\frac{V_o}{R_F} = -[\frac{V_1}{R_1} + \frac{V_2}{R_2}]$$ **(Note:** the (-) sign indicates output is inverted)

Substituting the relevant voltage and resistor values: and transposing

$$V_o = -[\frac{100k}{10k} \times -0.5V + \frac{100k}{15k} \times -0.33V]$$

$$= -5V_{DC} + 2.2Vp\text{-}p$$

The output voltage consists of an amplified and inverted AC component of 2.2Vp-p centred around a -5V DC voltage level as shown in Fig.5.

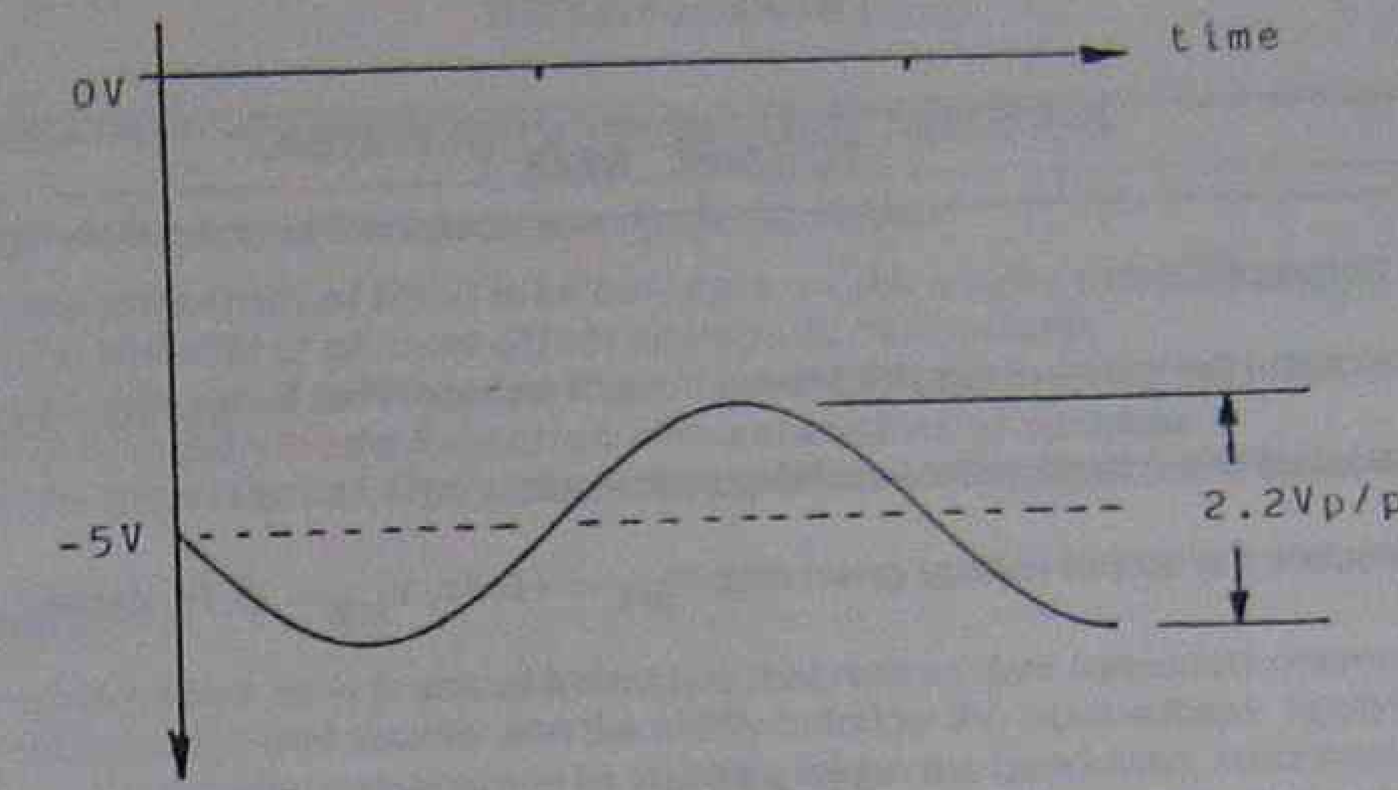


Fig.5: *A Composite Output Voltage*

The resulting output voltage is the amplified sum of the inputs, in which each signal has received different levels of amplification. In this way input voltages can be added to produce one complex output signal in preparation for further electronic processing.

****

TUTORIAL WEEK 7

1. For the circuit of Fig.1:

   (a) Calculate the values of current flowing in each resistor. That is, I_{R1}, I_{R2}, I_{R3} and I_F.

   (b) Calculate the voltage gain of each input channel.

   (c) Calculate the output voltage given that: V_{in1} = +0.4V, V_{in2} = -0.1V, and V_{in3} = +5V.

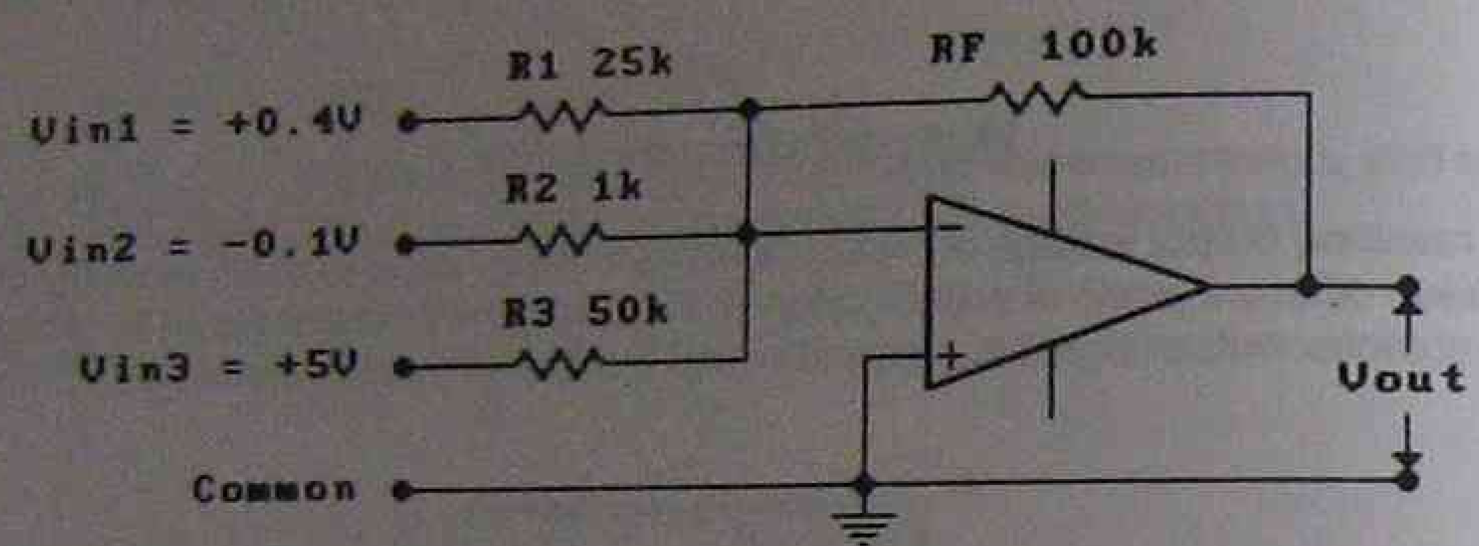


Fig.1.

2. For the circuit of Fig.2:

   (a) Calculate the differential voltage gain of the circuit.

   (b) Calculate the output voltage for the input voltage given on the circuit.

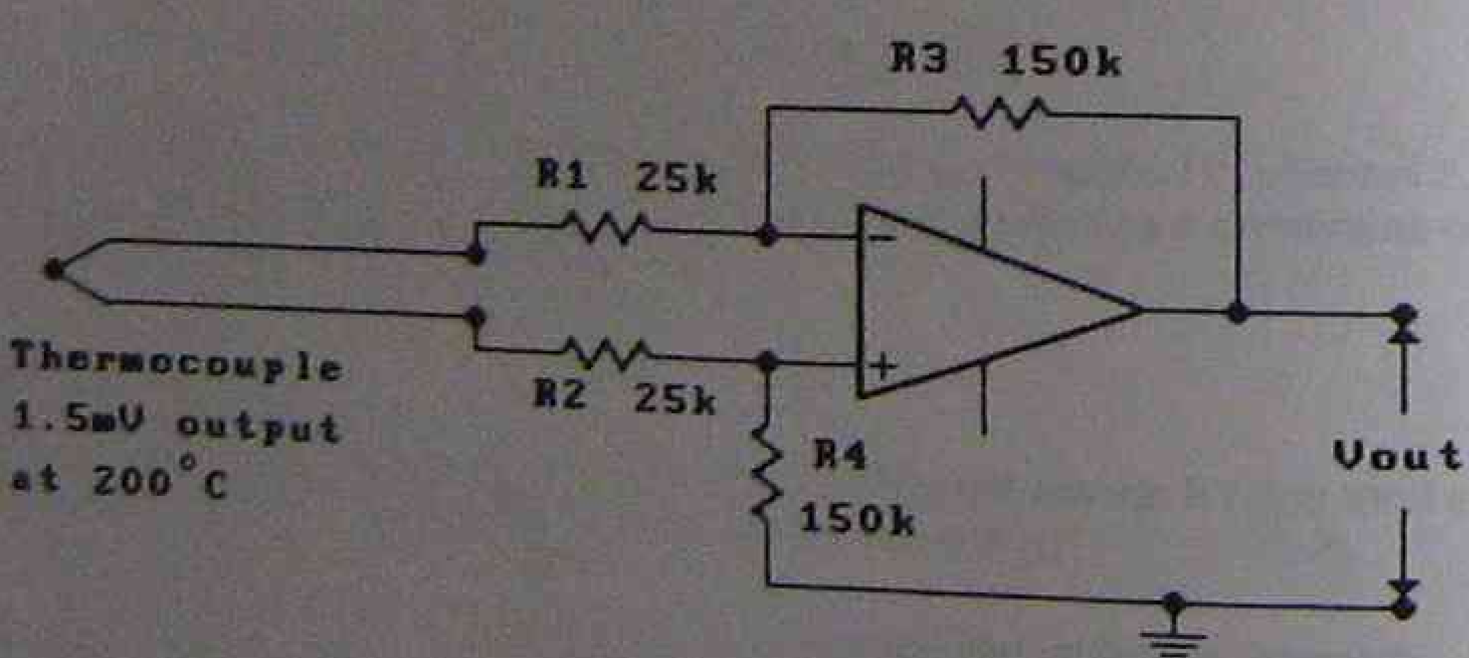


Fig.2.

****

LIMITATIONS OF OPERATIONAL AMPLIFIERS

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

(a) Describe the effect of input bias currents on the output of an op amp.
(b) Describe the effect of input offset voltage for an op amp.
(c) Draw circuits used to minimise the effect of offsets in an op amp.
(d) Identify Common Mode Rejection characteristics of op amps.
(e) Describe the effect of high frequency noise and indicate how it is minimised.

INTRODUCTION

Operational amplifiers have practical limitations that restrict their frequency response, output voltage accuracy, thermal stability and the ability to follow the input voltage. Applying negative feedback to any amplifier can improve its stability, widen the bandwidth, raise the input resistance and lower the output resistance, but at the expense of overall voltage gain. Applying external negative feedback to an op amp circuit achieves the same improvements, and as the open loop gain is very high the loss in gain is also a benefit. The DC circuit associated with an op amp creates other problems due to mismatches in the characteristics of the internal circuitry. As a result, a small DC voltage may be present at the output of the amplifier, even though the input voltage is zero. These notes discuss the practical limitations of operational amplifiers and the methods used to overcome the effects of DC offset.

1. THE INPUT OFFSET VOLTAGE

The *Input Offset Voltage* is the DC differential input voltage that exists internally between the inverting and non-inverting terminals of a practical op amp. Mismatching of the two input transistors in the differential amplifier stages causes this voltage, which is generally in the order of 1mV. As a result, the output can contain a DC offset voltage which is either positive or negative depending on the internal characteristics of the op amp. This output offset occurs because the input offset voltage is being amplified by the gain of the circuit.

Input offset voltage can be nulled by a compensating circuit and some op amps have pin connections that allow an *offset null* potentiometer to be externally connected, as illustrated in Fig.1, providing a positive or negative adjustment to compensate for the effects of the input offset voltage.

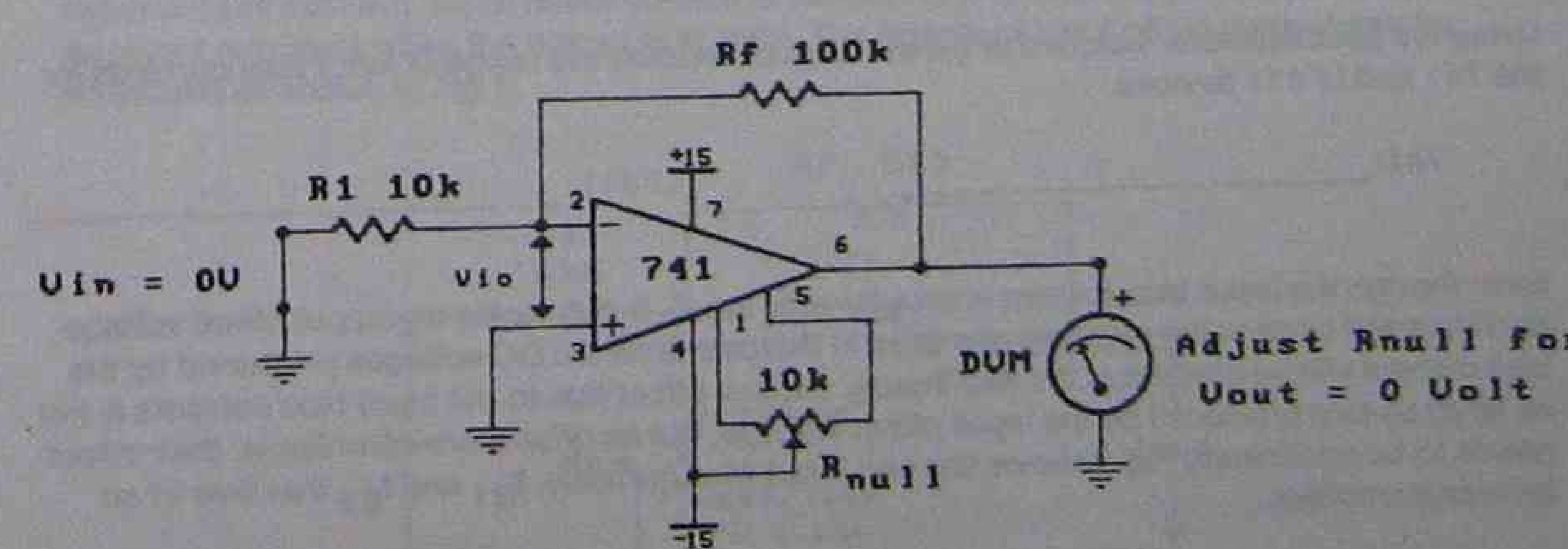


Fig.1: *The inverting amplifier with input offset adjustment*

The offset adjustment is achieved by shorting both inputs to ground and adjusting the potentiometer to give 0V DC at the output, as shown in both Fig.1 and Fig.2.

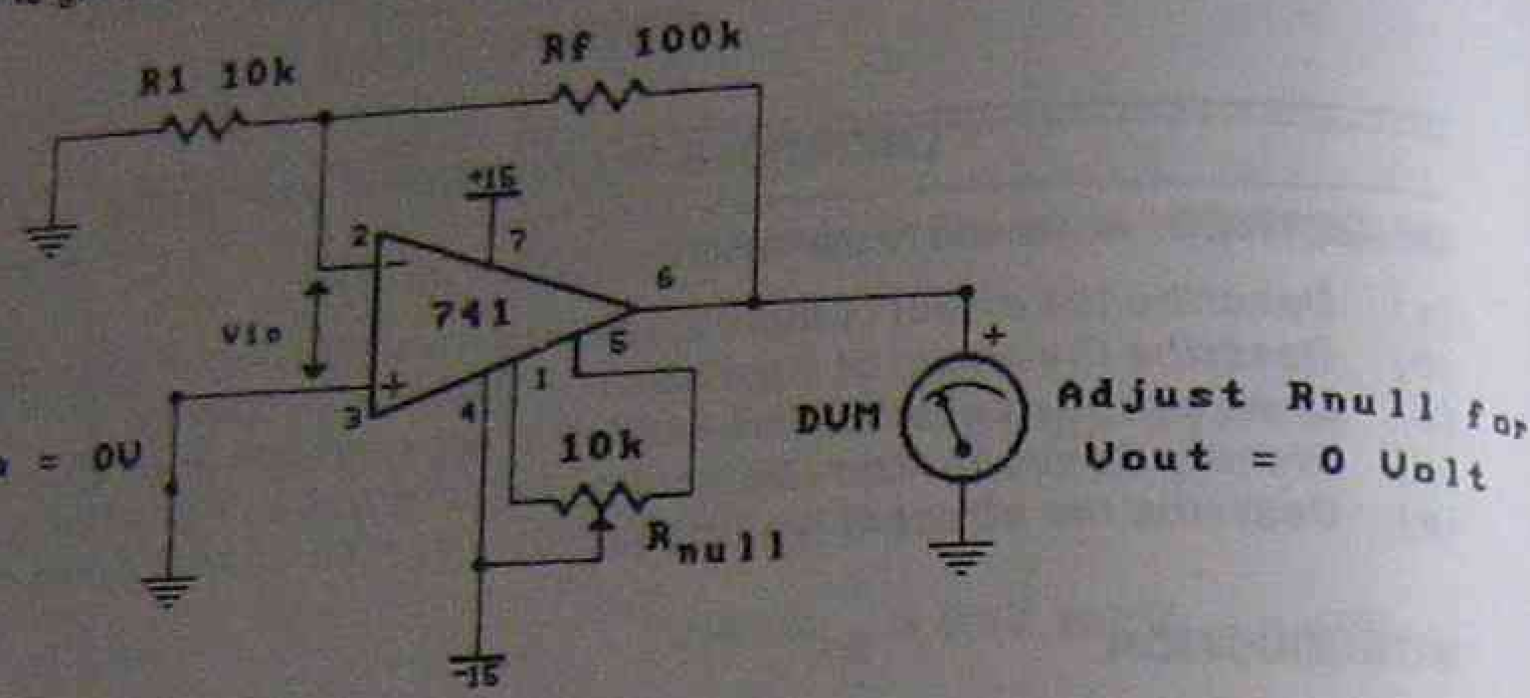


Fig.2. *The non-inverting amplifier with input offset adjustment*

Student exercise:

Using the specifications included in these notes determine the offset voltage range for the 741 and LF411 devices.

741:_______________ LF411:_______________

2. INPUT BIAS CURRENTS - cause O/P voltage not to = yero

Small DC currents, called input bias currents, flow into the input terminals of an op amp via any DC path from that input to ground. The bias currents flowing into each input are generally different in value and an average value is usually stated in manufacturers' specifications. That is:

$$I_B = \frac{I_{B1} + I_{B2}}{2} \quad \text{where:}$$

I_B = the input bias current.
I_{B1} = DC bias current flowing into the non-inverting terminal
I_{B2} = DC bias current flowing into the inverting terminal.

Student exercise:

Using the specifications included in these notes determine the range of input bias currents for the 741 and LF411 devices.

741:_______________ LF411:_______________

Even though the input bias current is usually very small, it can cause an output offset voltage, particularly if large value resistors are used in the circuit as the DC voltages produced by the bias current will be present at the two inputs. Output offset due to the input bias currents is not as large as that produced by the input offset voltage, but as offsets are cumulative, their effect needs to be minimised. Fig.3 shows the two input bias currents, I_{B1} and I_{B2} that flow in an inverting amplifier.

Note that the bias current flowing into the inverting input needs to flow in the parallel combination of resistors R1 and Rf while the bias current current flowing into the non-inverting input has a direct path to ground. Thus, by Ohm's law, a DC voltage will occur at the inverting input, while the non-inverting will have a potential of 0V.

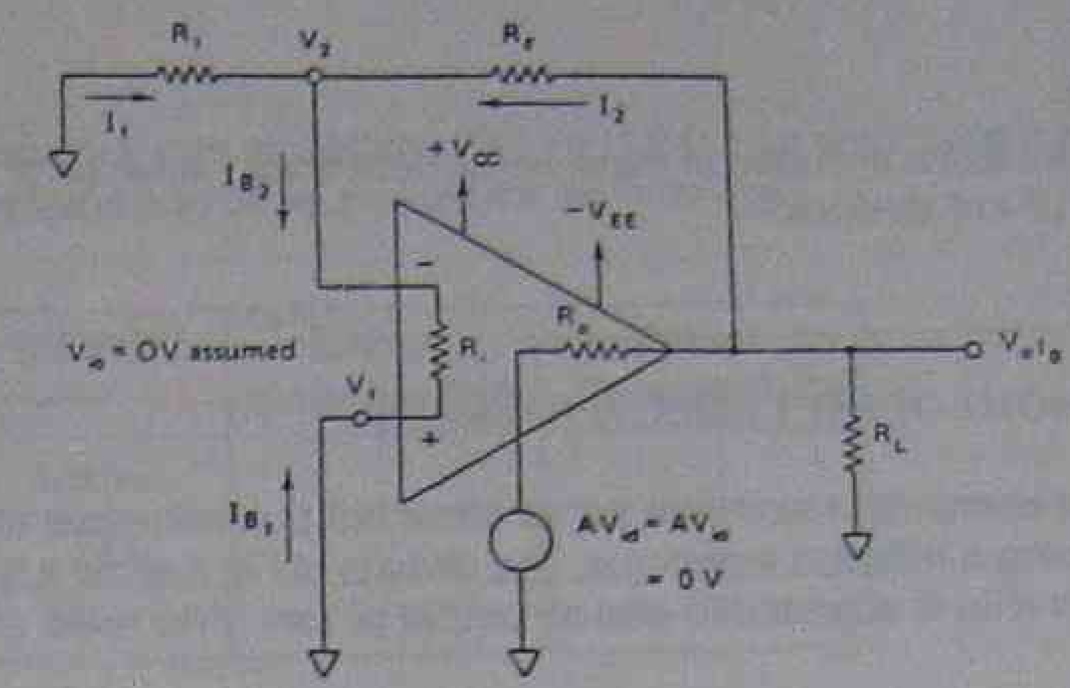


Fig.3. *Output offset voltage caused by input bias currents*

To balance these two voltages a resistor is placed between the non-inverting terminal and ground, as shown in Fig.4. The value of this offset minimising resistor (Req) is equal to the parallel combination of R1 and Rf. That is:

$$R_{eq} = R_1 // R_F = \frac{R1Rf}{R1 + Rf}$$

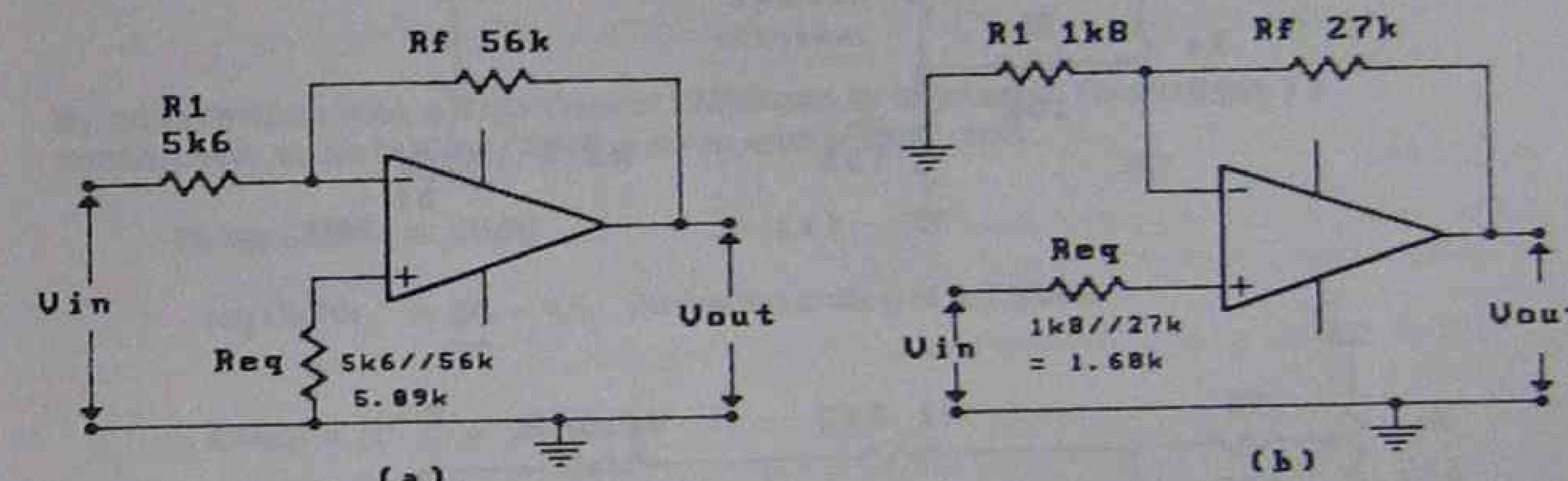


Fig.4. *The non-inverting and inverting amplifiers with bias current offset resistor*

3. INPUT OFFSET CURRENT when O/P Voltage is zero.

How closely the input currents of an op amp match is described in another specification, the *input offset current*. Input offset current is defined as the difference in the currents into the two input terminals when the output is at zero. The effects of input offset current can be minimised as shown in Fig.5.

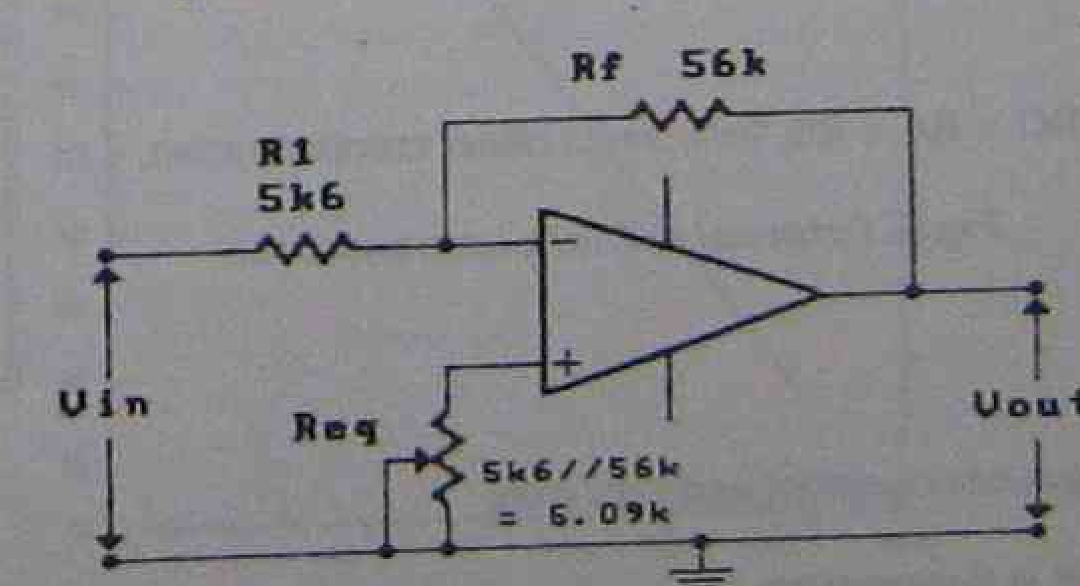


Fig.5. *Inverting amplifier with a trim-pot to minimise input offset current*

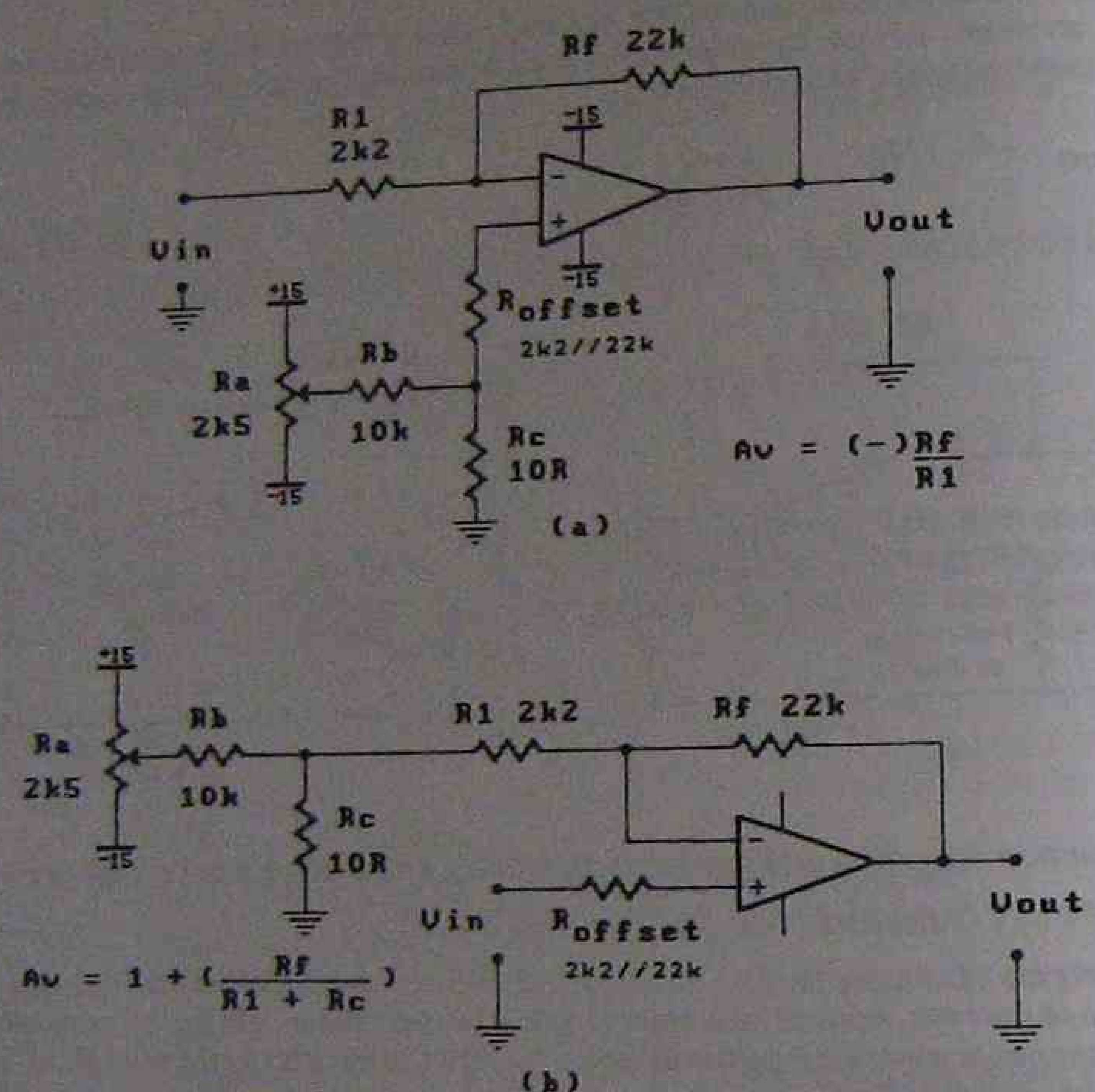
Student exercise:

Using the specifications included in these notes determine the range of input offset currents for the LM301 and LF411 devices.

LM301: ____________ LF411: ____________

4. OP AMPS WITHOUT OFFSET NULL TERMINALS

Op amps without connection terminals for an offset null potentiometer require offset null compensation using a different technique. The circuits of Fig.6 show a typical method, in which the potentiometer (Ra) is adjusted to give an output of zero. (Vin = 0V, at 25°C)



RC < RA < RB, for correct offset compensation.

Fig.6. *External offset compensation*

5. THERMAL DRIFT

The offsets described are not constant and vary with:

(a) Change in temperature.
(b) Change in supply voltages, $+V_{CC}$ and $-V_{EE}$.
(c) Time.

Student Exercise:

Using the attached specifications for an LM301, 741 and LF411 op amp, determine the values of thermal drift in offset voltage and offset current for each device.

DEVICE	Offset Voltage Drift	Offset Current Drift
LM301		
741		
LF411		

5. COMMON MODE REJECTION RATIO

Ideally an op amp only amplifies the differential input voltages (Vid) and it should completely reject common mode input voltages That is, Ac equals 0. However, due to mismatching of devices within the IC, common mode rejection, though usually very high, will not equal the ideal. Manufacturers usually state the common mode rejection ratio, (CMRR), as a value expressed in decibels. The value of CMRR is determined by:

$$CMRR = \frac{Ad}{Ac} \quad \text{or} \quad CMRR = 20 \log \frac{Ad}{Ac} \text{ (in dB)}$$

By using decibels (dB), a large value of CMRR can be expressed. For example, if a manufacturer states that the CMRR of an op amp is 90dB, then:

$$20 \log CMRR = 90dB$$

$$\log CMRR = \frac{90}{20} = 4.5. \text{ Taking the antilog of 4.5 gives:}$$

$$CMRR = 10^{4.5} = 31623$$

This is a typical value of CMRR for an op amp and the higher the value the better. However, as shown in Fig.7, CMRR decreases as frequency increases.

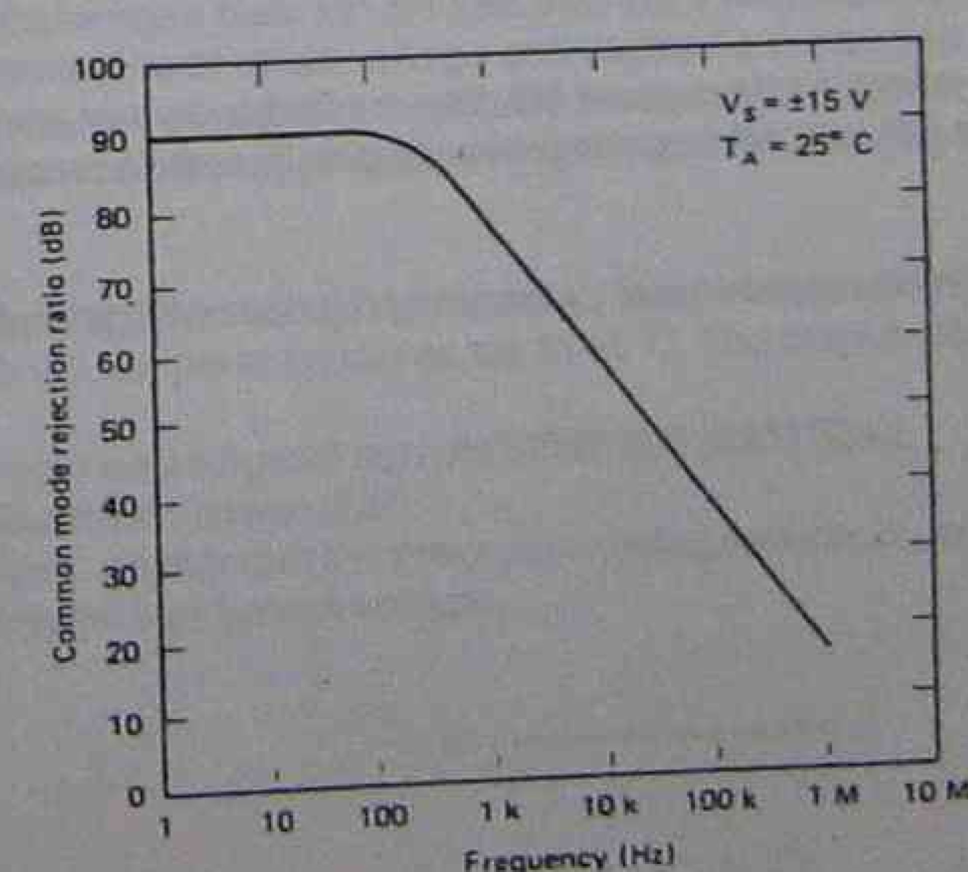


Fig.7: *Common Mode Rejection Ratio versus frequency*

6. NOISE IN THE OUTPUT VOLTAGE

Unwanted electrical signals present in the output signal are classified as *noise*. Drift and offsets can be considered as very low frequency noise, while extraneous signals can vary from 0.01Hz to frequencies over 1MHz. Examining the output of an op amp with a sensitive CRO will show this type of noise voltage, often called *hash*.

Internal noise is generated within any circuit due to random movement of electrons and is dependent on the internal impedances and the temperature.

Noise voltages occuring within the op amp are amplified and appear at the output. Manufacturers specify noise voltages in microvolts(rms) for different values of source resistance over a particular frequency range. For example, the 741 op amp has 2µV of total noise over a frequency range of 10Hz to 10kHz. This noise voltage is valid for source resistors (Rl) between 100Ω and 20kΩ, where Rl includes the input resistor R1 of the basic amplifier configurations. The noise voltage is directly proportional to Rl once Rl exceeds 20kΩ. Thus Rl should be kept below 20kΩ to minimise noise in the output.

Noise voltage is amplified in the same manner as offset voltage. Thus:

$$\text{Noise Gain} = \frac{R_f}{R_1} + 1$$

To keep the noise gain to a minimum place a small bypass capacitor (3pF) in parallel with Rf and keep R1 as small as possible.

7. SUMMARY

1. The input bias currents, input offset current and input offset voltage cause an offset in the output voltage when Vin is zero. Practical DC circuits require offset compensation.

2. The input offset voltage has the greatest affect on the output.

3. The effect of input offset current is minimised by ensuring the DC resistance to ground from each input is equal. However, temperature variations cause these offsets to change.

4. The input offset voltage and input offset current change with time, requiring careful choice of device for best long term stability, and periodic re-adjustment of any compensating network.

5. Input offsets can have an adverse effect on an AC output voltage, by reducing the available peak to peak swing. Thus either positive or negative clipping could occur due to a DC shift away from 0V.

6. Noise can be mimimised in the output signal by avoiding the use of large resistance values where possible.

****

TUTORIAL WEEK 8

1. To adjust the offset voltage of an op amp, a voltmeter should be connected between ground and:

   (a) The inverting input.
   (b) The noninverting input.
   (c) The output.
   (d) The offset terminal.

2. For the circuits of Fig.1, determine the value of the offset minimising resistor.

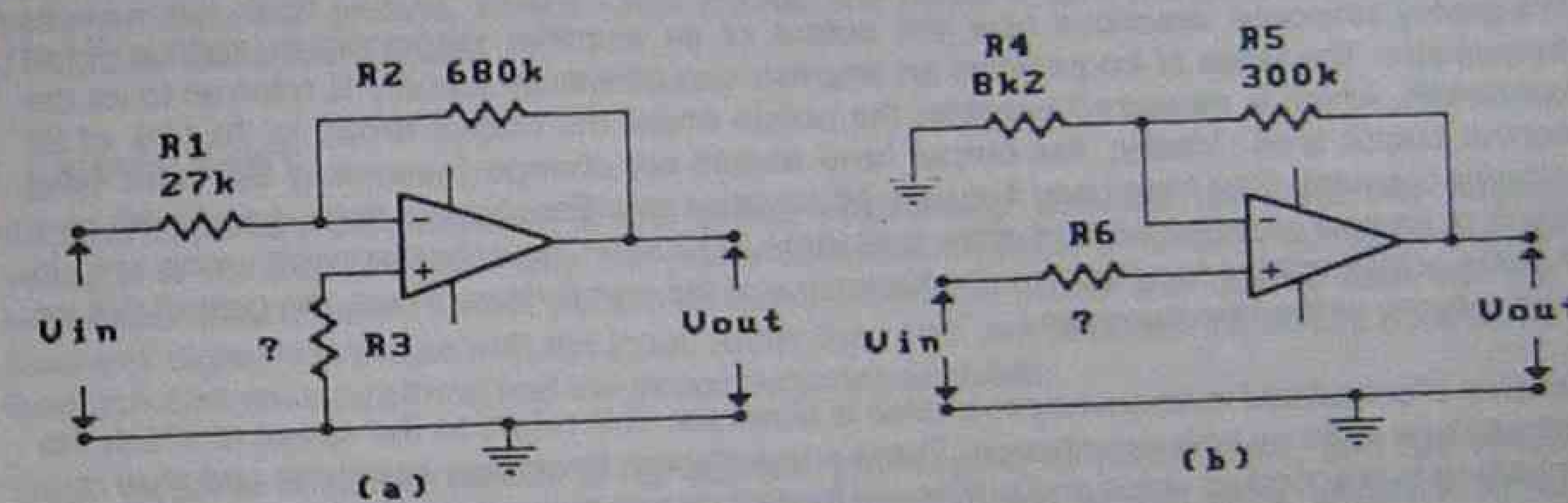


Fig.1.

3. A large resistance value for R_F will:

   (a) Increase input bias current and decrease the output offset voltage.
   (b) Decrease input offset voltage and decrease the output offset voltage.
   (c) Decrease input offset current and increase the output offset voltage.
   (d) Increase offset null adjustment and decrease the output offset voltage.

4. An op amp circuit correctly adjusted at room temperature is operating inside an instrument case in which the temperature increases to 45°C. The output offset will:

   (a) Remain unchanged if correct offset null pot is used.
   (b) Suffer from thermal drift.
   (c) Remain unchanged if correct minimising resistor is used.
   (d) Raise the input offset voltage.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS, 6016C

THEORY LESSON 9

OP AMP CHARACTERISTICS

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

(a) *Sketch a typical frequency response curve of a compensated op amp.*

(b) *Calculate the bandwidth of an op amp circuit, given the unity gain bandwidth and the gain of the circuit.*

(c) *Define slew rate and describe its effect on the output signal of an op amp.*

(d) *Calculate the time taken for the output voltage of an op amp to reach its peak value, given the slew rate and the peak to peak value of the waveform.*

(e) *Calculate the maximum frequency of a specified output signal level given the slew rate of the op amp.*

INTRODUCTION

Frequency response describes how the output of an amplifier responds to various input frequencies. The range of frequencies an amplifier can effectively amplify is referred to as the bandwidth, which is measured between the points where the output drops to 70.71% of its normal output level. Ideally, the output level should not change (assuming the input level remains constant) if the frequency if varied. All practical amplifiers have a finite bandwidth as a result of internal capacitance and other limitations. The most typical effect with an op amp is that the output voltage level falls as the frequency of the signal rises. Thus, the gain of the op amp reduces as the frequency rises.

Another effect related to frequency response is slew rate. This refers to the speed at which the output stage of an op amp can change. These notes discuss frequency response and slew rate limitations in practical op amps and how these limitations can be improved.

1. FREQUENCY RESPONSE OF AN OP AMP

Frequency response (or bandwidth) of an amplifier can be improved with the addition of feedback. An amplifier without feedback is operating in open loop, and will have a maximum gain as a result. The graph of Fig.1 shows how the bandwidth of an op amp is related to the open loop gain of the amplifier. Note that as the gain reduces, the bandwidth increases.

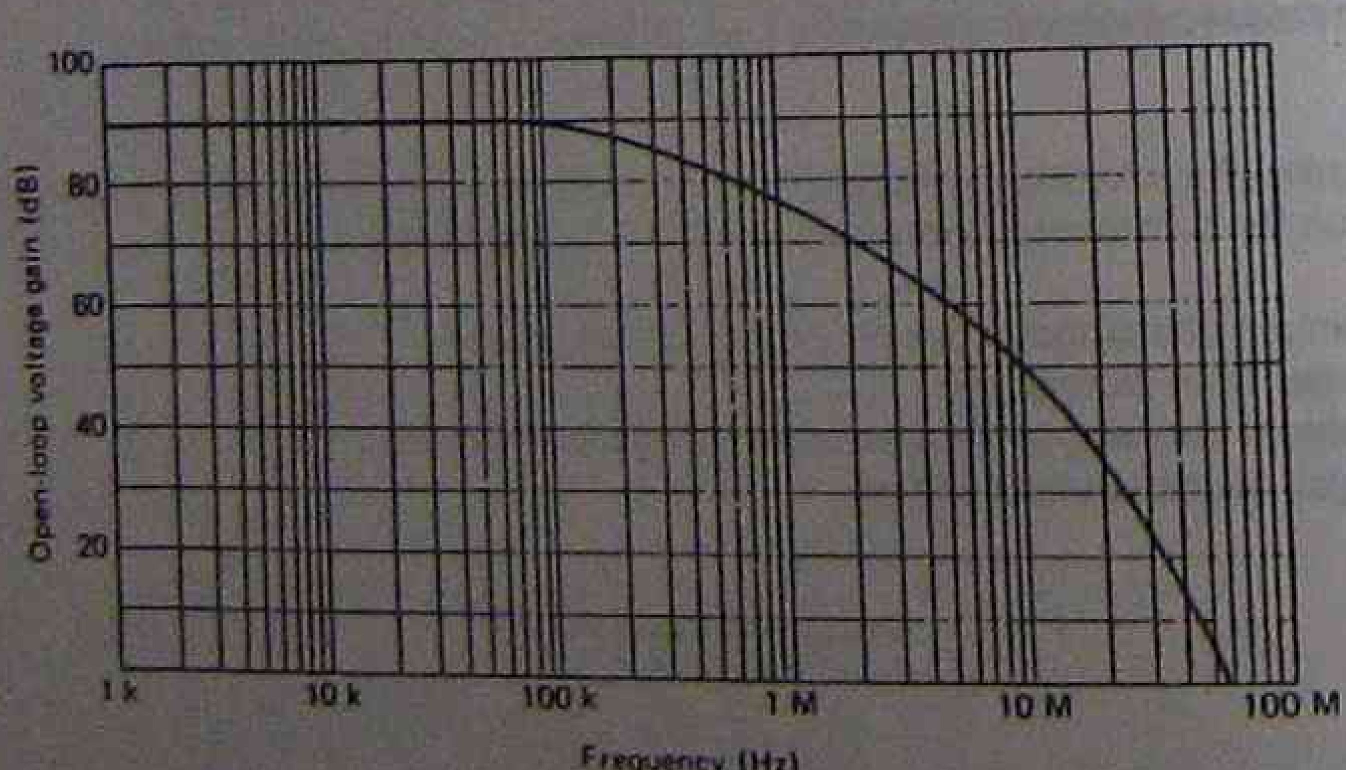


Fig.1: *Open-loop gain vs frequency response of a typical op amp*

There are two points to note about Fig.1:

(1) The output level is expressed in *decibels*.

(2) The horizontal axis is calibrated *logarithmically*.

The dB values can be changed to absolute values as shown: From Fig.1, the gain at 600kHz is approximately 80dB.

As Av (in dB) = 20 log Av Therefore: 80dB = 20 log Av

dividing both sides by 80 gives: $\log Av = \frac{80}{20}$ or log Av = 4

Taking the antilog of both sides: Av = antilog of 4, which can be found by $10^{4.0}$.

giving **Av = 10000**

From Fig.1, the open loop gain has decreased from about 32000 at a frequency of 100kHz to 10000 at 600kHz. This illustrates that the bandwidth, or range of frequencies the op amp can effectively amplify widens as the gain is reduced, which in turn is reduced as more negative feedback is applied. **NOTE: The bandwidth of an amplifier is that range of frequencies between the -3dB points, where -3dB equals the point where the output has fallen to 71% of its mid band value.**

1.1 PHASE SHIFT

Another important consideration is the phase relationship between the input and output voltages of the amplifier. Normally, the phase difference is either 0° or 180°, but as frequency increases, this can alter. Under certain circumstances, the phase shift can be such that the feedback signal is in phase with the input, rather than 180° out of phase. When this occurs, the feedback becomes *positive*, and the circuit becomes unstable.

Thus, negative feedback needs to be applied with regard to phase shift, and too much feedback can produce instability. To prevent instability, but to still allow feedback to be applied, compensation in the form of capacitance is required. There are two types of op amps: the internally compensated (most common) and the externally compensated.

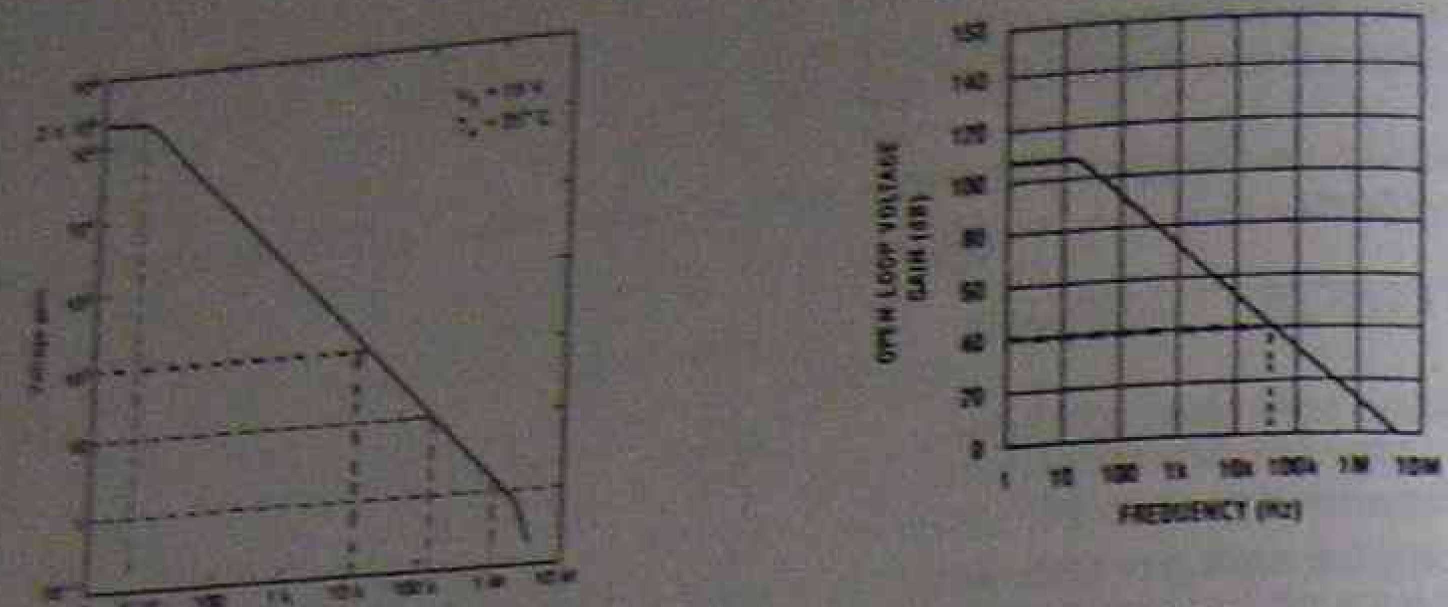
2. INTERNAL FREQUENCY COMPENSATION

Circuits that do not require wide bandwidths generally use internally compensated op amps, such as the 741, TL071, TL081 and LF411. An internal compensation network is used to tailor the frequency response for a reasonable range of frequencies at moderate closed loop gains (gain with feedback), providing protection against high frequency oscillation. Compensated op amps have an internal capacitor to control the shape of the frequency response curve. The gain versus frequency relationship is thus based on X_C where:

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

If the frequency of the input signal applied to the op amp increases by a factor of ten, Xc decreases by a factor of ten. Thus, the open loop gain of the op amp decreases by a factor of ten, affecting the closed loop gain by causing the gain to *roll off* as shown in Figs.1 and 2. The 10 times change in frequency is referred to as a *decade change* and if the compensation network only contains one capacitor, the roll off will be at a rate of a reduction in gain by a factor of ten for each decade increase in frequency. This works out to a change in gain at a rate of *20dB per decade*, as shown in the curve for the LF411 device in Fig.2.

Note that the curves start at the value of open-loop gain and decrease to a voltage gain of 1, or 0dB - also referred to as unity gain. Some manufacturers do not provide this graph in their specification sheets, instead they often provide a figure called the *gain-bandwidth product*.



741 LM411

Fig.2: *Frequency response curves for a 741 and LF411 op amp*

3. GAIN-BANDWIDTH PRODUCT

The *unity gain bandwidth* (UGB) of an internally compensated op amp is the bandwidth of that op amp when its closed loop gain is 1, or unity. This figure allows the bandwidth of an op amp to be calculated for any gain, in which the specified gain is divided into the unity gain bandwidth figure. That is, the gain multiplied by the bandwidth at that gain (gain bandwidth product or GBP) will equal the unity gain bandwidth figure.

GBP = Av(closed loop) x BW = unity gain bandwidth

For example, as shown in Fig.2, the UGB for a 741 is 1MHz, and 4MHz for the LF411. These curves also indicate the circuit bandwidth for any given closed-loop voltage gain. For example, if these two op amps are operating with a closed loop gain of 100 (shown dotted), the bandwidth for each op amp can be read from the curves as being 10kHz for the 741, and 40kHz for the LF411. Thus:

741: UGB = 1MHz

$$BW = \frac{1MHz}{100} = 10kHz$$

LF411: UGB = 4MHz

$$BW = \frac{4MHz}{100} = 40kHz$$

4. SLEW RATE

The slew rate specification of an op amp states how fast the output voltage of an op amp can change for a step change at the input. This value is usually stated for unity gain and is expressed in volts per microsecond.

$$\text{Slew Rate} = \frac{\text{Max change in Vo}}{\text{Change in time}}$$

For example, a slew rate of 5V/µs means that the output voltage can rise or fall no faster than 5 Volts in one microsecond. Ideally the slew rate should be infinity, meaning the output voltage would rise or fall simultaneously with a change in the input voltage. Practical op amps are available with slew rates which range from 0.1V/µs to well above 1000V/µs. The National Semiconductor LH0063C has a slew rate of 6000V/µs.

The unity gain-bandwidth is the small-signal, high frequency limitation of the op amp. Slew rate is a large signal parameter, affecting the maximum changes in output voltage. Generally small signals fall in the millivolt range, whereas large signals swing by several volts. Slew rate is affected by internal current limiting and the saturation of internal stages of an op amp, when a high frequency, large amplitude voltage signal is applied. The current is the maximum internal current available to charge the compensation capacitance network. Since a capacitor takes a finite amount of time to charge and discharge, these networks prevent the output from responding immediately to a fast change at the input.

4.1 SLEW RATE LIMITING OF SQUARE WAVES

Fig.3 shows the effect of slew rate on a square wave.

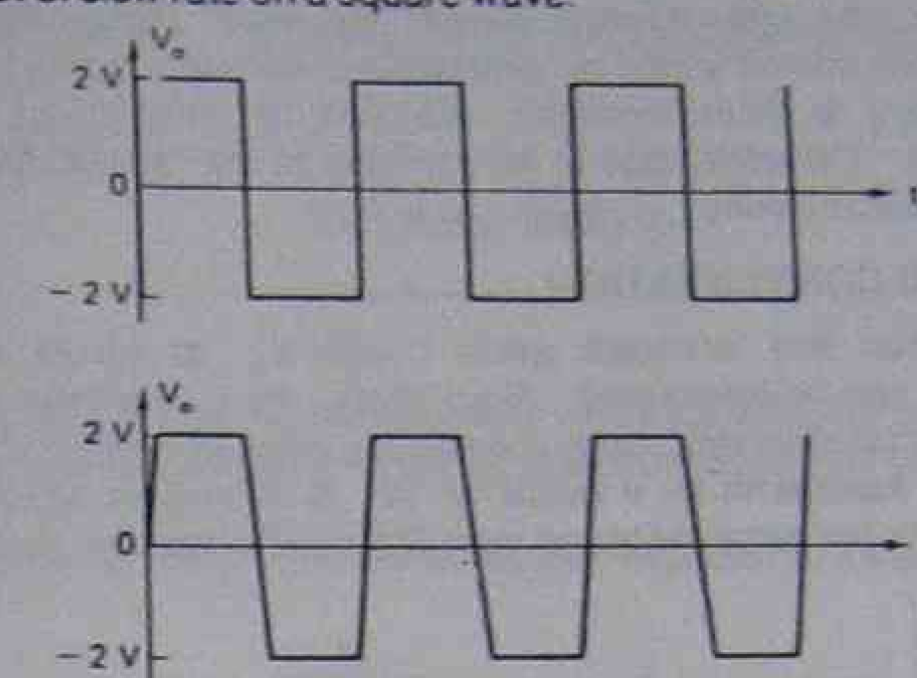


Fig.3: *Effects of slew rate on a square wave*

If the frequency of the input waveform is high, the effect of slew rate will be more noticeable. For example, if a 2Vp-p square wave at a frequency of 50kHz is applied to a voltage follower (gain of one) using an op amp with a slew rate of 0.5V/µs then:

(1) period (T) of the waveform = 1/f = 1/50kHz = 20µs

(2) time taken for the output to change by 2V $= \frac{\text{voltage change}}{\text{slew rate}} = \frac{2V}{0.5V/\mu s} = 4\mu s$

The waveform for this output signal will be almost a triangular wave, as the total time taken for both edges to complete their transitions is 8µs, which is almost the time required for each half cycle of the waveform (10µs).

4.2 SLEW RATE LIMITING OF SINEWAVES

Fig.4 illustrates the effect of slew rate on a sine wave.

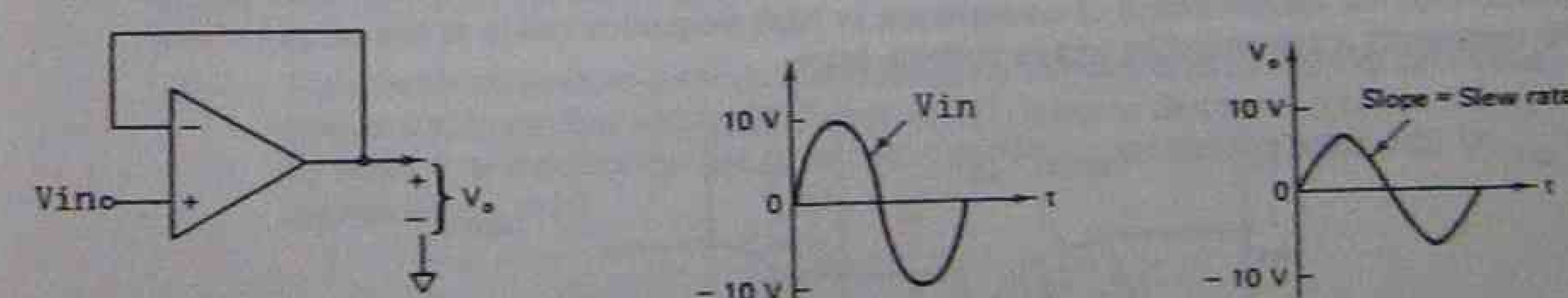


Fig.4. *Slew Rate limiting of a sine wave*

The output tries to follow Vin, but is distorted because of slew rate limiting. The maximum frequency of a sine wave at which an undistorted output can be obtained is found by:

$$f_{MAX} = \frac{\text{Slew Rate}}{2\pi \times Vo_{max}}$$

EXAMPLE: The slew rate of a 741 is 0.5V/µs. If it is configured as a voltage follower determine the maximum frequency that can be obtained from the circuit for the following output voltages:

(a) A 20Vp-p sinewave. (V_{max} therefore equals 10V)
(b) A 2Vp-p sinewave. (V_{max} therefore equals 1V)

(a) $f_{MAX} = \frac{0.5}{6.28 \times 10V \times 1\mu s} = 7.96kHz$ (b) $f_{MAX} = \frac{0.5}{6.28 \times 1V \times 1\mu s} = 79.6kHz$

Hence, if large output voltage swings are necessary, the slew rate will be the limiting factor.

5. EXTERNAL FREQUENCY COMPENSATION

Internally compensated op amps are usually very stable, but have limited small-signal bandwidth, slow slew rate and reduced power bandwidth (large signal response). An alternative to the internally compensated op amp is the *externally compensated* type. These op amps have from one to three terminals available for connecting external frequency compensating networks. The advantage is an increase in the bandwidth and speed of slew rate. The tradeoff is reduced stability.

5.1 SINGLE CAPACITOR COMPENSATION

The LM301 op amp, has two terminals (pins 1 and 8), to which a single frequency compensating capacitor can be connected. Fig.5 shows the gain-frequency response graph for two values of C1. With an externally compensated op amp, increasing C1 by a factor of 10, reduces the small-signal bandwidth by a factor of 10. A unity gain bandwidth of 10MHz is attainable with C1 = 3pF or the same response as a 741 can be attained by making C1 = 30pF.

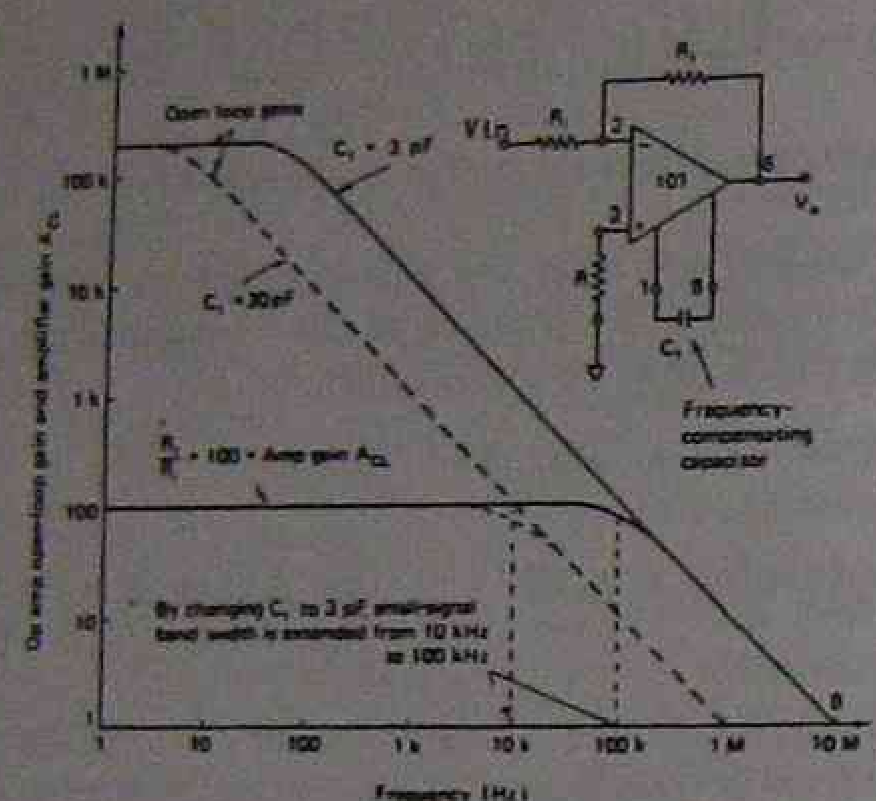


Fig.5: *Extended frequency response with external capacitor with a 301 op amp*

5.2 FEED-FORWARD COMPENSATION

This form of compensation, shown in Fig.6, will increase the slew rate and full power bandwidth, with the disadvantage of an increase in high frequency noise at the output. Note that the bandwidth is increased for higher voltage gains.

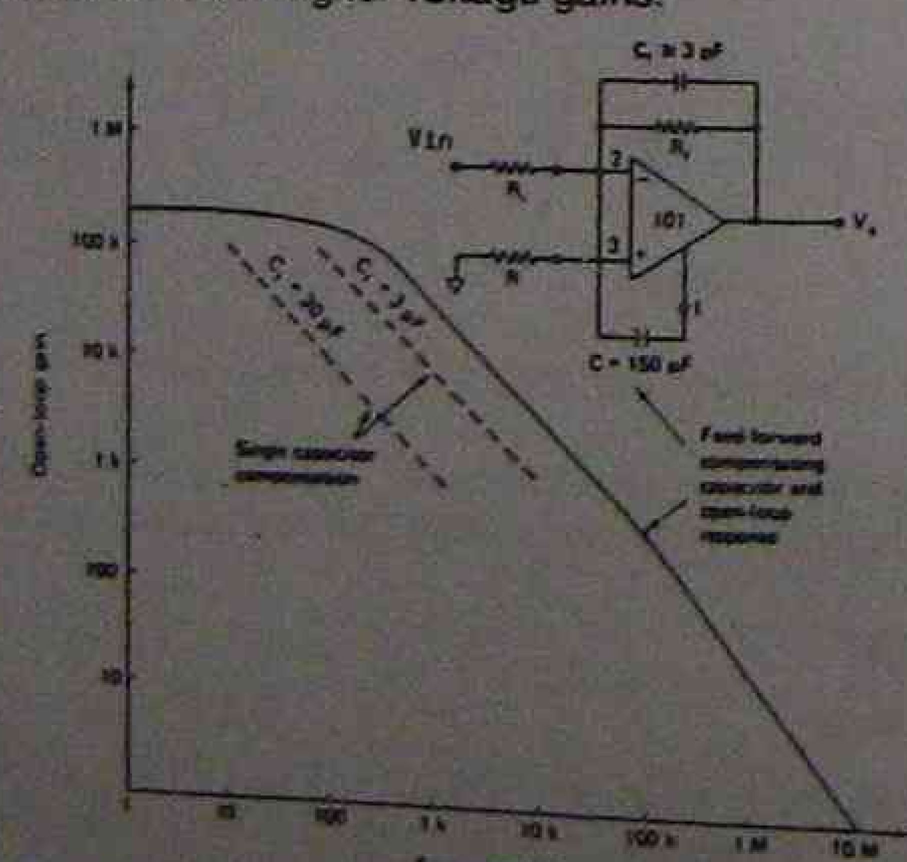


Fig.6: *Extended bandwidth with feed-forward capacitor*

External frequency compensation can improve bandwidth, but will increase the level of high frequency noise and give an increased risk of circuit oscillation.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

TUTORIAL WEEK 9

---

1. Determine the bandwidth of an inverting amplifier with a closed loop gain of (-)150, that is constructed with an internally compensated op amp that has a unity gain bandwidth of 2MHz.

2. Determine the slew rate of the op amp whose output response to a square wave is shown in Fig.1.

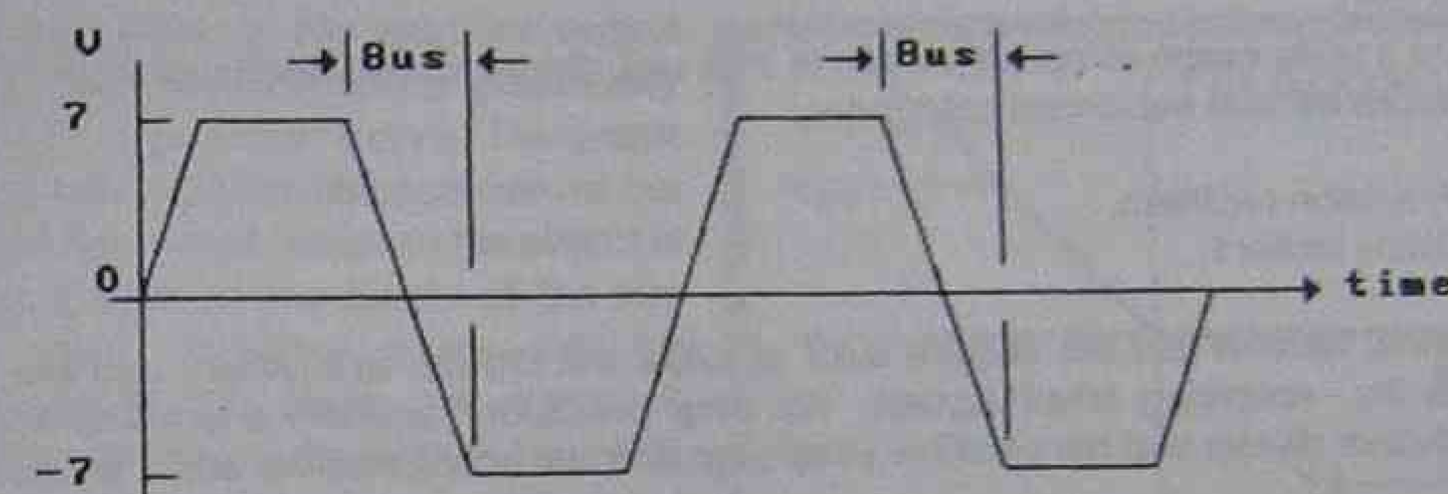


Fig.1.

3. The circuit of Fig.2 is a non-inverting amplifier constructed with a 741 op amp. If the slew rate of the 741 is 0.5V/μs and its unity gain bandwidth is 1MHz, determine:

   (a) The bandwidth of the circuit.

   (b) The maximum peak-to-peak output voltage for the circuit at its highest operating frequency, assuming a sinewave input. (HINT: Refer to Section 4.2 of the theory notes for week 7, and rearrange the equation to put Vo_{max} on the left hand side. Vo_{p-p} equals Vo_{max} x 2.)

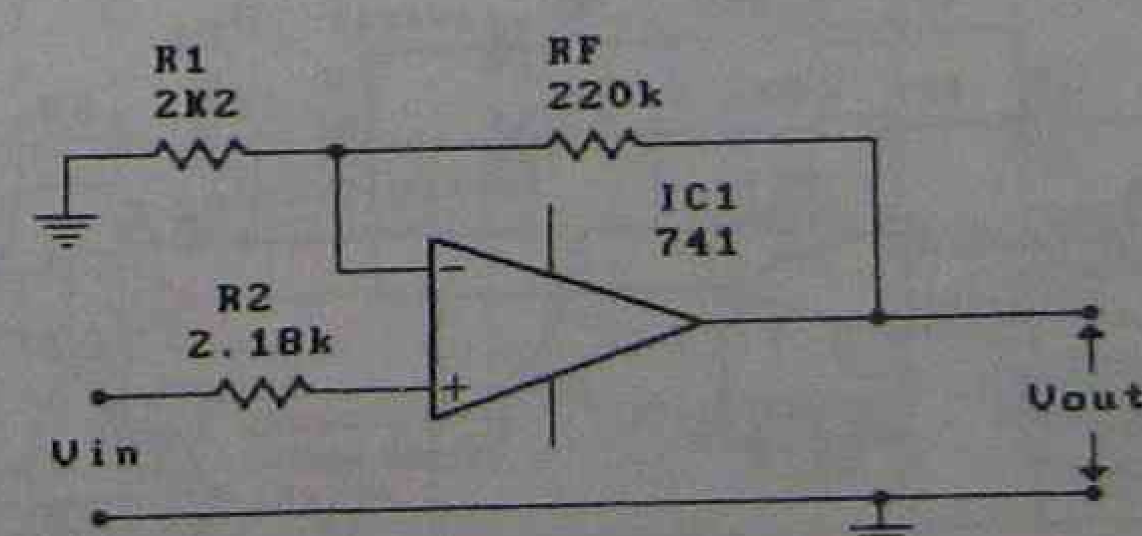


Fig.2.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS, 6016C

THEORY LESSON 10

OP AMP DIODE CIRCUITS

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

(a) *Identify the circuits of a precision half-wave and full-wave rectifier.*
(b) *Sketch the output waveform of a precision rectifier (half-wave and full-wave).*
(c) *Sketch the output waveform of a range of limiter circuits.*
(d) *Describe the basic operation of a peak detector.*

INTRODUCTION

There are a wide range of op amp circuits that use diodes in the feedback path. In general, these circuits fall into two broad categories:

- Precision rectifiers.
- Diode limiters.

Conventional rectifier circuits require 0.6V to allow the diodes to conduct and are therefore unsuitable for rectifying small signals. Op amp precision rectifiers overcome the forward voltage drop of diodes and hence allow small signals to be processed for applications such as:

- precision half-wave rectification.
- precision full-wave rectification.
- averaging circuits.
- peak value circuits.
- precision bound circuits or clippers.

Diode limiters 'bound' input signals by keeping their peak-to-peak values within restricted limits, before an output reaches saturation or in preparation for further signal processing. Some of these circuits are described in these notes.

1. THE PRECISION HALF-WAVE RECTIFIER:

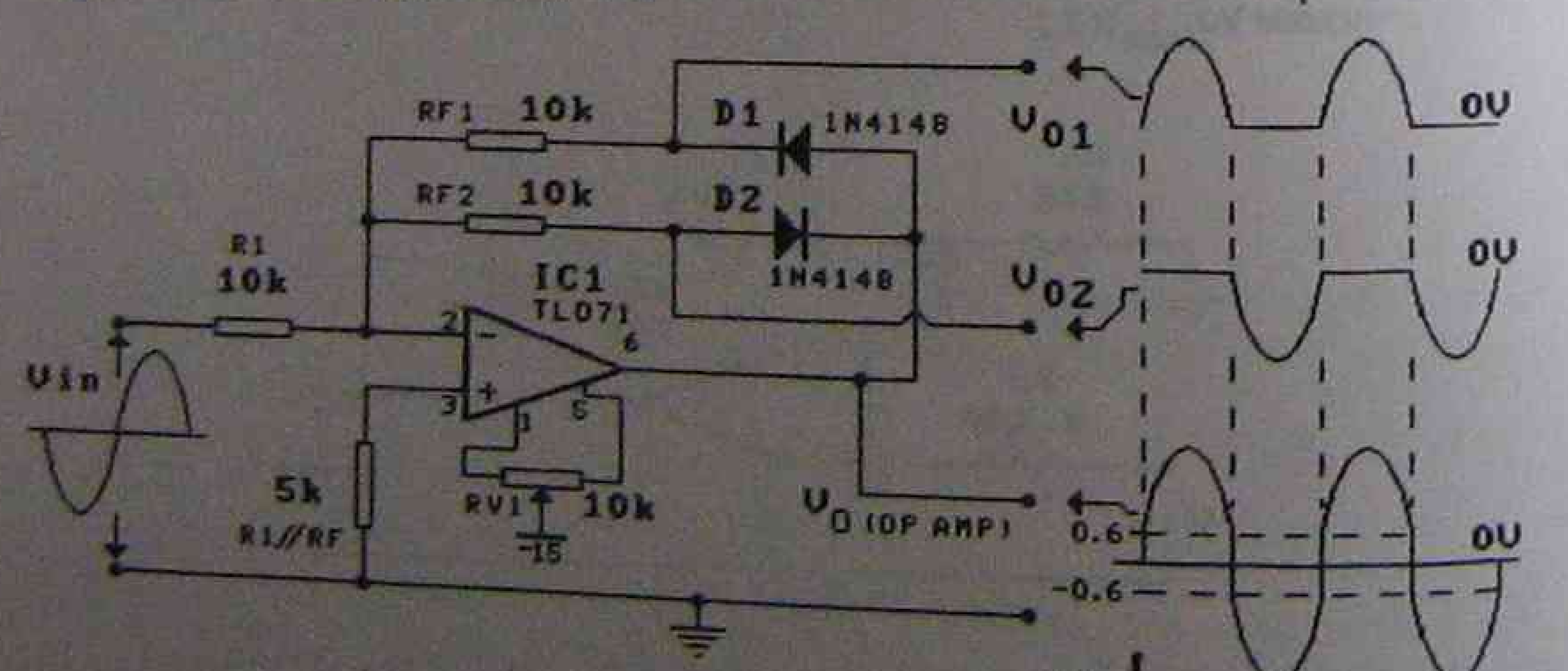


Fig.1: *A precision half-wave rectifier.*

The two outputs of Fig.1 produce half-wave rectified outputs, independent of diode forward voltage drop.

1.1 OPERATION FOR THE PRECISION HALF-WAVE RECTIFIER

When the input signal is positive, the output of the op amp will be negative. Under these conditions, D1 is reverse biased and D2 will be forward biased when the output of the op amp reaches -0.6V. Before the output has forwarded biased either diode, there will be no feedback, and the op amp will operate at maximum gain. Thus, the input signal only needs to be slightly positive for the output to swing to -0.6V. When this occurs, D2 becomes a closed switch, and the negative half cycle of the output will appear at V_{O2}.

When the input signal is negative, the same thing occurs, except D1 is forward biased and D2 is reverse biased. Thus the positive half cycle of the output signal appears at V_{O1}.

The output of the op amp therefore provides the 0.6V forward bias for either diode, allowing the input signal to be rectified with no losses.

1.2 TRANSFER CHARACTERISTIC OF THE HALF-WAVE RECTIFIER

The relationship between the input and output signal(s) of a circuit can often be graphically represented by a *transfer curve*. The graph of Fig.2 shows the transfer characteristic of the half-wave rectifier circuit, in which the output is a line passing through the center of the two axes. The graph therefore shows that for positive values of the input, the output is negative, with a direct relationship to the input. Similarly, for negative input signals, the output is positive, again with a direct relationship to the input. The important point to note is that the output responds to voltages that are very close to 0V, unlike a conventional rectifier, which only responds when the input signals exceeds 0.6V in either direction.

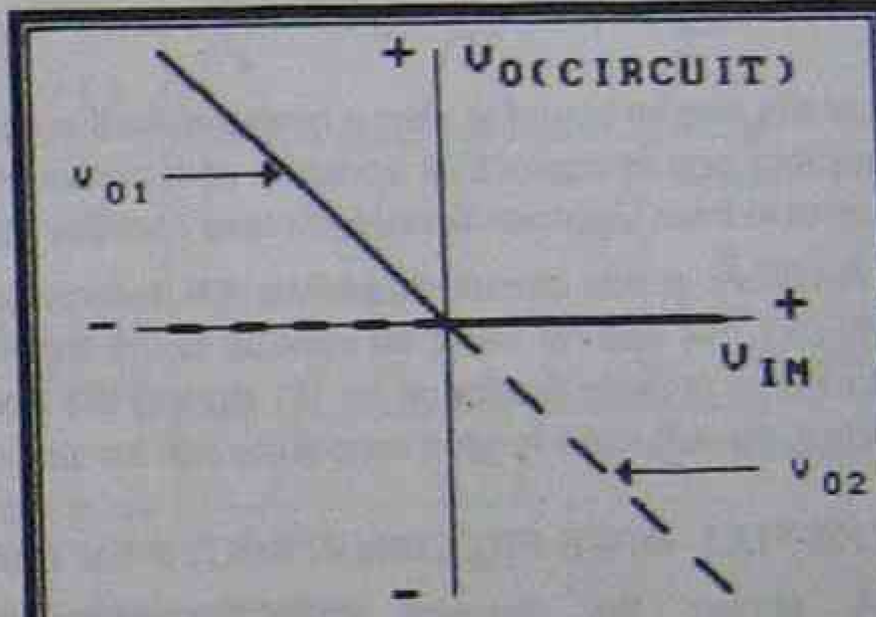


Fig.2: *Transfer Characteristic of the Half-Wave Precision Rectifier*

2. THE PRECISION FULL-WAVE RECTIFIER:

A precision full-wave rectifier can be constructed using a summing inverter with one of its inputs connected to an output of a precision half-wave rectifier and the other to the input signal, as shown in Fig.3.

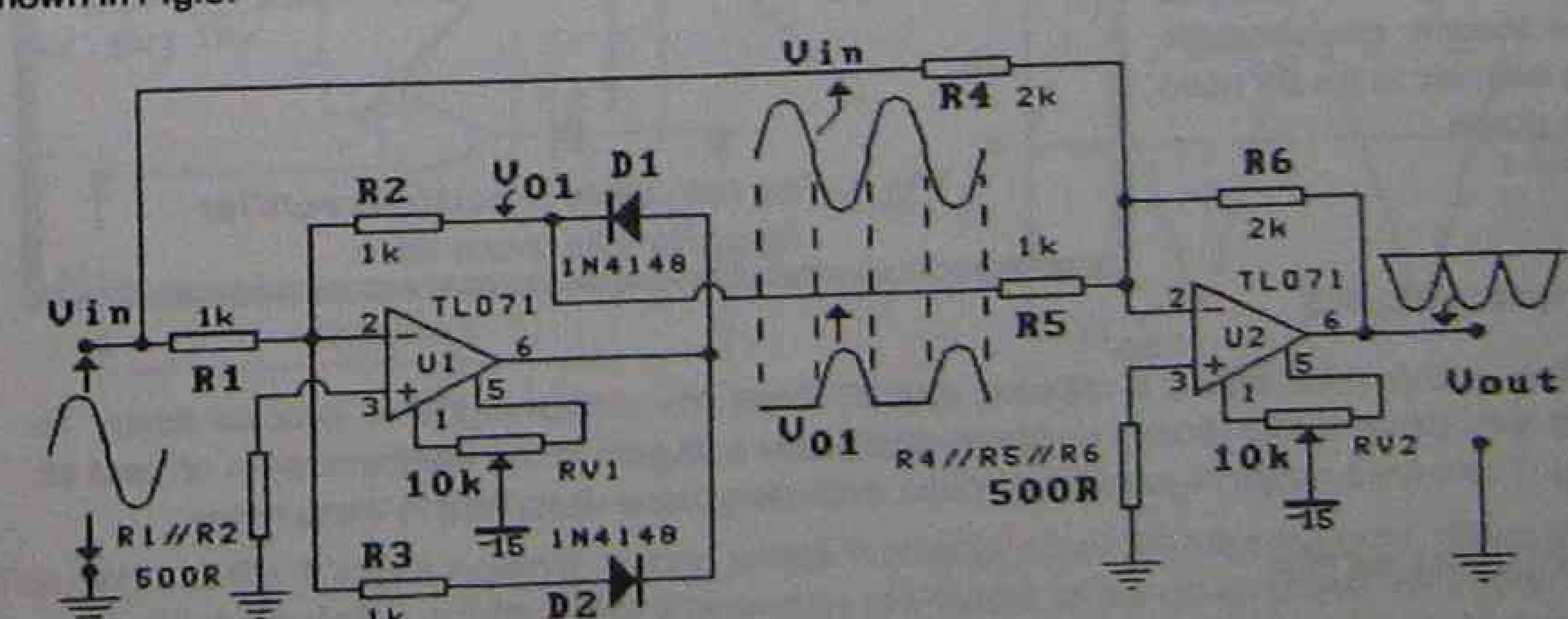


Fig.3: *The precision full-wave rectifier.*

In the circuit of Fig.3, U2 is the inverting summer and U1 the half-wave rectifier circuit as in Fig.1. On the positive half cycle of the input waveform, the output from the half-wave rectifier at the cathode of D1 (Vo1) will be 0V. Thus the signals applied to the summer are 0V and Vin, in which Vin is amplified by a factor of 1. The output of the summer will therefore equal:

- $V_{out} = (-)[V_{in} + 0V]$, which equals $-V_{in}$.

The output of the circuit (Vout) will therefore be the inversion of the positive half cycle of the input signal, giving a negative going half cycle as shown.

When the input is negative, the two signals applied to the summer are still the output of the half-wave rectifier (Vo1) and the input signal (Vin). The output of the half-wave rectifier (Vo1) is now a positive going half cycle, which is amplified by a factor of two by the summer. The input signal, (which is in its negative half cycle), is also amplified by the summer, but by a factor of 1. The output of the summer therefore equals:

- $V_{out} = (-)[2V_{o1} - V_{in}]$, which equals $-V_{in}$ as V_{o1} has the same amplitude as V_{in}.

Thus the output signal is also a negative half cycle, with the same amplitude as before, giving a complete output signal that consists of a series of negative half cycles. This is the same type of waveform from a conventional full-wave rectifier, except this circuit can rectify small signals.

All resistors in this circuit should be 1% tolerance to ensure exact cancelling as described. In particular, R4 and R6 must be exactly twice the value of R1, R2 and R5. Diode D2 and R3 are included to provide feedback for U1 during the positive half cycle of the input. Also, input offset needs to be adjusted to give zero volts out for zero volts in.

2.1 THE FULL-WAVE PRECISION RECTIFER TRANSFER CHARACTERISTIC

Fig.4 shows the transfer characteristic of the precision full-wave rectifier. As shown, the output is inverted when the input is positive (180° inversion), while for negative input signals the output is not inverted (0° inversion). Input signals down to virtually zero volts are amplified, indicated by the transfer characteristic which extends to the 0V point on the graph.

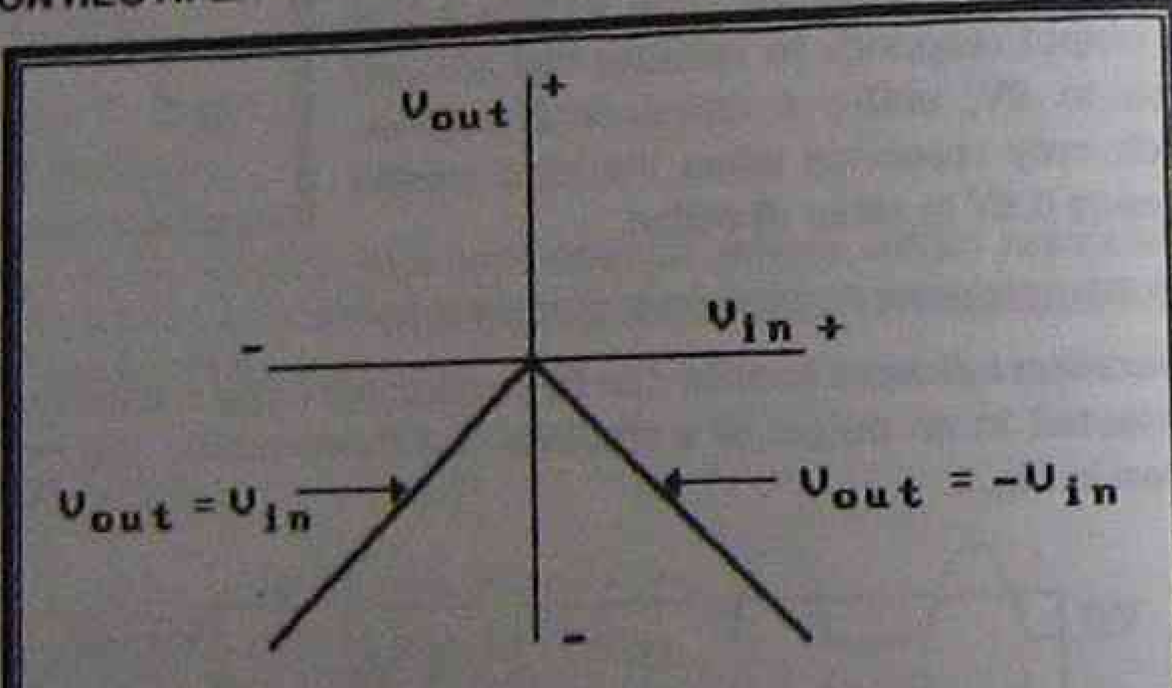


Fig.4: *The full-wave precision rectifier transfer characteristic.*

3. LIMITING CIRCUITS

A limiting circuit (also called a clipping circuit) limits the output signal to precise limits. To achieve this, either signal diodes or zener diodes are included in the feedback path around an op amp. There are a range of possible circuits, including those described in these notes.

3.1 UNIPOLAR LIMITER

The circuit of Fig.5(a) operates as a normal inverting amplifier within limits imposed by the zener diode D1. The zener limits the maximum positive output voltage to VZ when the zener is reverse biased, and the maximum negative output to -0.6V when the zener is forward biased. Fig.5(b) shows the output waveform in relation to the input waveform.

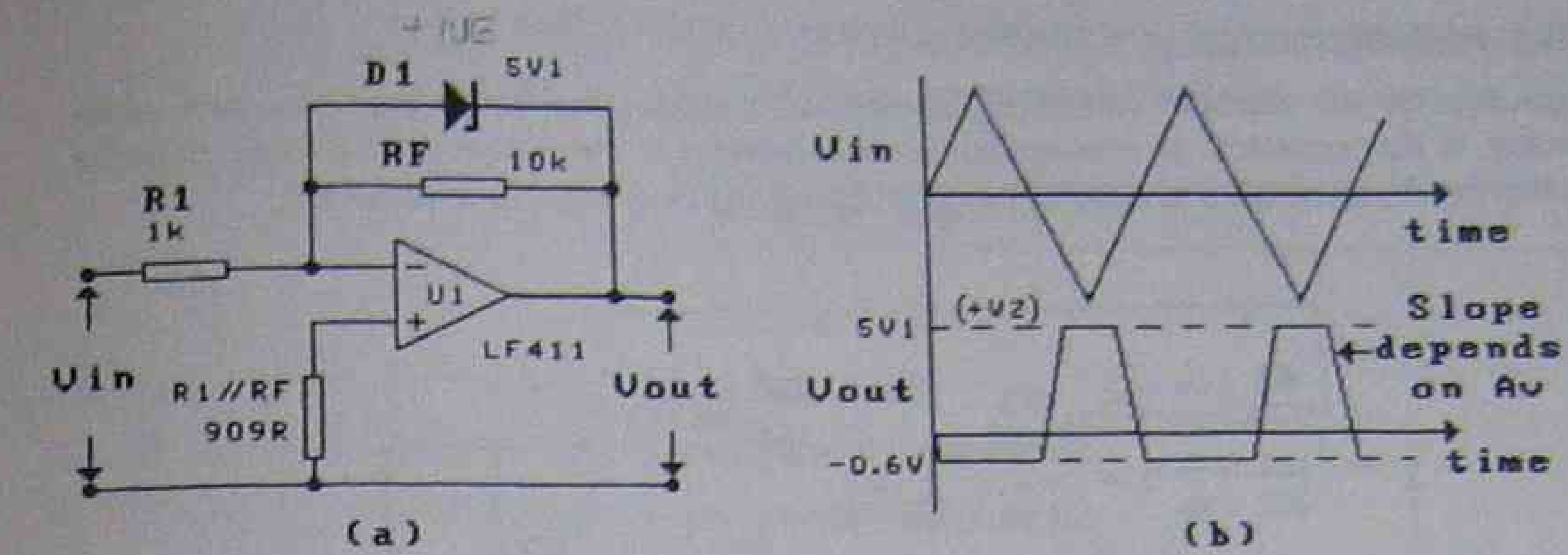


Fig.5: *The unipolar limiter.*

3.2 THE BIPOLAR ZENER LIMITER

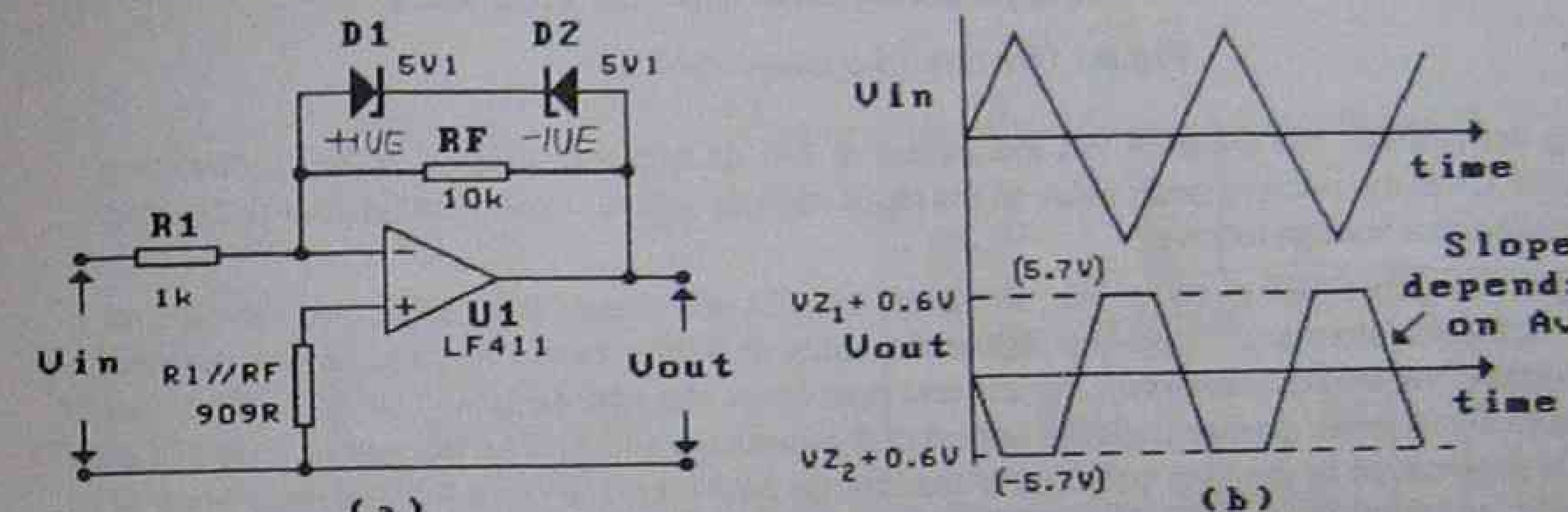


Fig.6: *The bipolar zener limiter.*

The circuit of Fig.6(a) operates as an inverting amplifier within limits set by zener diodes D1 and D2. The positive output is limited to [V_{Z1} + 0.6V] and the negative output to [V_{Z2} + 0.6V] as shown in Fig.6(b).

3.3 PRECISION CLIPPERS

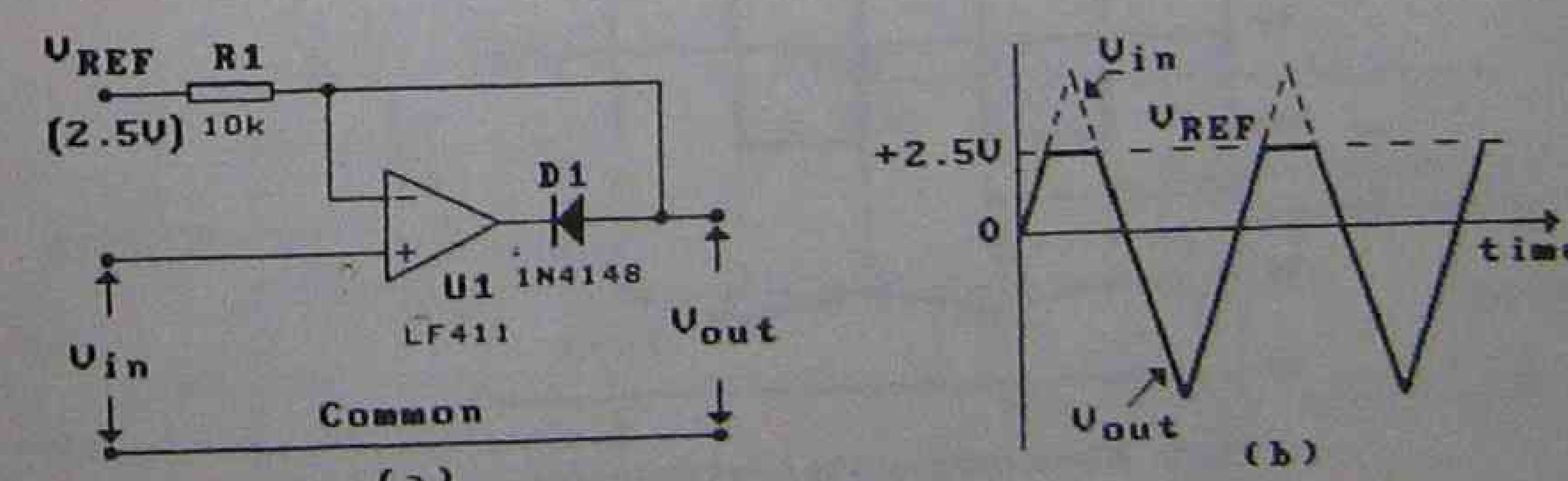


Fig.7: *Precision clipper.*

The circuit of Fig.7(a) is a positive clipper, where Vref determines the clipping level. When Vin is less than Vref, diode D1 will be forward biased by the output of the op amp and the circuit acts as a buffer in which Vout equals Vin. When Vin exceeds Vref, D1 will be reversed biased as the output of the op amp will exceed Vref. The output of the circuit will now equal Vref as shown in Fig.7(b). A negative clipper can be implemented by reversing D1 and the reference voltage.

3.4 THE PEAK DETECTOR

A conventional AC voltmeter is usually calibrated to display the RMS value of a sine wave. However, if the waveform is non-sinusoidal, the reading is meaningless. The peak detector circuit of Fig.8 can be used to allow a meter to display the peak value of a waveform.

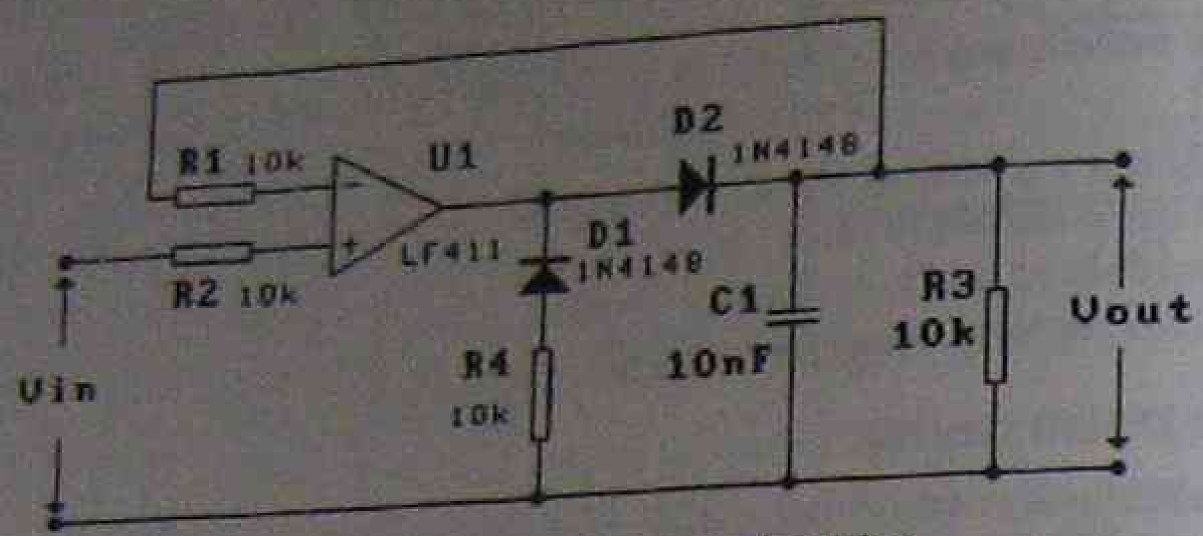


Fig.8: *The positive peak detector*

During the positive half cycle of Vin the output of the op amp will forward bias D2, charging capacitor C1 to the positive peak value of the input voltage. Under these conditions, the op amp is operating as a voltage follower.

During the negative half cycle of Vin, D2 is off and C1 will slowly discharge through R3, as shown in the waveforms of Fig.9. The higher the value of R3 the longer the discharge time and the smoother the output waveform. For correct operation, the time constant (= RC) of R3 and C1 should be 10 times greater than the period of the input waveform. The frequency limit of the circuit is determined by the time constant of the charge path for C1, (where R is the resistance of the diode), which should be one tenth of the period of the input waveform.

By connecting a DC voltmeter across the output of the circuit, the peak value of the input waveform will be displayed. To minimise loading, a buffer stage should be connected between the output of the peak detector circuit and the load.

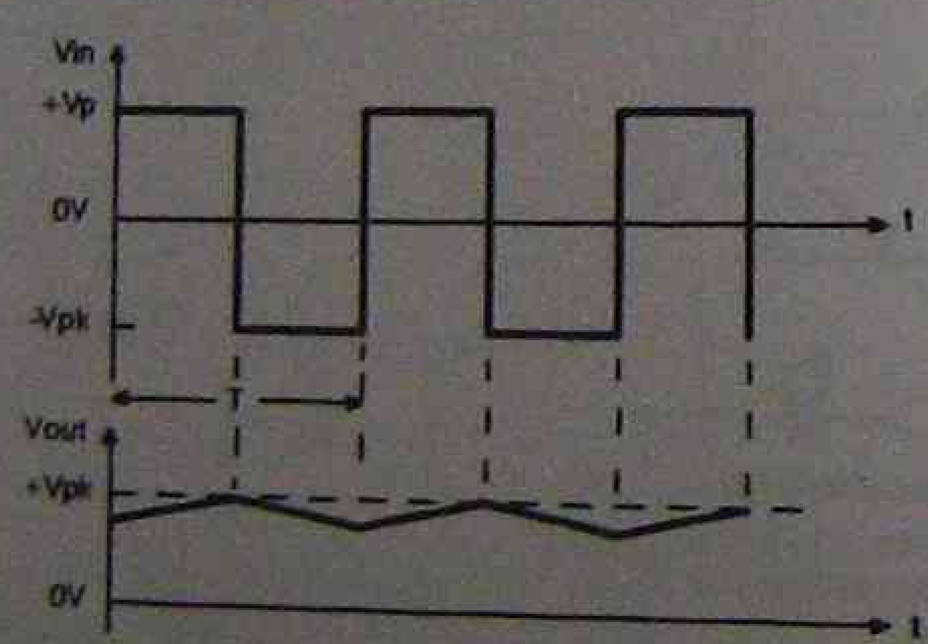


Fig.9: *Waveforms of the peak detector*

****

TUTORIAL WEEK 10

1. For the circuit of Fig.1, (assuming ideal components):
   (a) State the function being performed by the circuit around U1.
   (b) State the function being performed by the circuit around U2.
   (c) Using a common set of axes, sketch the waveforms of V_{IN}, V_{OUT1} and V_{OUT2}. Include the peak voltages and the period of each waveform.
   (d) Determine the value of the output voltage across R_L (V_{OUT3}).

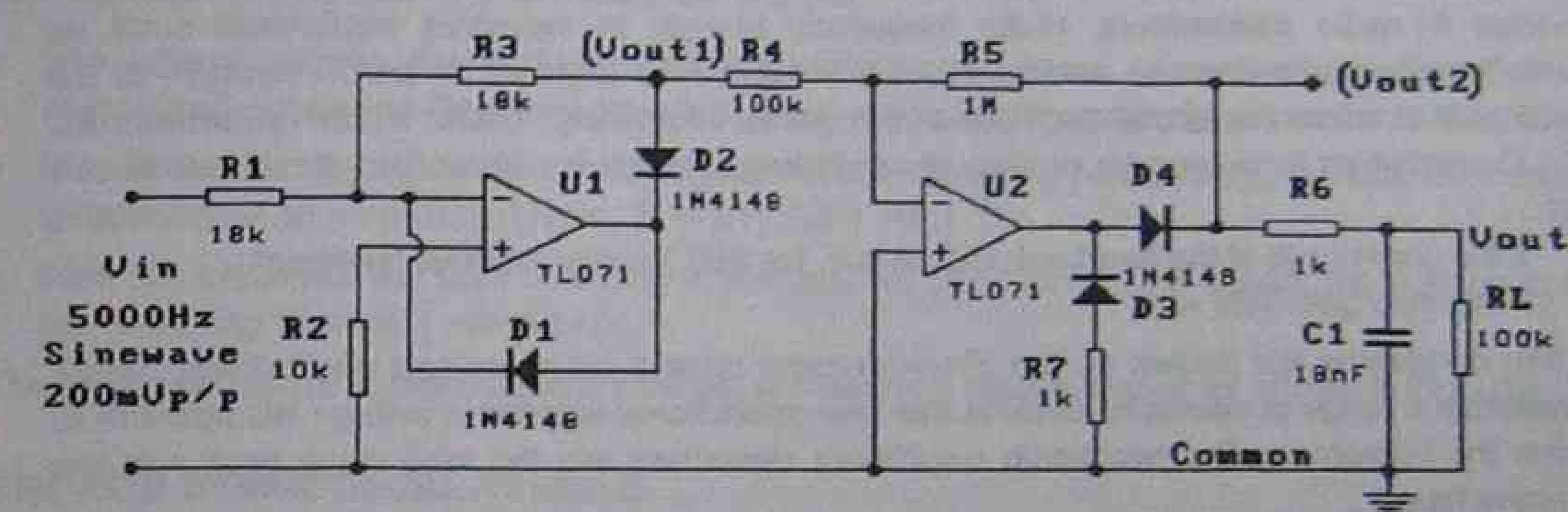


Fig.1

2. For the circuits of Fig.2, sketch the output waveform, assuming the input signal is a $10V_{p-p}$, 1kHz sinewave. Include relevant voltage values on both waveforms.

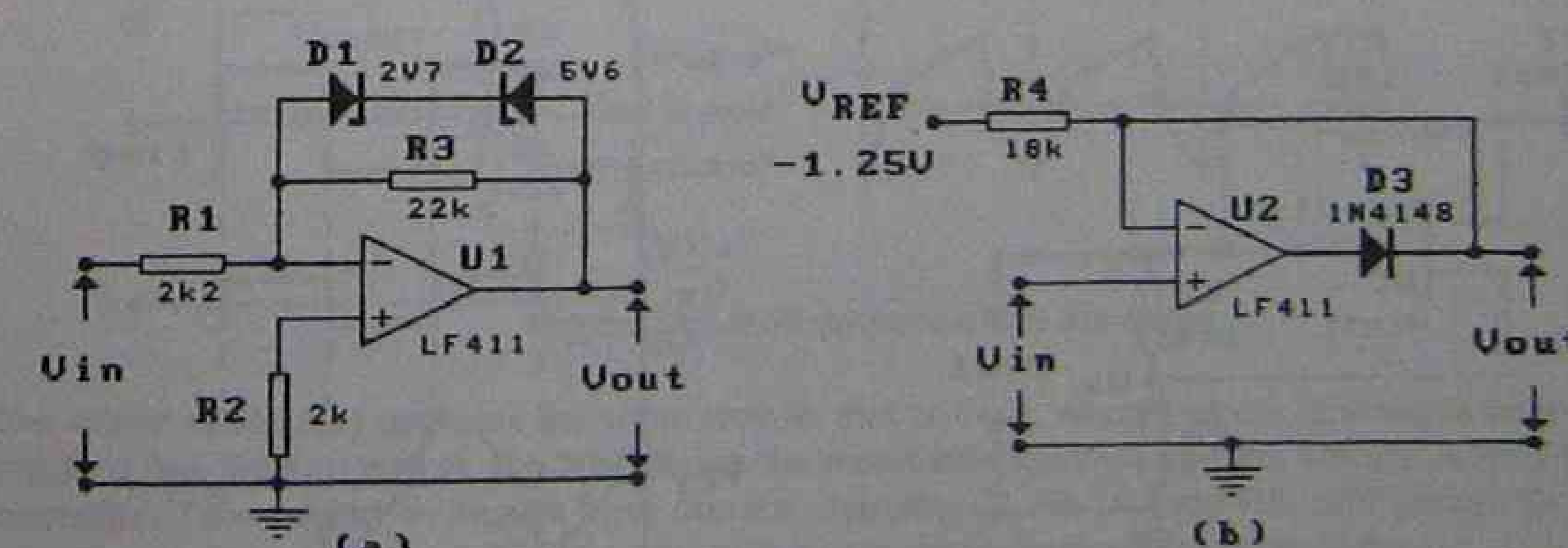


Fig.2

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

THEORY LESSON 11

SINE AND SQUARE WAVE OSCILLATORS

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

(a) *Calculate the operating frequency of an op amp astable oscillator, a monostable, a phase shift oscillator and a Wein bridge oscillator.*

(b) *Sketch the output waveform of each type of oscillator.*

(c) *State the minimum gain required for oscillation to occur in a phase shift oscillator and a Wein bridge oscillator.*

INTRODUCTION

In industry, oscillators are used to provide test signals, timing and synchronising signals, carrier frequencies in radio transmitters, radio frequency energy in industrial equipment such as dielectric heaters and so on. An oscillator usually has an RC network or an LC network in the feedback path to make the circuit oscillate at a required frequency. (Refer to the recommended text on LC oscillators for a broader outline of oscillators). A circuit will oscillate if two conditions are met:

- If the phase shift of the feedback signal is 0° (or 360°) giving positive feedback.
- If the loop gain (AB) = 1.

These two conditions are known as the *Barkhausen* criteria for sustained oscillations. These notes describe a range of oscillator circuits that use operational amplifiers with an RC network to determine the frequency. The two basic oscillators described are the sine wave type and the square wave type.

1. THE OP AMP ASTABLE

An astable circuit as in Fig.1(a) is one that continuously changes between two possible states, giving a square wave output as shown in Fig.1(b).

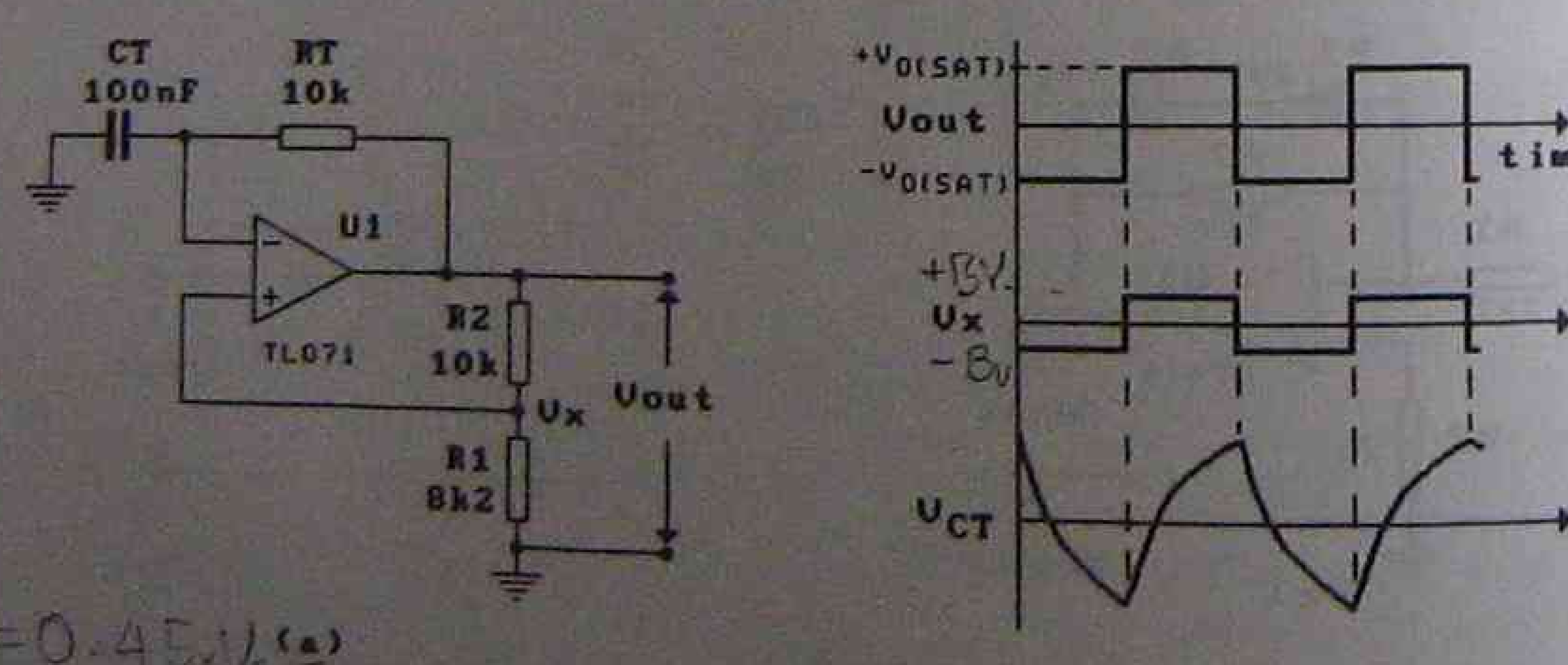


Fig.1: *The astable oscillator*

The circuit of Fig.1(a) is a comparator with positive feedback (via R1 and R2), applied to the non-inverting input of the op amp. The output voltage of the comparator swings between $\pm V_{O(SAT)}$, which is applied across the timing components R_T and C_T.

For this circuit, if R2 equals 1.16R1, the period of the output waveform (T) equals:

- $T = 2R_TC_T$ or frequency equals $\frac{1}{2R_TC_T}$

PROOF:

(1) Period (T) equals $2R_TC_T \ln\frac{(1+B)}{(1-B)}$ Where B equals $\frac{R1}{R1+R2}$

(2) If the natural log (ln) term equals 1, $T = 2R_TC_T$, which occurs if B = 0.462 of if $R_2 = 1.16R_1$

NOTE: B = the feedback factor, and for oscillation to occur, the gain of the amplifier (A) must have a loop gain (AB) of 1. (Barkhausen criteria).

1.2 CIRCUIT OPERATION:

For Fig.1(a), assume that V_{OUT} is initially at $-V_{O(SAT)}$.

- The voltage across the timing capacitor will charge 'down' towards the voltage at the non-inverting terminal. R_T allows the charge current to flow through the capacitor to V_{OUT}.
- When V_{CT} falls slightly below the voltage at the non-inverting [$-BV_{O(SAT)}$], the output of the op amp changes to $+V_{O(SAT)}$. [Note: B = R1/(R1 + R2)]
- Now the capacitor will start charging in the positive direction until it equals the voltage at the non-inverting terminal [$+BV_{O(SAT)}$].

The circuit continues in this oscillatory mode, producing a square output as shown in Fig.1(b).

2. THE NON-SYMMETRICAL ASTABLE

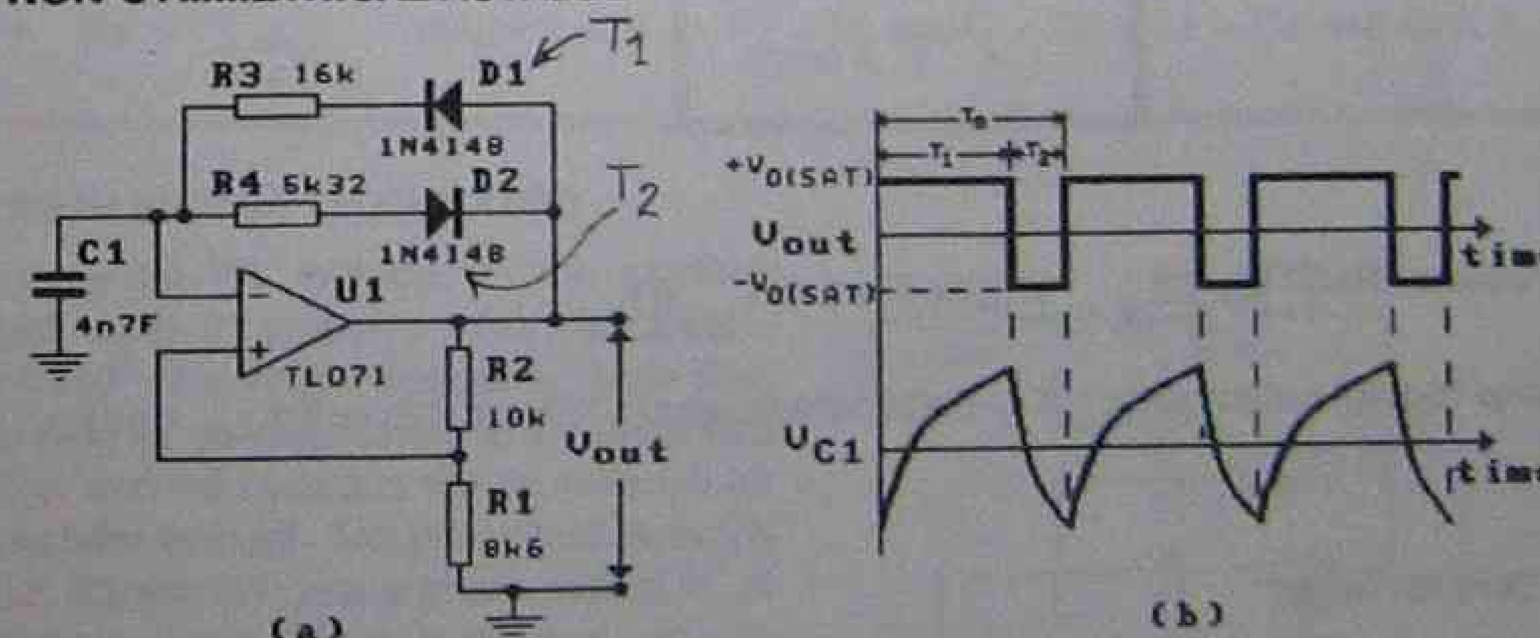


Fig.2: *The Non-symmetrical astable.*

The circuit of Fig.2(a) operates the same way as that of Fig.1, except diode steering is used to produce two time constants. Fig.2(b) shows the waveforms for V_{out} and the voltage across the capacitor. Time period T_1 results from current charging C1 via D1 and R3, and period T_2 by charge path D2 and R4. As in Fig.1(a), this circuit is useful for frequencies up to 1MHz.

The period of oscillation, $T_O = T_1 + T_2$, and if $R_2 = 1.16R_1$:

- $T_1 = R_3C_1$ and $T_2 = R_4C_1$

By selecting suitable values for R3 and R4, the mark-space ratio (high time-low time) of the waveform can be adjusted to suit.

3. THE OP AMP MONOSTABLE

A monostable is a circuit with *one stable state*, and requires a trigger pulse to cause it to enter the unstable state, in which it remains for a predetermined period before reverting back to its normal or stable state. Fig.3(a) is a monostable, in which a negative pulse is used at V_{in} to trigger the circuit to its unstable state. The stable state for this circuit is when V_{out} is at $+V_{O(SAT)}$.

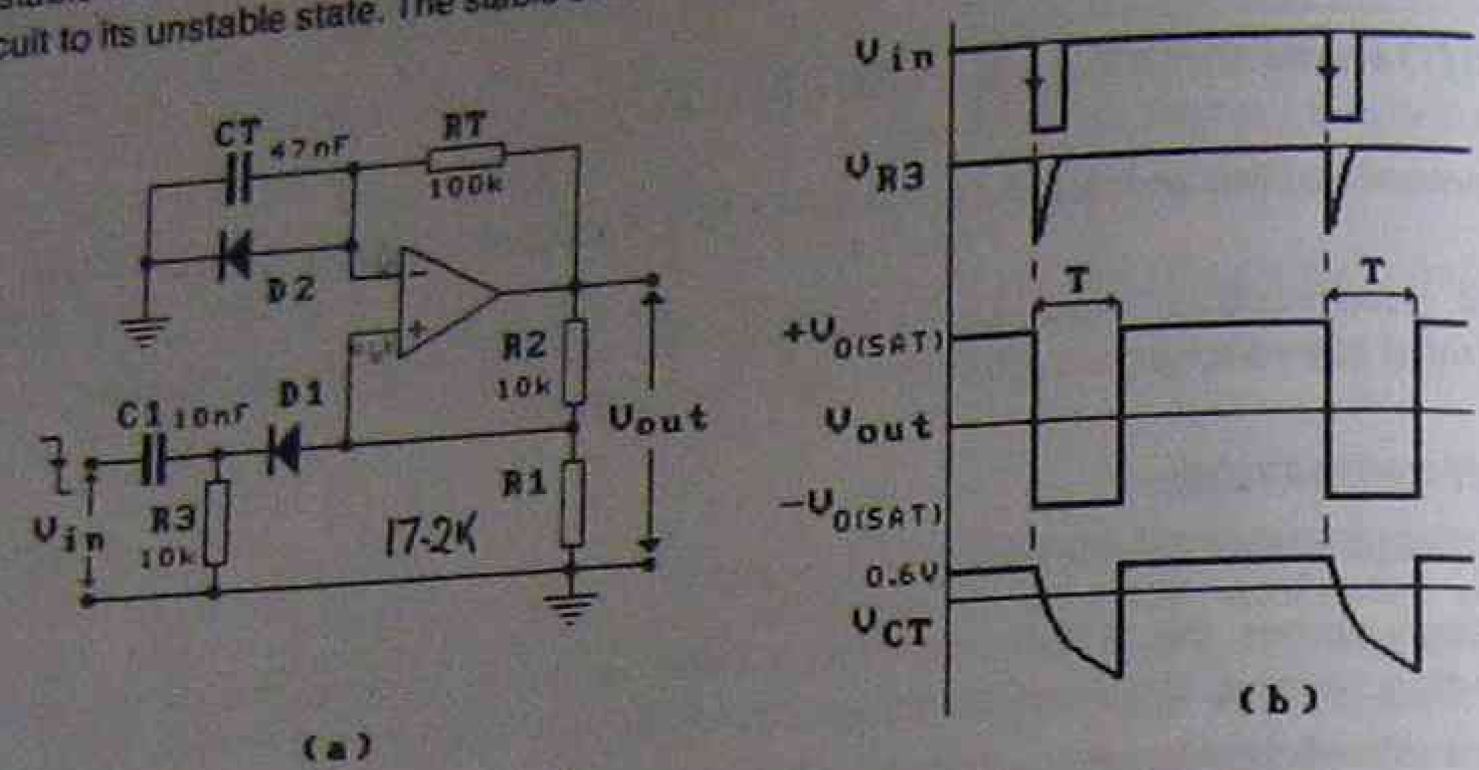


Fig.3: *The op amp monostable.*

Positive feedback is applied to the non-inverting input of the op amp via the feedback network of R1 and R2.

For this circuit, if **R1 equals 1.72R2**, time (T) equals:

- $T = R_TC_T$

PROOF:

(1) Time (T) equals $2R_TC_T \ln \frac{1}{(1-B)}$ Where B equals $\frac{R1}{R1 + R2}$

(2) If the natural log (ln) term equals 1, $T = R_TC_T$, which occurs if B = 0.632 or if R1 = 1.72R2

3.1 CIRCUIT OPERATION:

- The stable state for Fig.3(a) is the condition of $V_{out} = +V_{O(SAT)}$. Thus, the voltage at the (-) input is 0.6V, (caused by D2) and that at the (+) input some higher value. [$B \times V_{O(SAT)}$].
- If a negative voltage is applied to the (+) input, Vout will switch to $-V_{O(SAT)}$. Timing capacitor C_T will now charge via R_T in the negative direction and the voltage at the (+) input will equal be a negative voltage [$-B \times V_{O(SAT)}$].
- When C_T has charged to a value greater than the voltage at the (+) input, the output of the op amp will switch back to $+V_{O(SAT)}$. The time taken for this to happen depends on the value of R_T and C_T as described above and the circuit has now returned to its stable state.

The waveforms of Fig.3(b) show the input trigger, the output waveform and the voltage across C_T. Note that when a negative going pulse is applied to the input, C1 and R3 convert it into a negative going spike as shown. During the unstable state, further input trigger pulses will have no effect.

4. THE PHASE SHIFT OSCILLATOR

The next two circuits are *sine wave* oscillators, and rely on the feedback being positive at *one* frequency only. To achieve this, an RC network is used in the positive feedback network and negative feedback is applied in the usual manner for an inverting amplifier. Fig.4 shows the phase shift oscillator, which consists of three RC cascaded networks in the positive feedback path. Op amp U1 buffers the feedback network from the inverting amplifier U2.

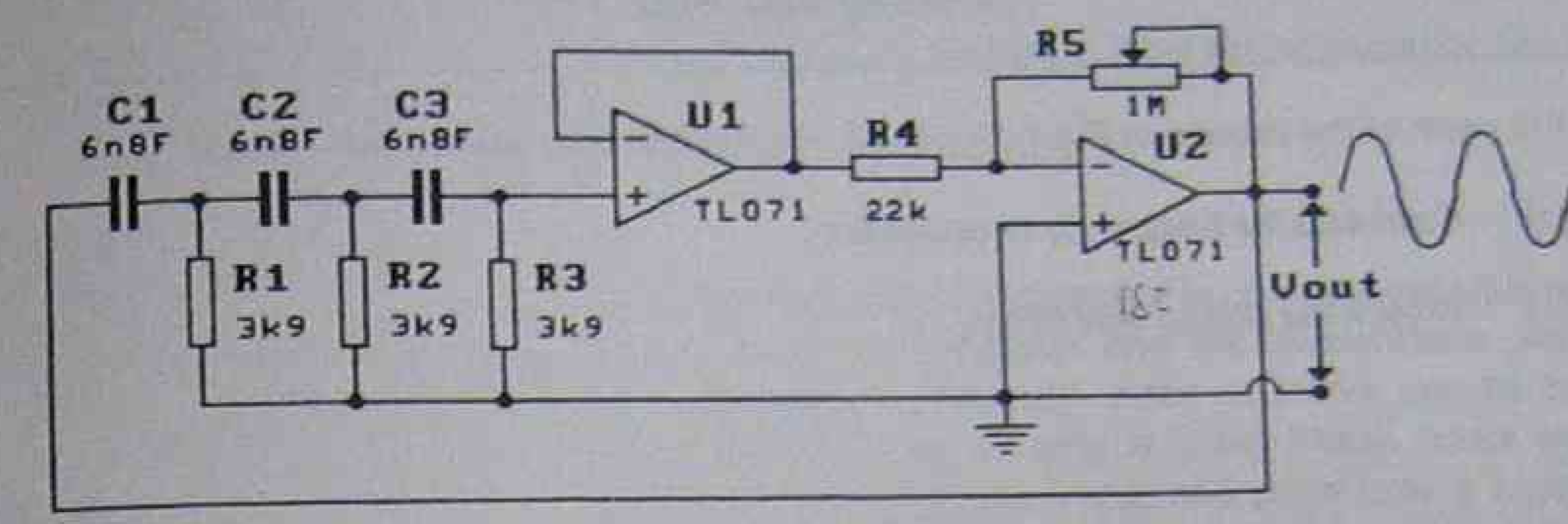


Fig.4: *The phase shift oscillator.*

The inverting amplifier U2 provides 180° of phase shift, and the feedback network provides a second 180° phase shift, giving a total of 360° phase shift. The RC network has 180° phase shift at only one frequency, and if the amplifier gain is large enough to make the loop gain, AB = 1, oscillation will occur. The attenuation of the RC network at 180° phase shift is 1/29, thus the gain of U2 must be 29. The equation for the frequency of oscillation, f_o is;

- $f_o = \frac{1}{15.4\,RC}$ Where $R = R_1 = R_2 = R_3$ and $C = C_1 = C_2 = C_3$ and $A_{V\,(U2)} = 29$ $\equiv \frac{0.065}{RC}$

5. THE WEIN BRIDGE OSCILLATOR

Fig.5 uses an RC network in the positive feedback path to create a total loop phase shift of 360° at a single frequency. Note that the op amp is connected as a non-inverting amplifier, and the output of the op amp will be in phase with its input. The phase shift network (R4, C2, R3 and C1) has a phase shift of 0° at only one frequency, which is the frequency of oscillation. R1 and R2 provide negative feedback for U2 which needs a gain of 3 to overcome the attenuation of the phase shift network.

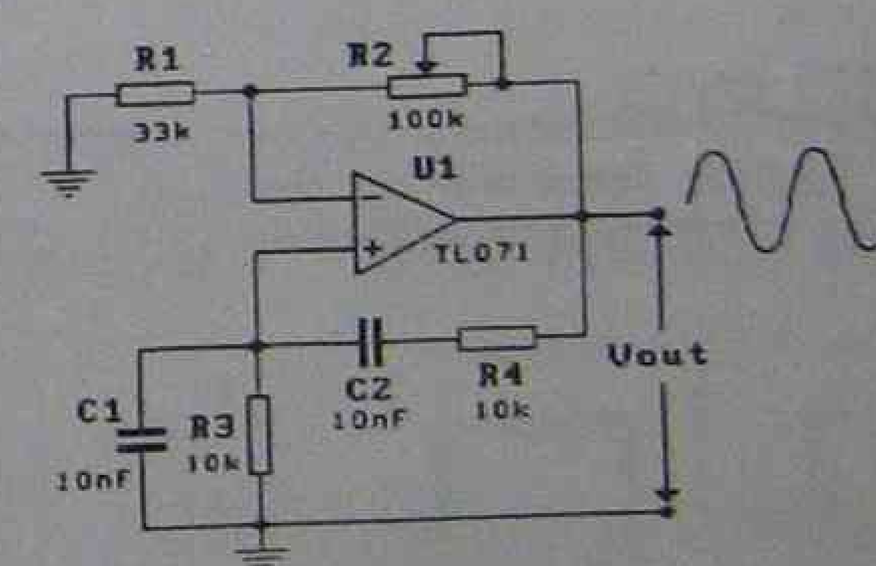


Fig.5: *The Wein bridge oscillator*

- $f_o = \frac{1}{6.28RC}$ Where $R = R_3 = R_4$ and $C = C_1 = C_2$ and $A_{V\,(U2)} = 3$

This basic circuit suffers from distortion at the peaks of the output sine wave which can be overcome by placing a thermistor in series with R2.

****

TUTORIAL WEEK 11

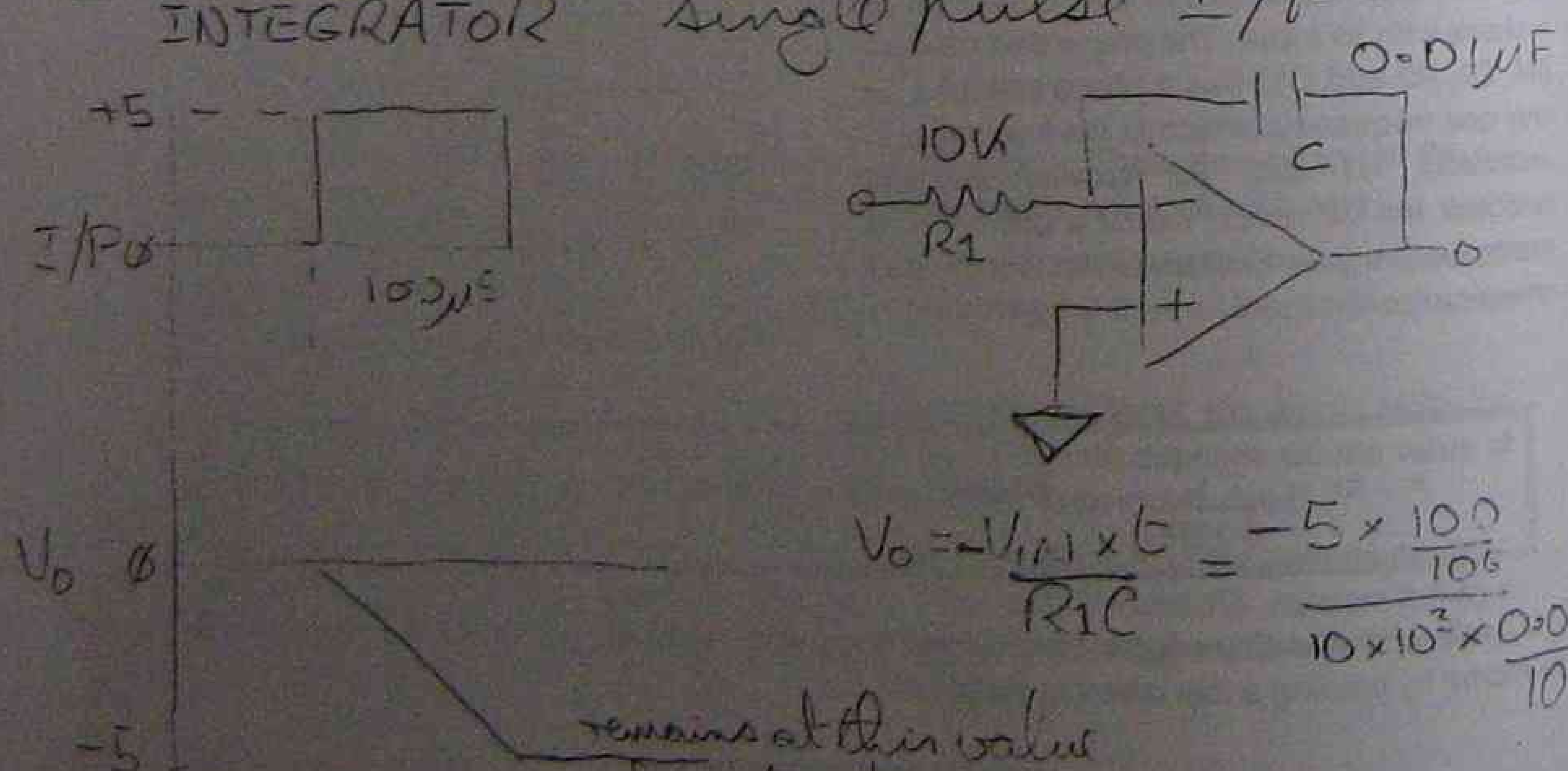
1. For each of the circuits of Fig.1, Fig.4 and Fig.5 in the associated theory notes:
   (a) Calculate the frequency of oscillation.
   (b) Sketch the output waveform. -

2. For the circuit of Fig.2 in the theory notes:
   (a) Calculate the periods labelled as T_1, T_2 and T_0.
   (b) Calculate the frequency of oscillation.
   (c) Sketch the output waveform, labelling the maximum output voltage and time axis.

3. For the circuit of Fig.3 in the theory notes calculate the period of the unstable state (labelled as 'T').

****

Ref FLOYD
P 762
INTEGRATOR single pulse I/P



$V_o = \frac{-V_{I/P} \times t}{R_1C} = \frac{-5 \times \frac{100}{10^6}}{10 \times 10^3 \times \frac{0.01}{10^6}}$

Inverting amp goes negative going for +V I/P

OP AMP RC CIRCUITS

OBJECTIVES: At the end of this lesson you should be able to:
(a) *Sketch the output waveform of an op amp integrator and differentiator for a square wave input.*
(b) *Sketch the output waveforms of a function generator.*

INTRODUCTION

Changing the shape of a waveform using a resistor and a capacitor is common practice and the two basic circuit configurations are referred to as the integrator and the differentiator. An integrator smoothes a wave and a differentiator sharpens it. Although these networks can be constructed without an amplifier, best performance results if an op amp is used. These notes examine the integrator and the differentiator circuits based around op amps, and show how a square wave is affected by these circuits. An application is also described in which an integrator is combined with a square wave oscillator giving a function generator.

1 THE INTEGRATOR

The basic RC integrator network is shown in Fig.1 which shows that the output waveform is developed across the capacitor. Because the resistor limits the rate at which the capacitor can be charged and discharged, the output waveform for a square wave input is a series of charge/discharge curves as shown. If a triangular waveform is required, the capacitor must be charged and discharged at a constant rate, giving a linear change in voltage, which can be achieved using an op amp.

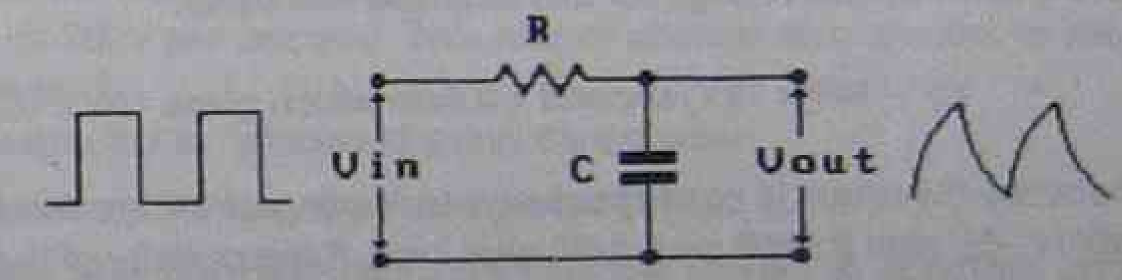


Fig1. *The basic integrator*

1.1 THE OP AMP INTEGRATOR

The circuit of Fig.2 shows the basic op amp integrator, in which the capacitor is connected in the feedback path of the op amp, and the resistor is in series with the input signal. Although a full analysis of this circuit is complex, a simplified analysis will show how this circuit operates.

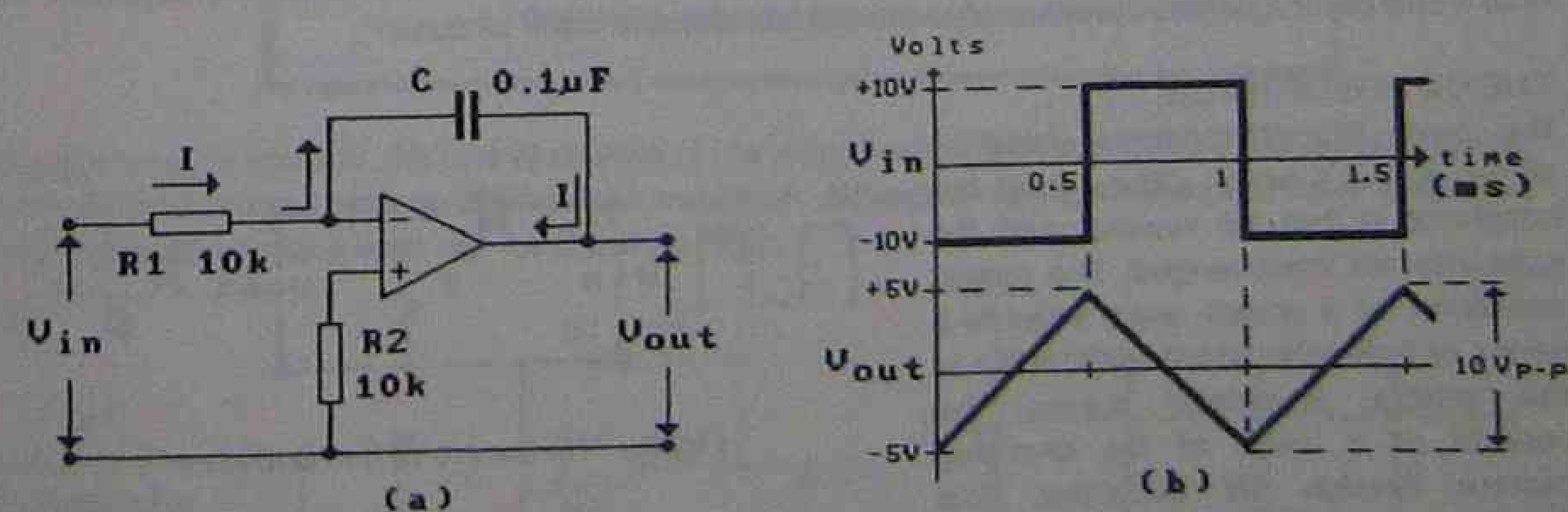


Fig.2: *The op amp integrator*

- Assume the input voltage is +10V. Because the (-) input of the op amp is a virtual ground, the current in R1 will equal Vin/R1, in this case 1mA.

- This current will flow through the capacitor as there is no other path for the current. As the current is constant (at 1mA in this example), the voltage across the capacitor will increase at a constant value.
- The rate at which this voltage changes depends on the value of the capacitor and the value of the charging current, and the general equation to find the voltage across the capacitor (and hence at the output of the op amp) is determined by the equation:

$V_o = V_c = \frac{I}{C} \times t$ where I = current (amps), C = capacitance (F) and t = time (seconds).

As $I = \frac{V_{in}}{R_1}$ this equation becomes

$V_o = V_c = \frac{(V_{in}) \times t}{R_1 C}$ .... (1)

or $V_o = V_c = V_{in} \times \left(\frac{1}{R_1 C}\right)$ .... (2)

> **Note**
> This analysis assumes ideal components
> These results may not be obtained in practice

Thus if R1 = 10k and C = 0.1uF, the term (R_1C) equals 10k x 0.1uF which gives 0.001. If the rate of change is to be expressed in volts per second, the value of t needs to be 1. Thus, from equation (2), the rate of change is 1/0.001 or 1000V per second. If Vin is 10V, the output will equal 10V x 1000V/s, or 10kV after one second. Obviously the limitation will be the maximum output voltage of the op amp [= $V_{O(sat)}$], and realistic time values will need to be much less than 1 second. If the time interval is 0.5mS, the output voltage for this example will equal:

- Vo = 10V x 1kV/second x 0.5ms which gives 10V x 0.5 = **5V**.

Because the circuit is connected as an inverting amplifier, the output will be negative, and the voltage will be -5V after 0.5mS and -10V after 1ms. If the polarity of the input is reversed, the direction of the output voltage will change, and the output will *ramp* in the opposite direction. Thus, if the input is a 1kHz, 20V peak to peak square wave, the output waveform will be a triangular wave with a peak to peak value of 10V, or a waveform centred around 0V with maximum values of +5V and -5V.

> Thus, $V_o = (-) V_{in} \times \frac{1}{R_1 C} \times$ time in seconds

The '*rate of change of* V_{OUT}' therefore depends on V_{IN} and the values of R1 and the integrating capacitor C. Thus if a square wave is applied to the input a triangular wave will be produced. If the input voltage is zero, then the '*rate of change of* V_{OUT}' is zero. In this mode the capacitor can hold a charge for a period of time, and is used in 'sample and hold' circuits.

2 THE DIFFERENTIATOR

The basic RC differentiator circuit is shown in Fig.3 which differs from the integrator in that the resistor and the capacitor are interchanged. The output waveform for a square wave input is a series of spikes, in which each spike is the voltage developed across the resistor as a result of the charging current through the capacitor. The differentiator produces an output that is proportional to the *rate of change* of the input signal. An ideal differentiator would therefore produce a series of very narrow pulses, rather than the waveform shown in Fig.3. By adding an op amp to the differentiator circuit of Fig.3, a closer approximation to the ideal can be obtained.

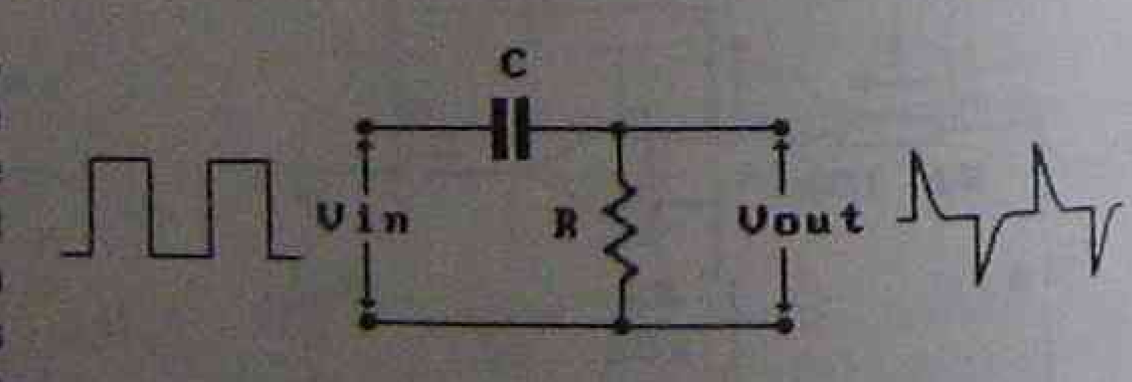


Fig.3: *The basic differentiator*

2.1 THE OP AMP DIFFERENTIATOR

The circuit of Fig.4(a) shows the basic op amp differentiator. Because a differentiator is the opposite to an integrator, analysis of the circuit will show that if a triangular wave is applied to the input of the differentiator, a square wave will result at the output. For example if the output waveform of Fig.2 is applied to the input of Fig.4, a 20Vp-p square wave should result at the output.

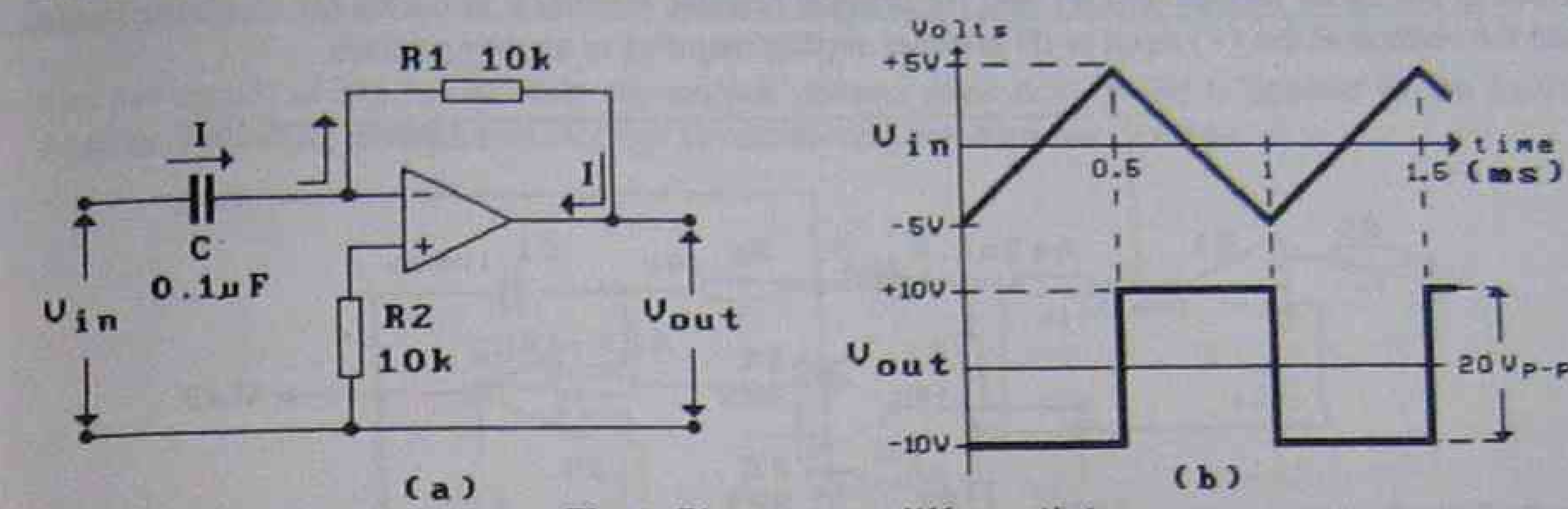


Fig.4: *The op amp differentiator*

PROOF:

- The input waveform is a 1kHz 10Vp-p triangular wave. The period of the wave is therefore 1ms, giving a rate of change for the positive going section of 10V per 0.5ms. This gives a rate of change of 20V per ms, or 20kV per second. This rate of change also applies to the negative going section of the wave.
- The current (I) flowing into the capacitor is calculated using the equation

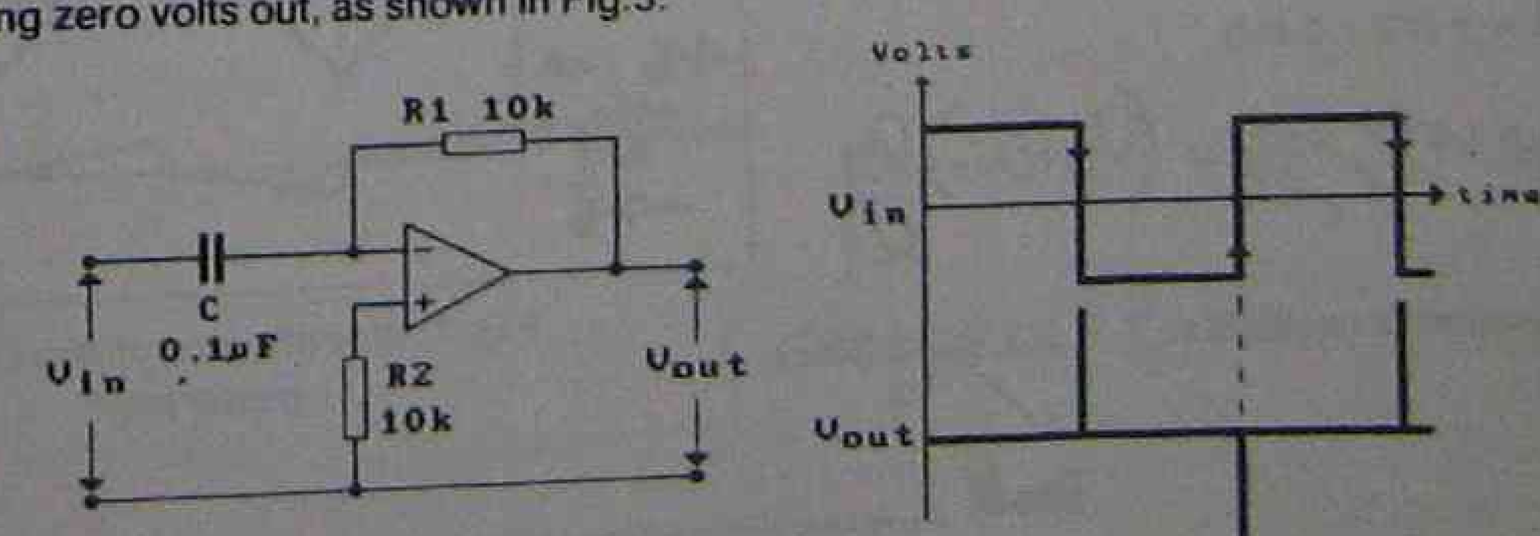
  Ic = C times the rate of change of the applied voltage (Vin) .... (1)
- Because the voltage is changing at a constant rate (20kV/s) the current is constant and equals 0.1uF x 20kV/s which gives 2mA.
- The current flowing in the capacitor also flows in R1, as there is no other path for the current, giving a voltage across the resistor of 2mA x 10k which equals 20V. As current flows from positive to negative, and as the (-) input of the op amp is a virtual earth (0V), the output must be -20V.
- Reversing the direction of Vin to the negative going section of the triangular wave will similarly give +20V at the output of the op amp.
- Thus, as shown in Fig.4(b), the output waveform is a 20Vp-p square wave as predicted.

As the current Ic also equals Vout/R1, equation 1 can be rearranged to give the general equation of:

> $V_{out} = (-) R_1 C \times$ rate of change of V_{in} (in volts per second)

If a square wave is applied, the rate of change of the edges will (theoretically) be infinitely fast, giving an infinitely high output voltage during each transition. Between each transition, the input voltage is not changing, giving zero volts out, as shown in Fig.5.

Fig.5 *Output of the differentiator for a square wave input.*



3. THE FUNCTION GENERATOR:

A function generator is a circuit used to produce a range of different waveshapes. Fig.6 shows a basic function generator with three outputs; a high level square wave, a square wave of restricted amplitude and a triangular wave. The circuit contains a comparator, (U1), and an integrator, (U2). Resistors R3 and R5 minimise the effect of input bias currents on the output waveforms, R4 limits the current to the zener diodes and R1 and R2 provide positive feedback to cause U1 to change state when the voltage at the (+) input of U1 is either slightly negative or slightly positive.

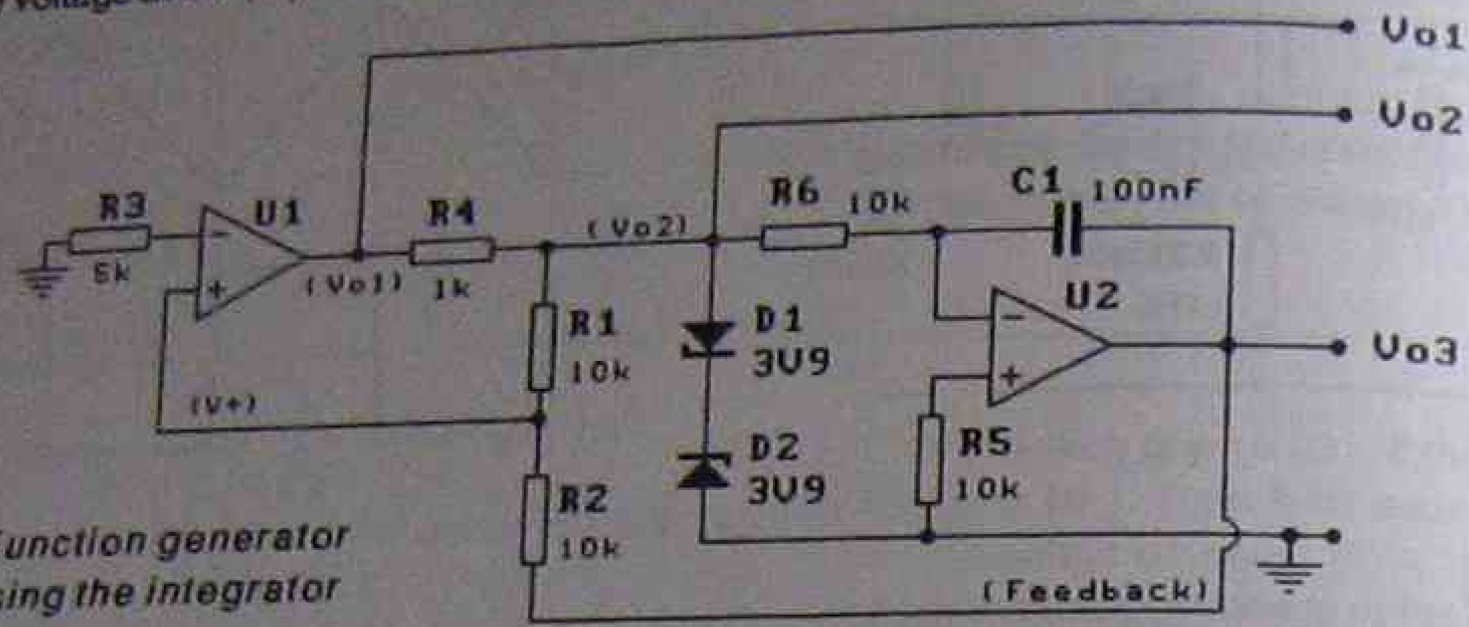


Fig.6: *Function generator using the integrator*

3.1 CIRCUIT OPERATION

- When the output of U1 (Vo1) is at +Vo(sat), the voltage applied to the integrator (Vo2) will be limited to V_{Z1} + 0.6V (= 4.5V). The integrator will ramp in the negative direction at a rate determined by R6 and C1.
- The output of the integrator is applied to R2, and the junction of R1 and R2 is connected to the (+) input of U1. As the top of R1 has a voltage of +4.5V, when the output of the integrator is sufficiently negative, the (+) input of op amp U1 will also become slightly negative. This will cause the comparator to change states, and Vo1 will now change to -Vo(sat).
- The cycle will continue, but now in the reverse direction.

The waveforms for the circuit are shown in Fig.7, including the voltage present at the (+) input of op amp U1. Note that this waveform will only vary by a very small amount around the zero line, less than 1mV, depending on the open loop gain of U1.

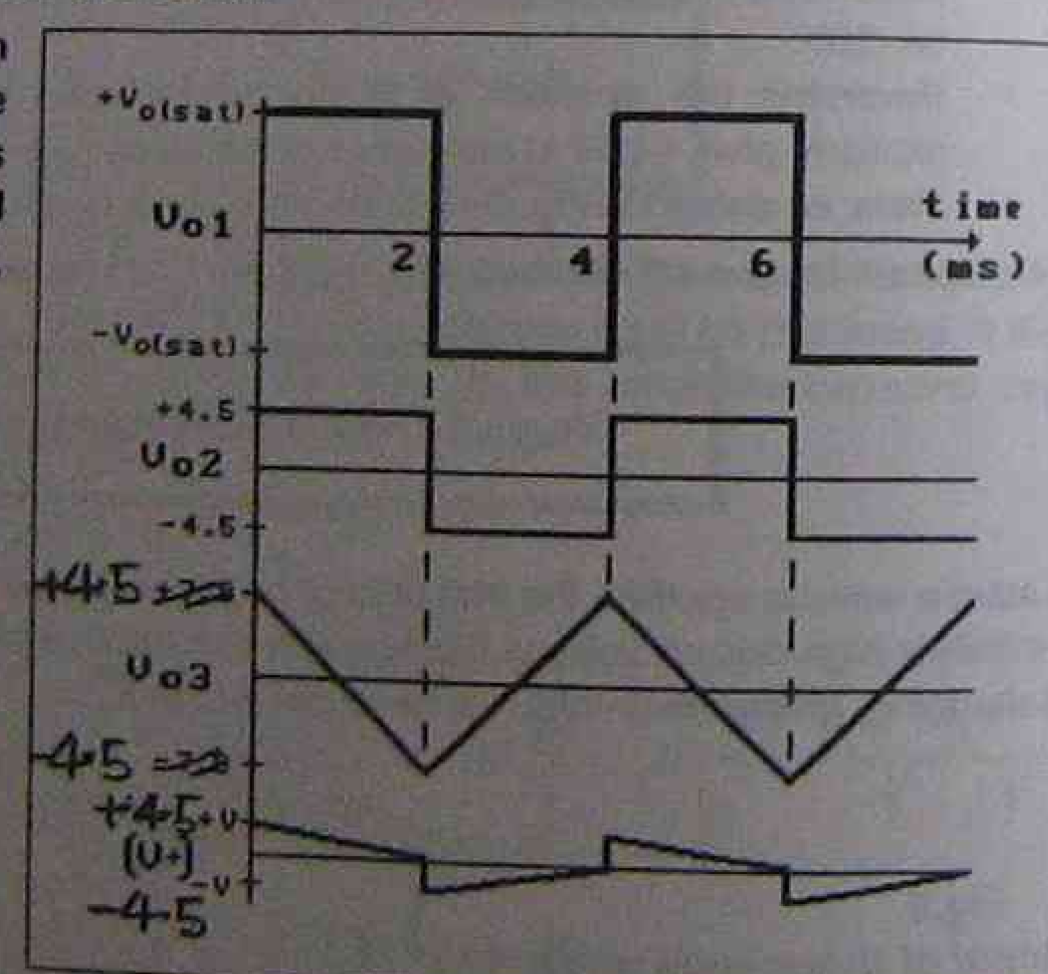


Fig.7: *Waveforms of the function generator.*

3.2 CIRCUIT CALCULATIONS:

$V_{O1} = \pm V_{O(SAT)}$

$V_{O2} = \pm (V_Z + 0.6V)$

$V_{O3} = 2 \times V_Z \times R_2 / R_1$ $= (V_Z + 0.6)\frac{R_2}{R_1}$

The period of oscillation, $T = \frac{2 \times V_{O3} \times R_6 C_1}{V_Z}$

Peak

****

TUTORIAL WEEK 13

1. For the circuit of Fig.1, calculate the output voltage after 2ms, if -5V is applied to the input. Assume the output voltage was 0V prior to connecting the -5V input. voltage.

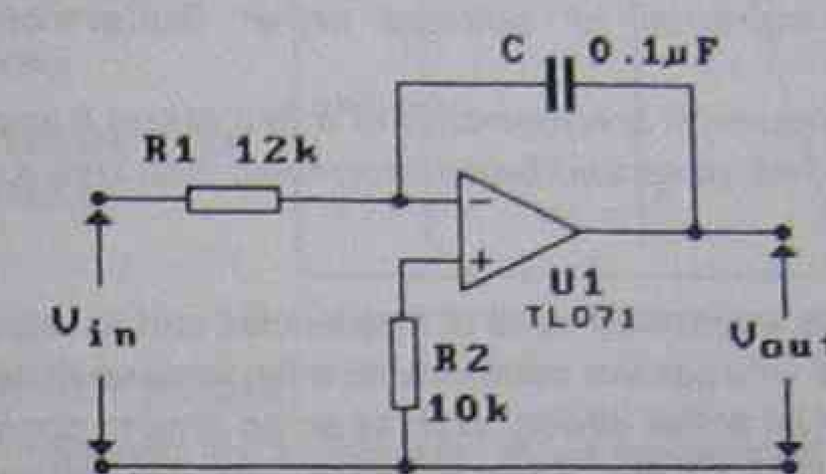


Fig.1

2. For the circuit of Fig.2, sketch the output waveform if a 200Hz triangular wave with a peak to peak voltage of 10V is connected to the input. Label the waveform to show relevant voltage and time values.

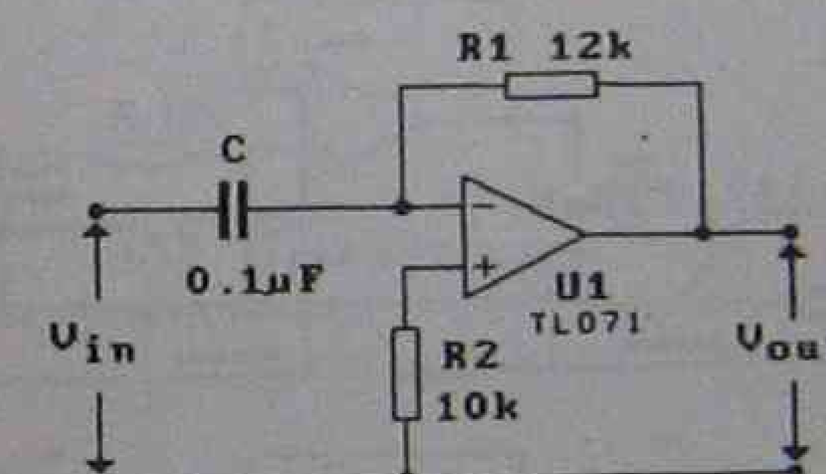


Fig.2

3. For the circuit of Fig.6 in the theory notes, determine the value of Vo2 and Vo3 if Vo1 equals +15V and capacitor C_1 is a short circuit.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

THEORY LESSON 14

ACTIVE FILTERS

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

(a) *State the difference between a first order and second order filter.*
(b) *Sketch the ideal pass-band of the four types of filters.*
(c) *Calculate the cut-off frequency of second order Butterworth high-pass and low-pass filters.*
(d) *Calculate the low and high cut-off frequencies of a two stage band-pass filter.*
(e) *Identify the circuits of a band-pass and band-stop filter that use a single op amp.*

INTRODUCTION

A filter is a circuit designed to pass a specified band of frequencies and to reject others outside the band. Filter networks can be made with passive components only, such as resistors, capacitors and inductors, or they can incorporate an active device, such as an op amp in conjunction with passive components. When an op amp is used, the characteristics of the filter are improved and the circuit is referred to as an active filter.

There are four types of filters; the low pass, high pass, band pass and band stop (or notch filter). The effectiveness of a filter is determined by how well it rejects frequencies outside its pass band, and an ideal filter will completely reject all frequencies once its cut-off point is reached. The diagrams of Fig.1 show the typical and ideal response curves of each type of filter.

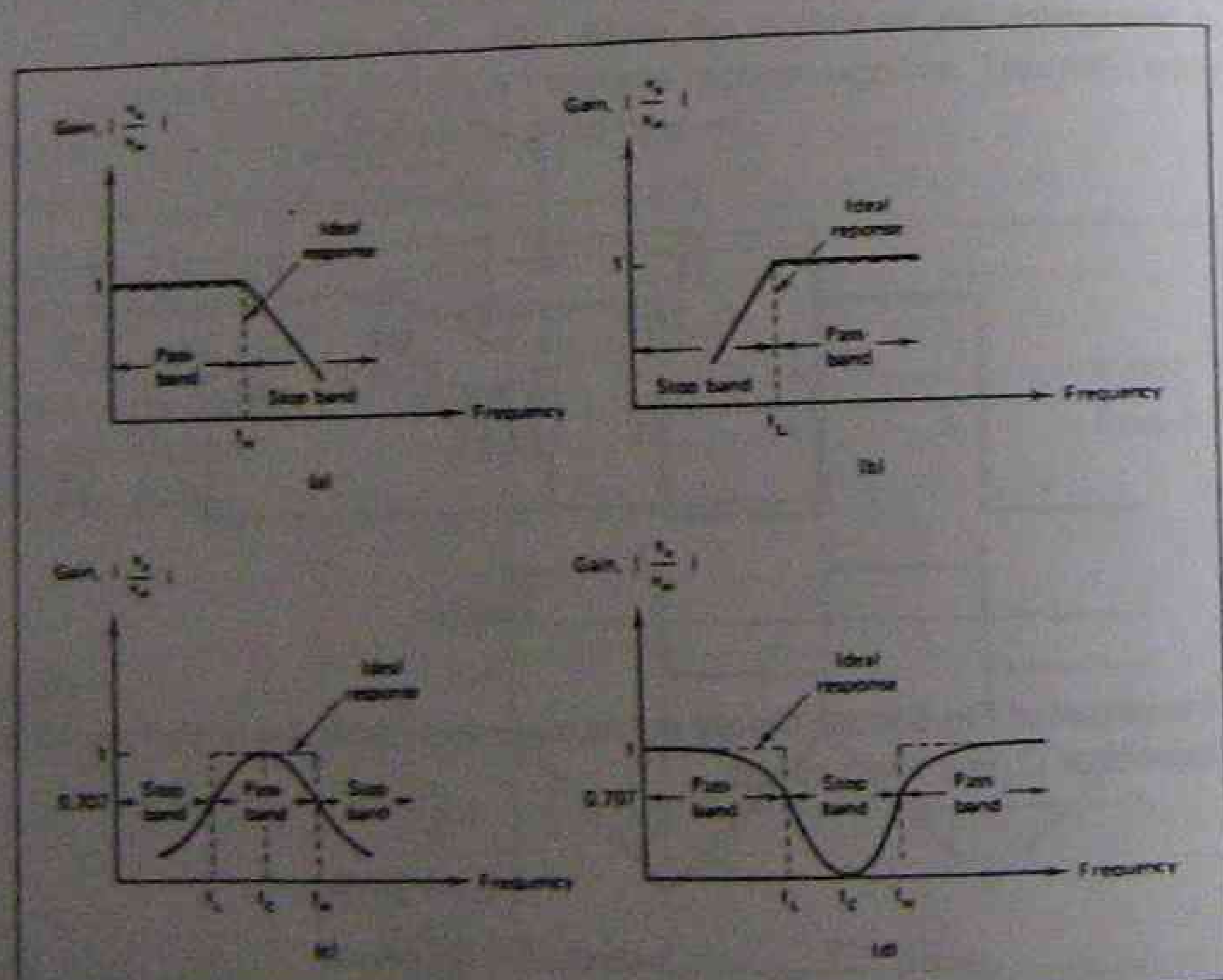


Fig.1: *Frequency response of filters.*

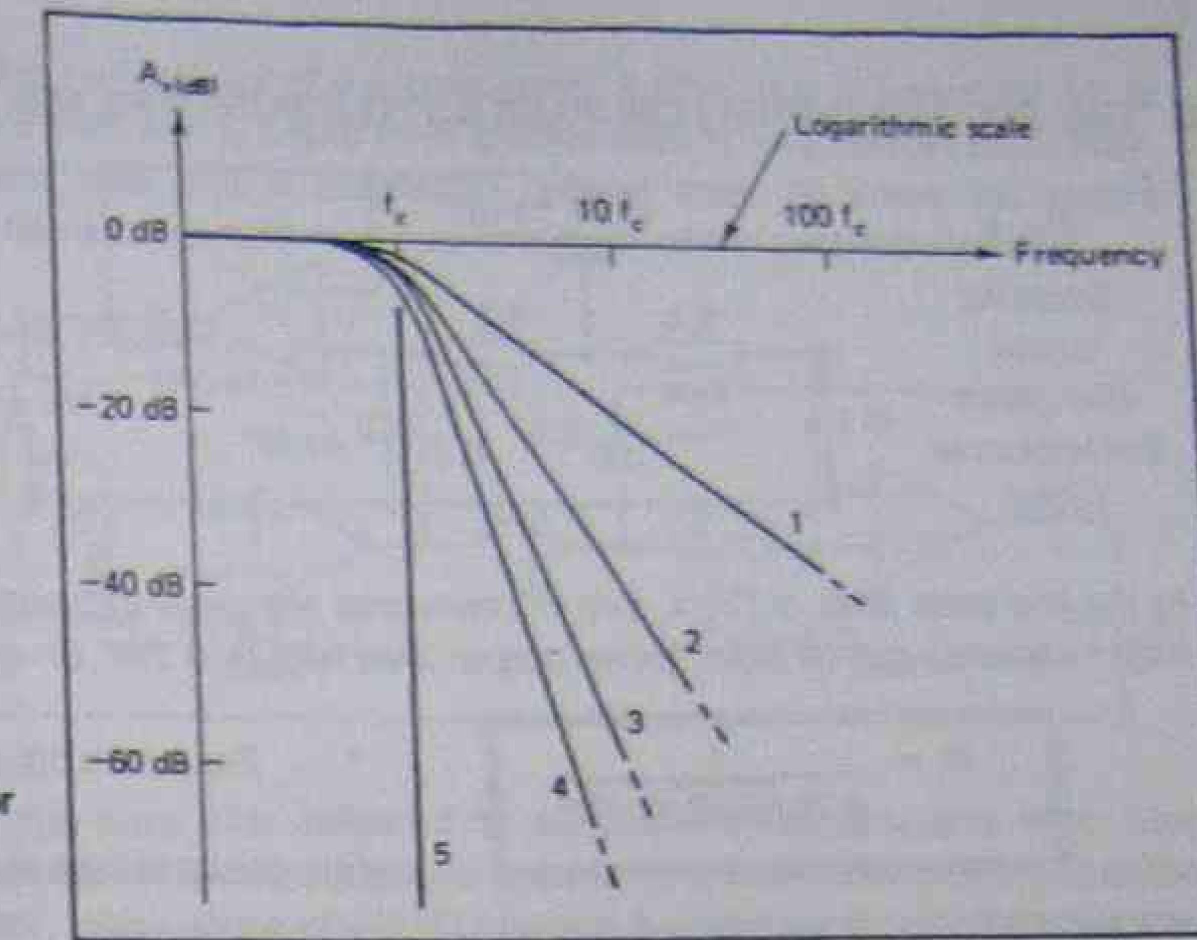
(a) Low pass
(b) High pass
(c) Band pass
(d) Band stop or notch filter

Filters are classified according to the shape of their response, as shown in Fig.2 which illustrates the response curves a low pass filter for five *orders*. As a single RC network provides one order, or -20dB/decade of roll off, each increase in order requires another RC network. Most filters are 2nd order, and therefore have two RC networks in conjunction with an op amp.

Fig.2: *Filter classification*

1 1st order (-20dB/decade).
2 2nd order (-40dB/decade).
3 3rd order (-60dB/decade).
4 4th order (-80dB/decade).
5 ideal (total rejection).

(These response curves are all for a low pass filter.)



1. SINGLE ORDER ACTIVE FILTERS

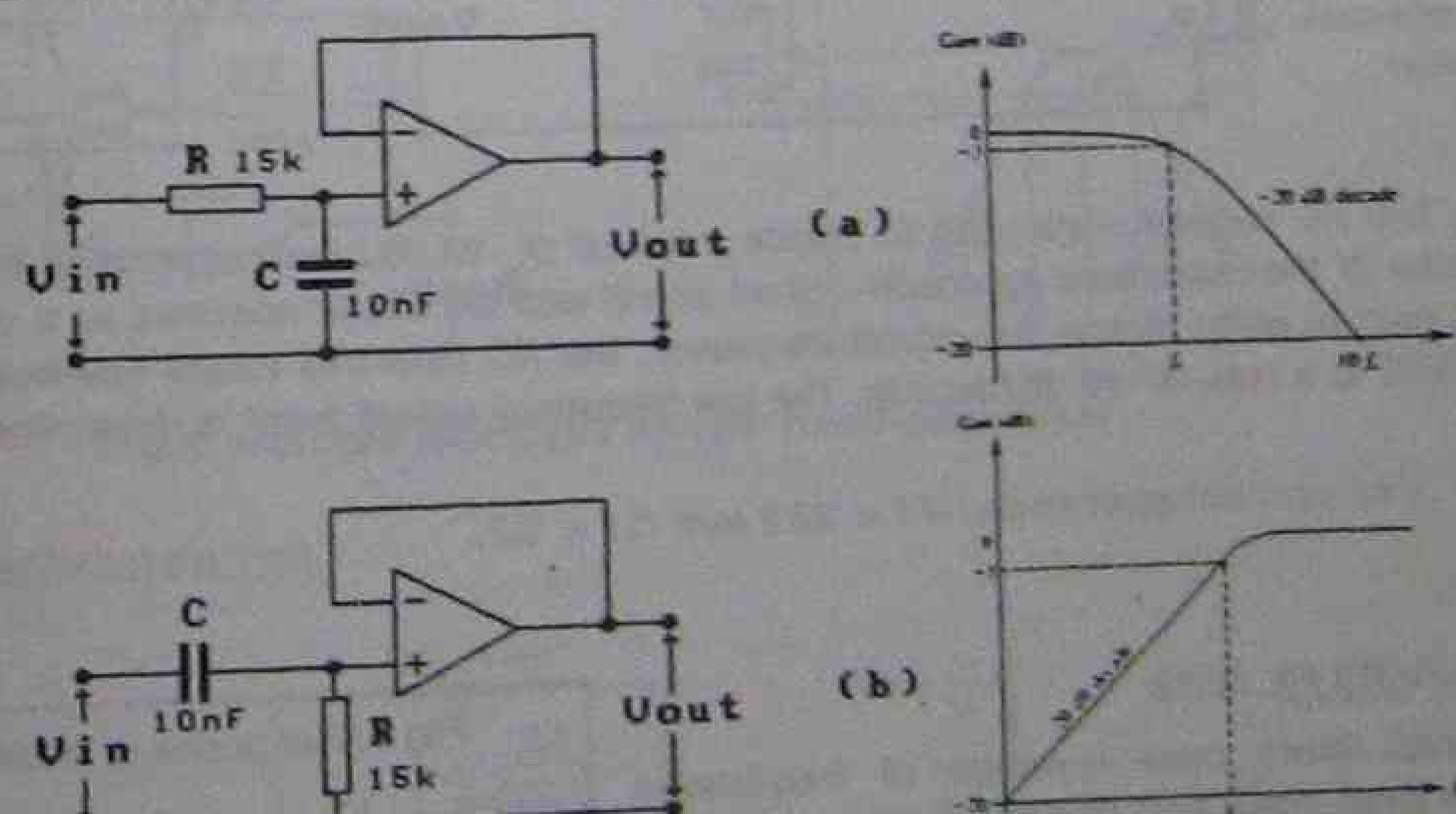
Active filters use RC networks to *'attenuate'* those frequencies outside the pass band. Fig.3(a) shows a first order low-pass filter with a roll-off above the cut-off frequency (f_H) of -20dB/decade.

- At frequencies below f_H, the reactance of the capacitor C is large compared to R, and the circuit behaves as a voltage follower with a gain (Av) of 1.
- When the reactance of the capacitor equals R, the gain of the circuit will fall to 0.71, (output is down by -3dB). This point is the *cut-off* frequency (f_H).
- As the frequency rises, the reactance of C will further decrease, and the gain of the circuit will fall, at a rate of -20dB for each decade (10 times) increase of the input frequency.

Fig.3: *First order filters*

(a) low pass

(b) high pass



For the first order high-pass filter shown in Fig.3(b).

- At low frequencies, the reactance of C is high, and the output voltage of the circuit will be less than the input. Thus the overall gain of the circuit is less than one.
- At the cut-off frequency, the reactance of C equals R, and the output voltage will be 0.71 times the input (gain of the circuit is 0.71).
- Above this frequency the reactance of C drops further and the circuit gain rises to unity.

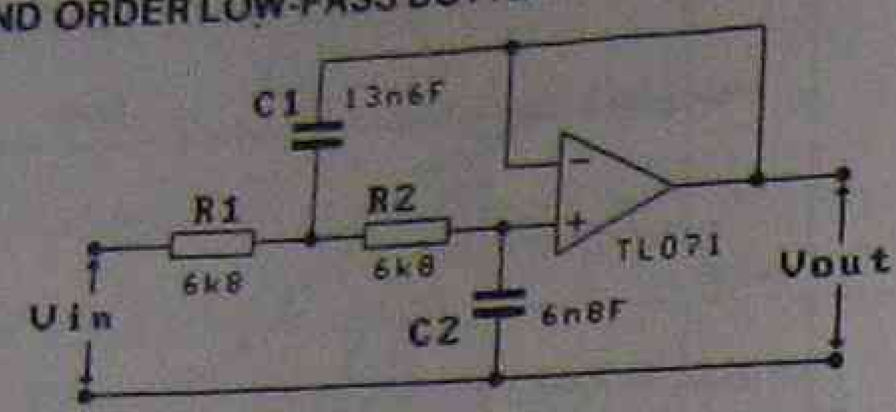
The cut-off frequency (when gain = 0.71) (-3dB Point) in both circuits can be calculated from:

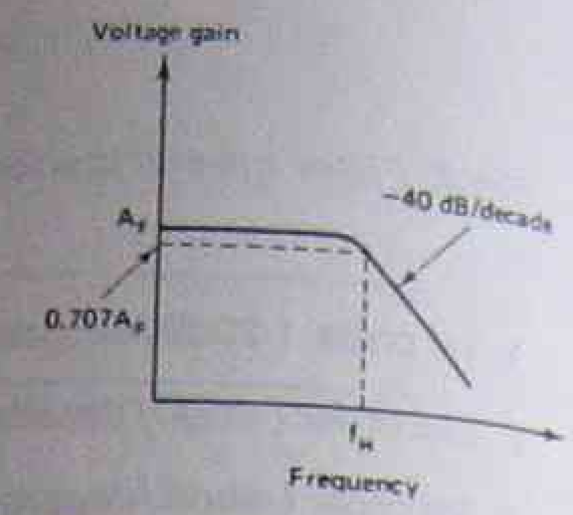
$$f_c = \frac{1}{6.28RC}$$

• **Equation 13.1**

2. THE SECOND ORDER LOW-PASS BUTTERWORTH FILTER

Fig.4 *Second order low-pass Butterworth filter.*





In the low pass filter of Fig.4, two RC networks are used to create a roll-off of -40 dB/decade. The high frequency cut-off point (when output level falls by 0.707, or -3 dB) can be determined by:

$$f_c = \frac{1}{6.28\sqrt{R_1R_2C_1C_2}}$$

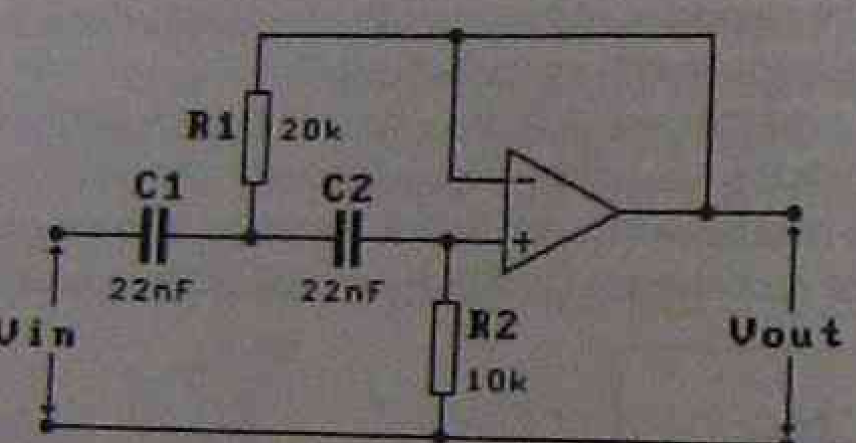
* **Equation 13.2**

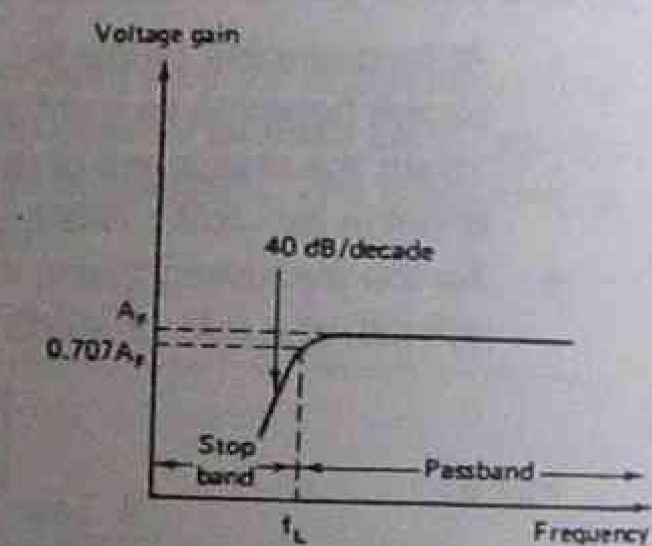
For correct operation, R1 = R2 and C1 = 2C2.

Below the cut-off frequency the reactance of the capacitors is large, and the circuit operates as a voltage follower with Av = 1, passing low frequencies. Above the cut-off frequency the reactance of the capacitors causes the voltage gain to reduce at the rate of -40 dB/decade.

3. THE SECOND ORDER HIGH-PASS BUTTERWORTH FILTER

Fig.5: *Second order, High-pass Butterworth filter.*





Again, two RC networks are used to create a roll-off of -40 dB/decade. At high frequencies, the reactance of the capacitors approach a short circuit and the circuit operates as a voltage follower with a gain of unity. Below the cut-off frequency, the RC networks cause low frequencies to be attenuated at a rate of -40 db/decade. The low frequency cut-off point is determined by equation 13.2.

- *For correct operation, R1 = 2R2 and C1 = C2.*

4. BAND-PASS FILTERS

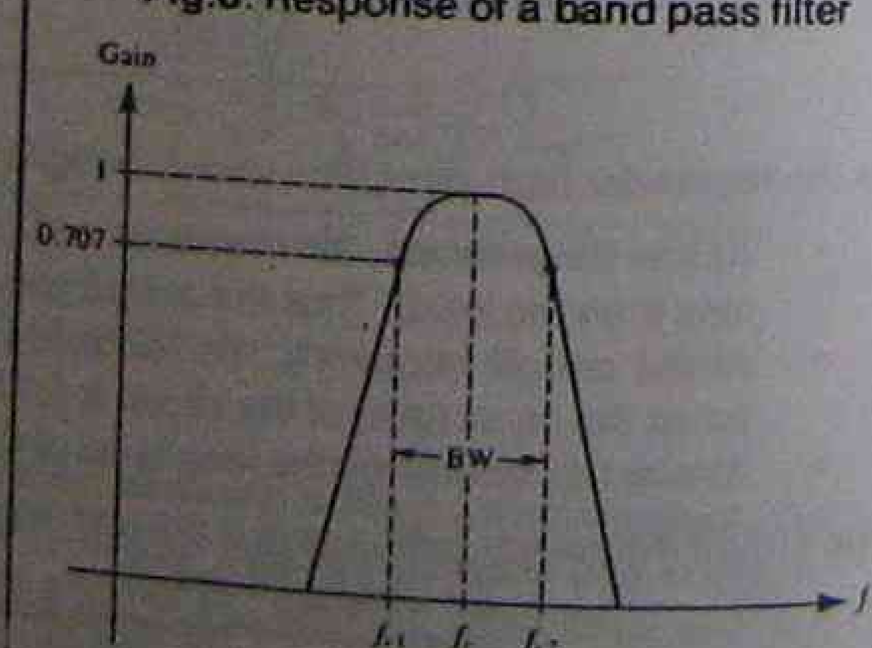
Band-pass filters pass a range of frequencies between their lower and upper cut-off frequencies, as shown in Fig.6. The range of frequencies between f_L and f_H is termed the bandwidth (BW) where:

- $BW = f_H - f_L$

The frequency mid way between the cut-off points is the centre frequency or resonant frequency, fr. Band-pass filters also have a *'Quality factor'*, called Q, in which:

- $Q = \frac{\text{resonant frequency}}{\text{bandwidth}}$

Fig.6: Response of a band pass filter



The value of Q is an indication of the *selectivity* of the pass-band. A filter with Q less than 10, is classified as a wide band filter, and a filter with Q equal to or greater than 10 is a narrow band filter. In terms of frequency, a wide-band filter has a bandwidth greater than 10 times the centre frequency, and a narrow band filter has a bandwidth less than one tenth of the centre frequency.

5. THE SECOND ORDER BAND-PASS FILTER:

Fig.7: *The second order band-pass filter.*

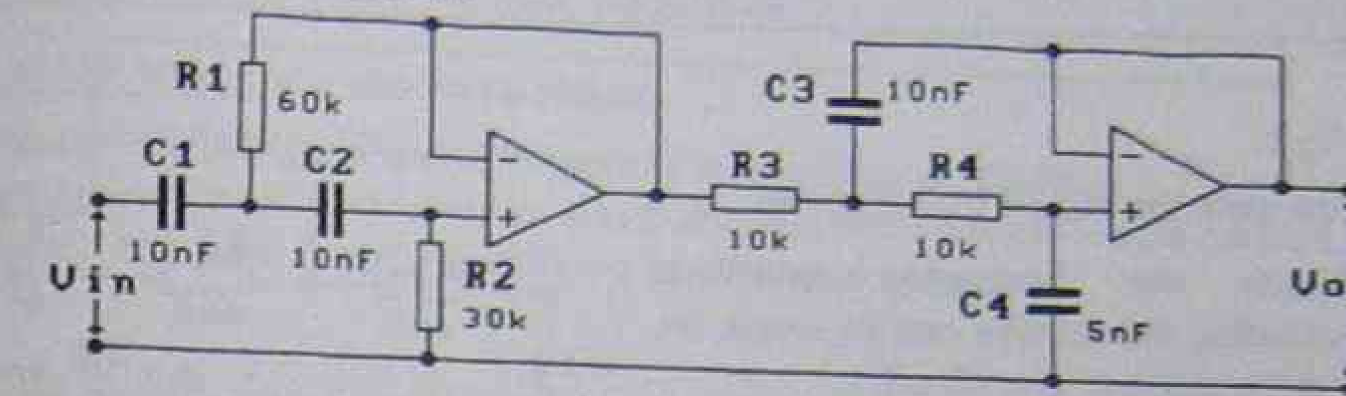
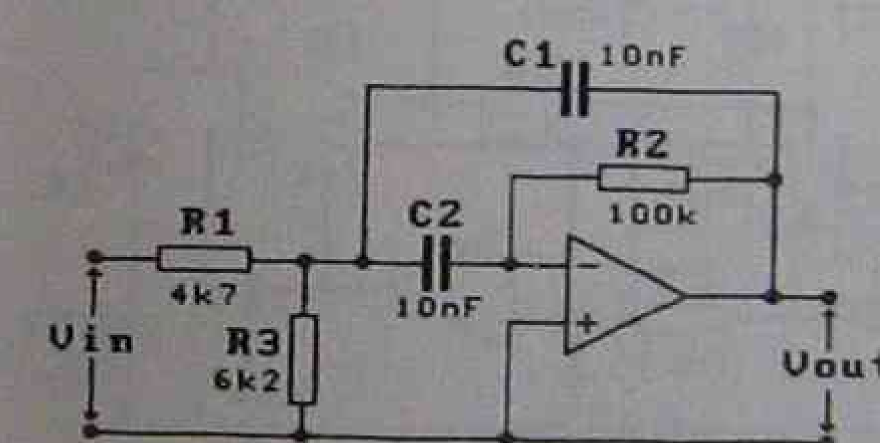


Fig.7 comprises a second order high-pass filter followed by a second order low-pass filter. The cut-off frequencies are determined with the same equations used for the individual filters (equation 13.2). This circuit is a wideband filter, with a slope of -40 dB/decade beyond each cut-off frequency. The centre frequency of the pass-band can be calculated using the equation:

$$f_r = \sqrt{(f_L \times f_H)}$$

* **equation 13.3**

An improvement on this circuit is shown in Fig.8. Apart from only requiring a single op amp, it can be designed as either wide band or low band by selecting suitable values of each resistor and capacitor. The equations for this circuit are outside the scope of these notes but for the values shown in Fig.8, the centre frequency is 973Hz, and the Q of the circuit is 3.



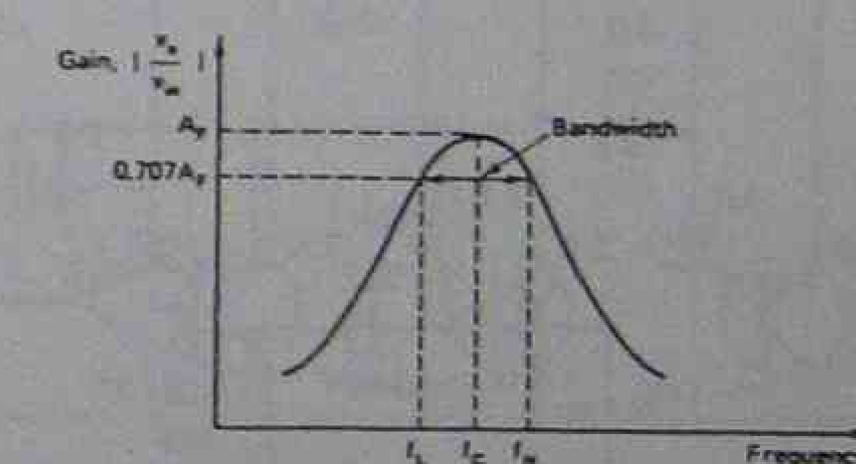
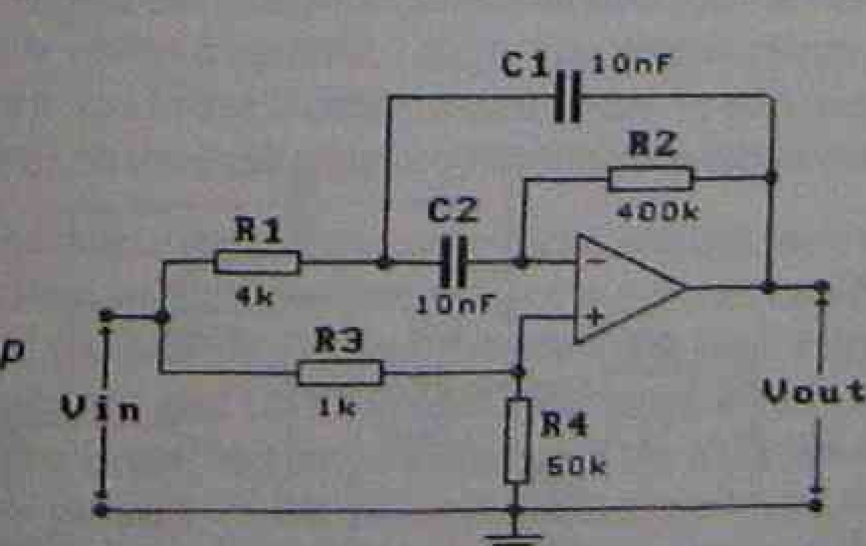
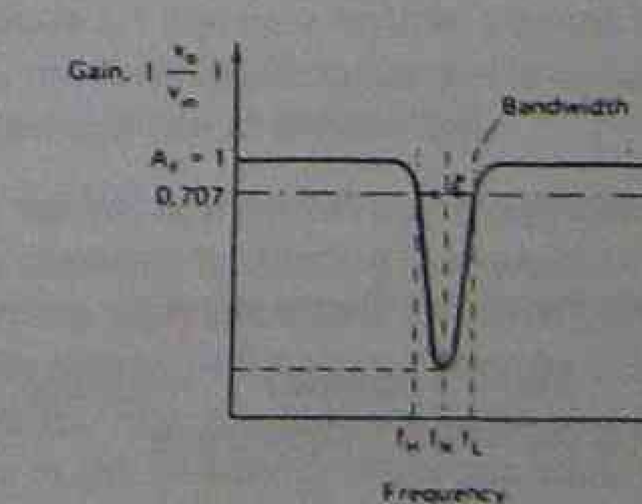


Fig.8: *Multiple feedback narrow band-pass filter.*

6. BAND-STOP (NOTCH) FILTER

Fig.9: *The band-stop filter*





The band-stop filter, also called a *Notch filter*, is used to reject an unwanted frequency. Various circuit configurations are possible, including that shown in Fig.9, which is a variation of Fig.8 in which the input signal is also applied to the (+) input by R3 and R4. Usually R4 is made adjustable to allow the circuit to be 'tuned'. The centre frequency is calculated using equation 13.2

****

TUTORIAL WEEK 14

1. For Fig.1:
   (a) Calculate the cut-off frequency f_H.
   (b) Sketch the frequency response, indicating the relative output levels, f_H, and the slope of the roll-off.

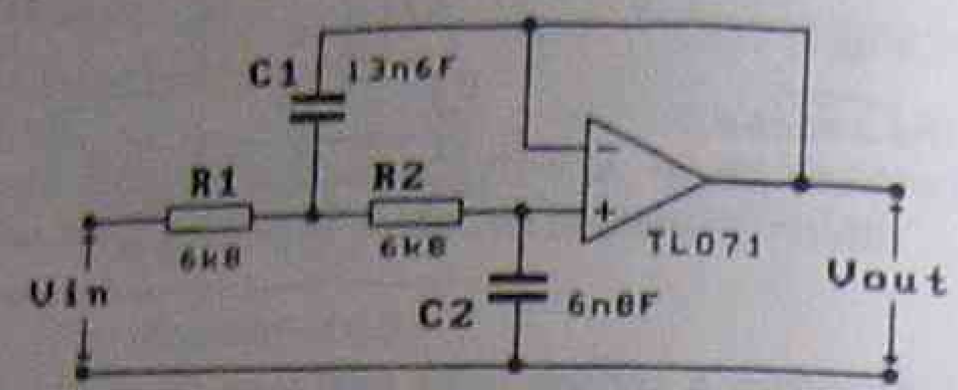


Fig.1.

2. For Fig.2:
   (a) Calculate the cut-off frequency f_L.
   (b) Sketch the frequency response, indicating the relative output levels, f_L and the slope of the roll-off.

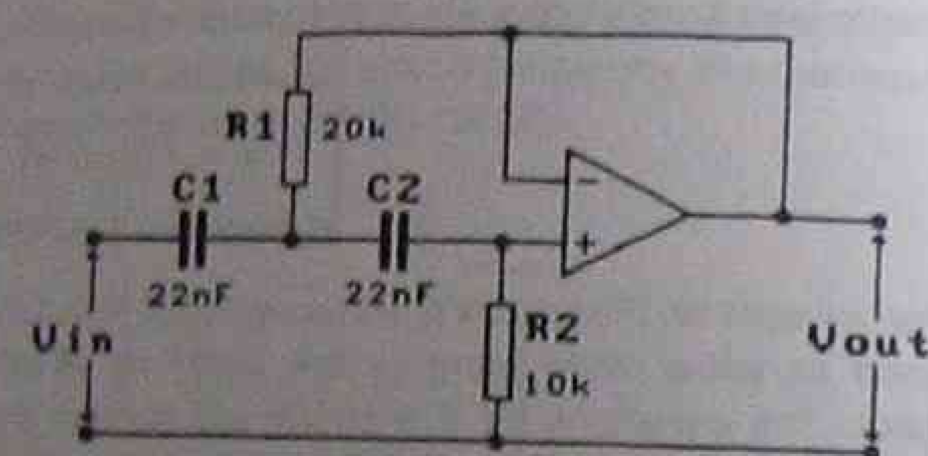
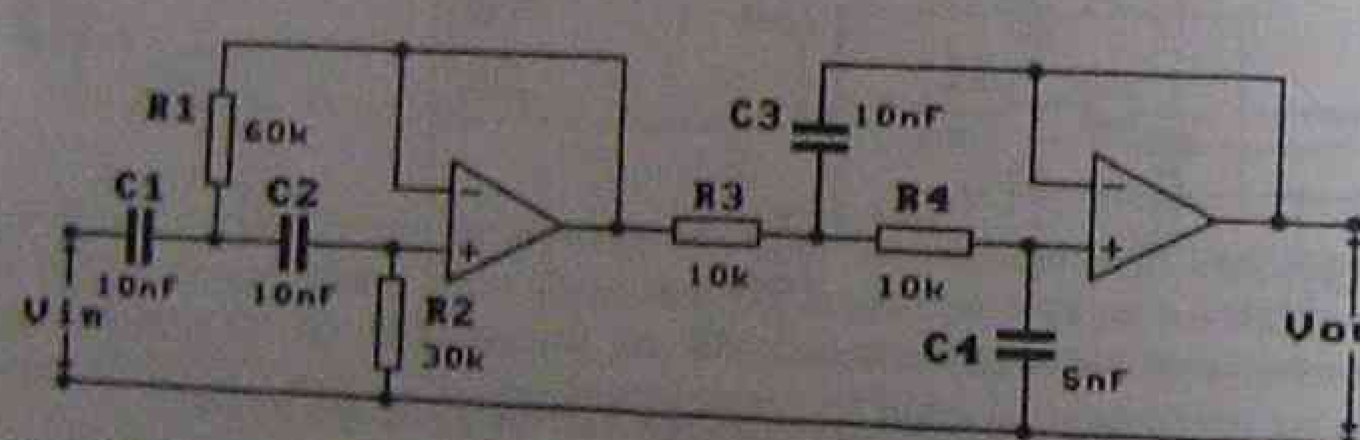


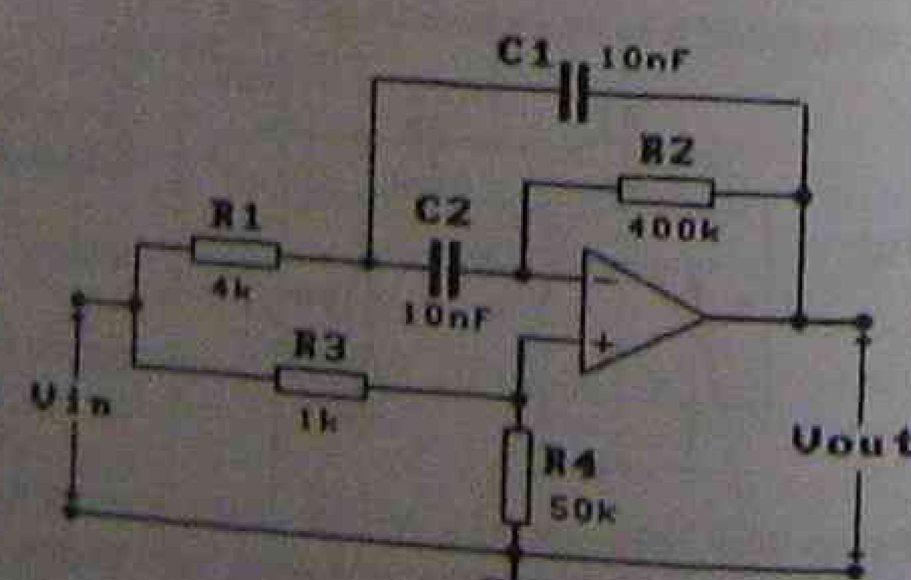
Fig.2.

Fig.3



3. For Fig.3:
   (a) Calculate the cut-off frequencies f_L and f_H.
   (b) Sketch the frequency response, indicating the relative output levels, f_L, f_H and the slope of the roll-off.

4. For Fig.4:
   (a) Calculate the centre frequency (f_c).
   (b) Sketch the frequency response, indicating the centre frequency f_c.



****

Fig.4

THEORY LESSON 15

MULTISTAGE OP AMP CIRCUITS

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

(a) *Identify the op amp configurations used in typical multistage circuits.*
(b) *Analyse the performance characteristics of each stage within a multistage circuit.*
(c) *Describe a procedure for fault analysis within a multistage circuit.*

INTRODUCTION

Multistage circuits that use op amps generally contain a combination of the circuits that have so far been described. Often the practical configurations are more complex than the basic circuits, as they have added components to tailor a response or to eliminate noise signals. These notes examine two multistage op amp circuits.

NOTE: This lesson should be presented as an interactive tutorial.

1. A LIGHT LEVEL INDICATOR:

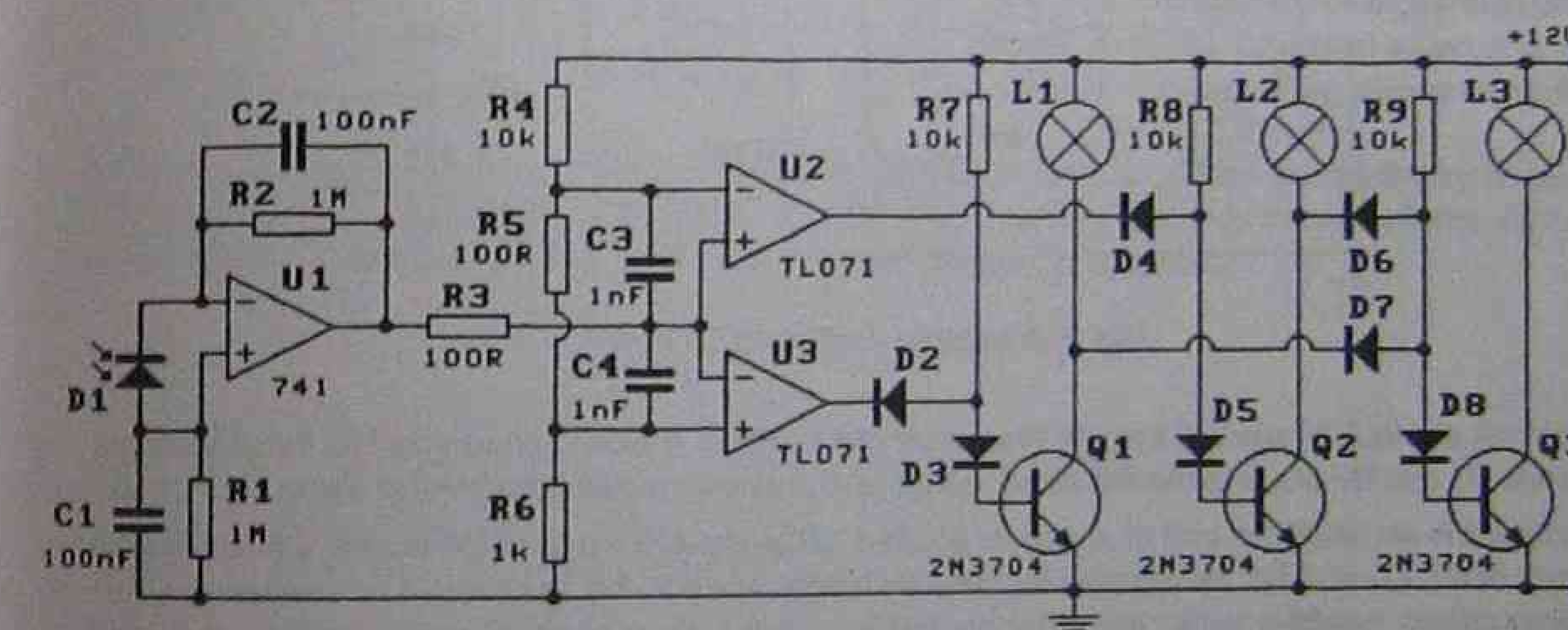


Fig.1: *A light level indicator*

The circuit of Fig.1 is used to check the illumination of a light source and to give an indication whether it falls within a specific range. The silicon photo diode D1 allows a reverse current to flow that is proportional to the incident illumination. In darkness, the reverse photo-current is about 1nA, and depending on the radiant energy striking the diode current can rise to around 2µA.

Op amp U1 converts the photo-current to a voltage, as the current flowing through the photo diode also flows in R1 and R2. The sensitivity of the circuit can be changed by altering the values of these two resistors. Capacitors C1 and C2 eliminate high frequency signals picked up as noise at the input. Op amps U2 and U3 form a window comparator, with reference voltages determined by R4, R5 and R6. The circuit therefore indicates if the light level is too low, too high, or within the preset 'window'. The network of R3, C3 and C4 reduce instability in the switching characteristics of the comparators. By replacing R5 with a 50 ohm resistor a narrower window can be created.

CLASS EXERCISES:

(a) Determine the output voltage of U1 when the diode current is 1nA (dark) and 2uA (light).
(b) Determine the upper and lower threshold voltages V_{LT} and V_{UT} for the window comparator of U2 and U1.

(c) Determine which lamp indicates:

- light level too low.
- light level too high.
- light level within limits.

2. A SAMPLE AND HOLD CIRCUIT:

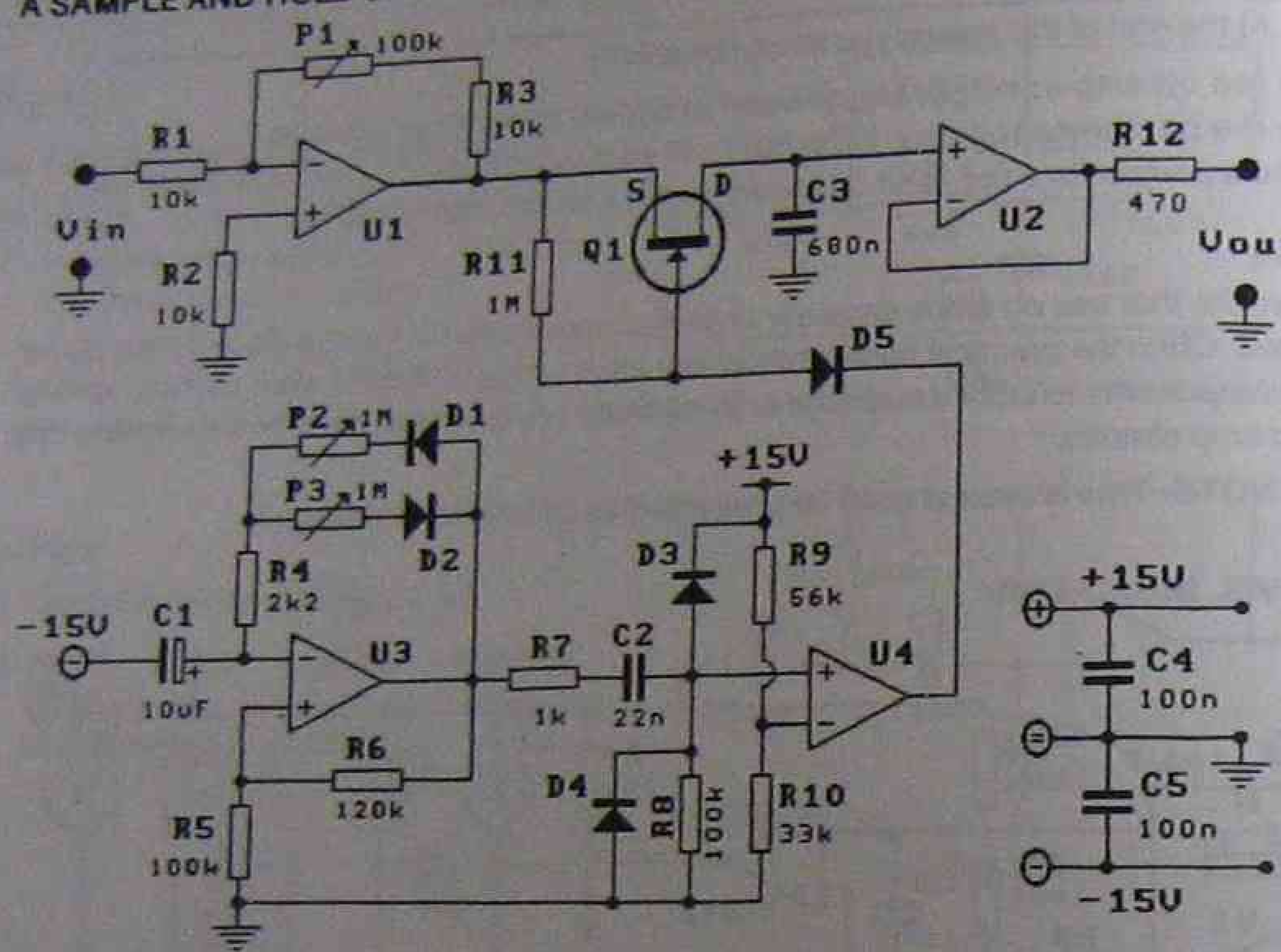


Fig.2: A sample and hold circuit

The circuit of Fig.2 is used to sample then store the value of a slowly changing DC input voltage. Op amps U1 and U2 process the DC input voltage and present the sampled level at the output of U2.

Op amp U3 is an oscillator and its output is applied to the input of op amp U4 via the differentiating network of R7, C2 and R8. Because of the differentiating network, the input to U4 is a series of short duration pulses, positive going when the oscillator output swings positive, and negative when the oscillator output swings negative.

Op amp U4 is wired as a comparator and at each positive transition of the oscillator the output of the comparator switches positive, reverse biasing diode D5. Under these conditions, the gate/source voltage of FET Q1 will equal 0V, allowing it to turn on, thus causing C3 to charge to the output level of U1. As the duration of the pulse at the input of U4 is very short, the output of U4 quickly switches negative again, forward biasing D5 and turning off the FET.

CLASS EXERCISE:

(a) Determine the maximum and minimum gains for amplifier U1.

(b) Determine the voltage gain of amplifier U2.

(c) Explain the purpose of amplifier U2.

(d) If potentiometers P2 and P3 are set to zero ohms, determine the frequency of oscillation of U3.

(e) Repeat (d), but with the potentiometers set to their maximum values.

(f) Describe the effect on the output waveform of U3 if P2 and P3 are set to diffferent values.

(g) Determine the value of the reference voltage for U4.

(h) Determine the output voltage levels of U4.

3. FAULT FINDING PROCEDURES:

Fault finding usually requires systematic measurements of voltages within the circuit combined with correct interpretation of the measurements. An over-riding factor is whether there is a circuit diagram available. If so then:

1. Measure the DC supply voltage *on the actual supply pins* of each IC using a high impedance voltmeter (DVM).
2. Measure any DC reference voltages, keeping in mind the input impedance of the measuring instrument. Any input signal(s) should be set to 0V.
3. Measure the DC voltage level at the output of each op amp. This must be done with the input signal(s) set to 0V. Usually the output voltage should be zero, except for comparators and amplifiers using mid point DC biasing. The output of a comparator should give an output measurement within one or two volts of either of the two supply rails
4. If all DC conditions appear satisfactory, connect the input signal to measure the AC conditions within the circuit. Either an oscilloscope or, if the input signals are below a few kHz, with a digital multimeter on the AC range. High frequency measurements should be performed with an oscilloscope.
5. Trace the AC signal through the circuit to locate the faulty section. In a complex circuit, it is helpful to first determine if the correct signal appears at a point half way within the circuit, and to work either backwards or forwards from this point, (half split method). If a signal is lost, it is often due to a fault in the preceding stage, hence check both present and preceding stages carefully.

Fault finding without manufacturers' diagrams is more difficult and it may be necessary to sketch part of the diagram from the printed circuit board before fault analysis may be undertaken. Once an understanding of the circuit is attained the method of analysis is as above.

****

TUTORIAL WEEK 15

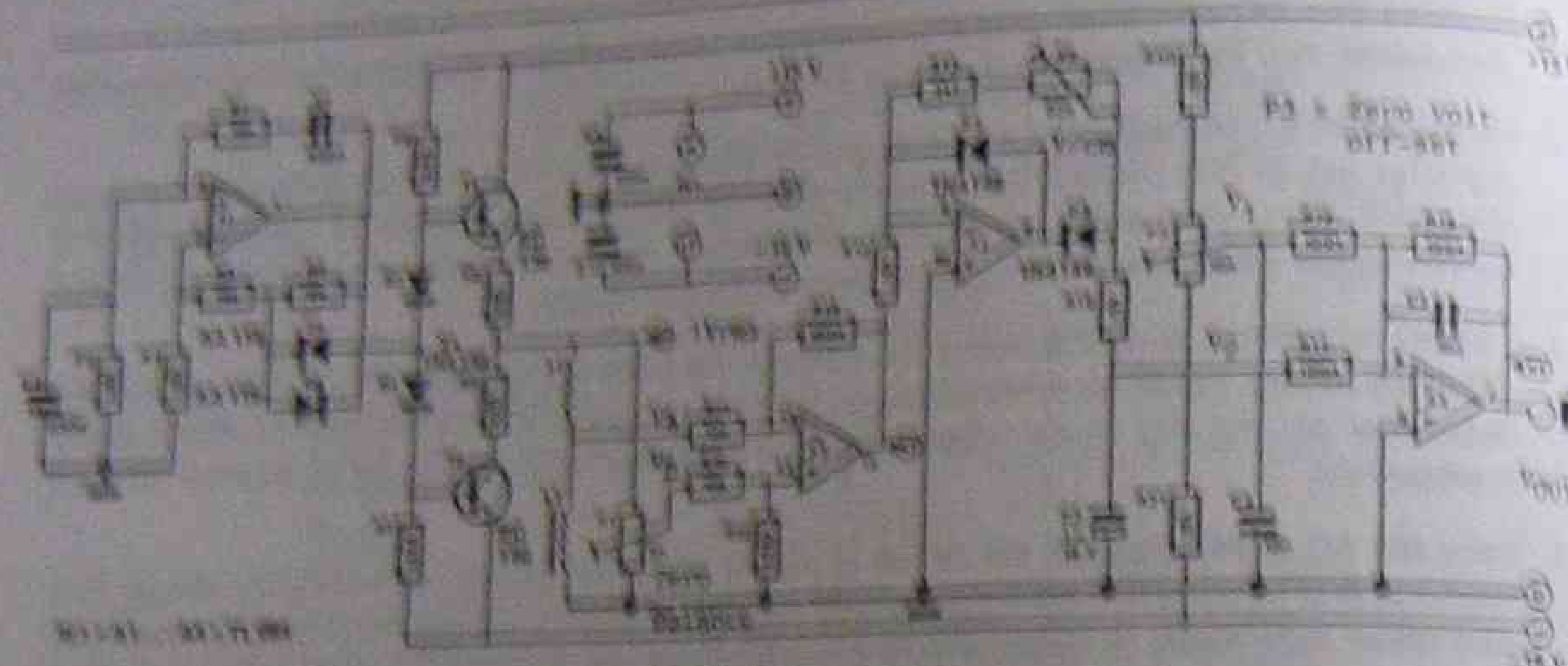
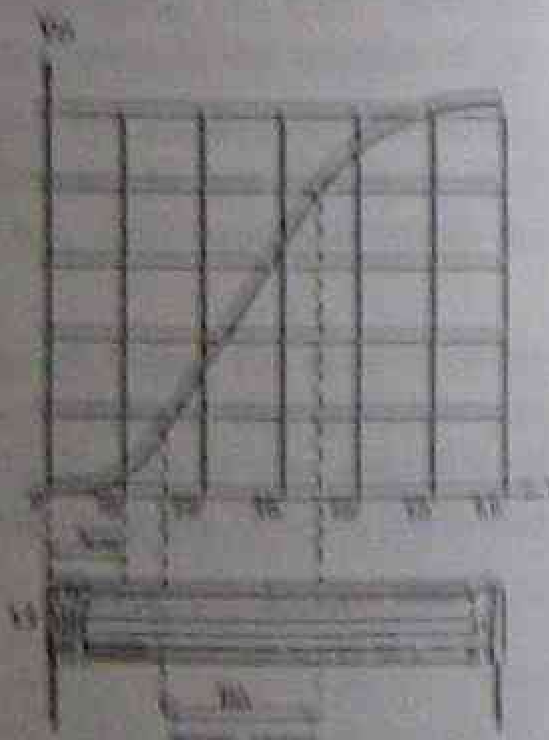


Fig 1: *Inductive sensor*

The circuit of Fig 1 is used to measure the distance travelled by a shaft. The principle of operation is based on changing the inductance of L1 and L2, situated in one arm of a Wheatstone bridge. The second arm of the bridge is the two resistance legs of potentiometer P1. The AC voltage, V_{AB} (between R10 and R11) is initially zero, set for zero displacement of an iron core in L2, attached to the shaft. As the shaft moves, it causes an unbalance in the inductance of the bridge and the voltage V_{AB} increases in proportion to the distance travelled. The circuit amplifies the AC voltage and converts it into a DC voltage proportional to distance travelled. The table in Fig.2 shows the output voltage in mV against displacement in millimeters.

Fig.2 *Graph and table showing the relationship between the movement of the shaft and the output voltage. The circuit is calibrated to operate in the linear region of the curve.*

X(mm)	Vo(mV)
[illegible]	[illegible]



ASSIGNMENT

1. Identify the oscillator configuration of op amp A1, and calculate its frequency of operation.
2. State the purpose of the network comprising R3, R4, R5, D1 and D2.
3. Identify the function of the circuit around transistors T1 and T2.
4. Identify the amplifier configuration of op amp A2, and calculate its voltage gain.
5. Identify the function being performed by A3, and sketch the waveform at the anode of D6.
6. Sketch the waveform that will appear across C3.
7. Identify the function being performed by A4. (Note, capacitors C4 and C5 can be ignored)
8. Write an equation to determine the output voltage of A4 in terms of V_1, V_2, R17, R18 and R19.

\$\$\$\$\$\$\$\$\$\$\$\$\$\$\$\$\$\$

THEORY LESSON 16

REFERENCE: *Specifications from various manufacturers*

POWER OP AMPS

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

(a) *List the categories of power op amps.*
(b) *List typical applications for power op amps.*
(c) *Apply manufacturers data to determine the short circuit current limit or limiting resistance value.*

INTRODUCTION

Op amps are available for applications that require an extension of one or more electrical parameters. Generally these ICs are categorized as *POWER OP AMPS*, although the word 'power' often means many of these devices only have one enhanced parameter. These devices can be grouped into those with the following characteristics.

1. High output current.
2. High output voltage.
3. High output power.

Power op amps have complementary output stages, comprising components designed to carry the enhanced ratings. Many of these devices also have overload sensing circuitry, which is used to shut-down or limit the load current of the op amp if its thermal ratings are exceeded. Note that power op amps are generally expensive, costing as much as $100.

1. POWER OP AMPS - ENHANCED CURRENT

Op amps with ratings that state 'high output current' fall into two categories: those with a small output power capability, and those able to deliver larger output powers.

1.1 HIGH CURRENT: LOW POWER TYPES

An example of an enhanced small power op amp is the National Semiconductor type LM 13080. This IC is packaged in a standard 8 pin DIP package and can sink or source an output current up to 250mA, while operating from supply rails ranging from ±3V to ±15V. This IC is rated at 1.25W at 25°C. Fig 1 shows the internal diagram and package outline.

The input stage of this op amp can be set with an external resistor connected to pin 7. If this input, together with pin 2 is connected to ground, bias currents can flow and the output is enabled. If these two pins are open circuit then the output of the op amp will be turned off.

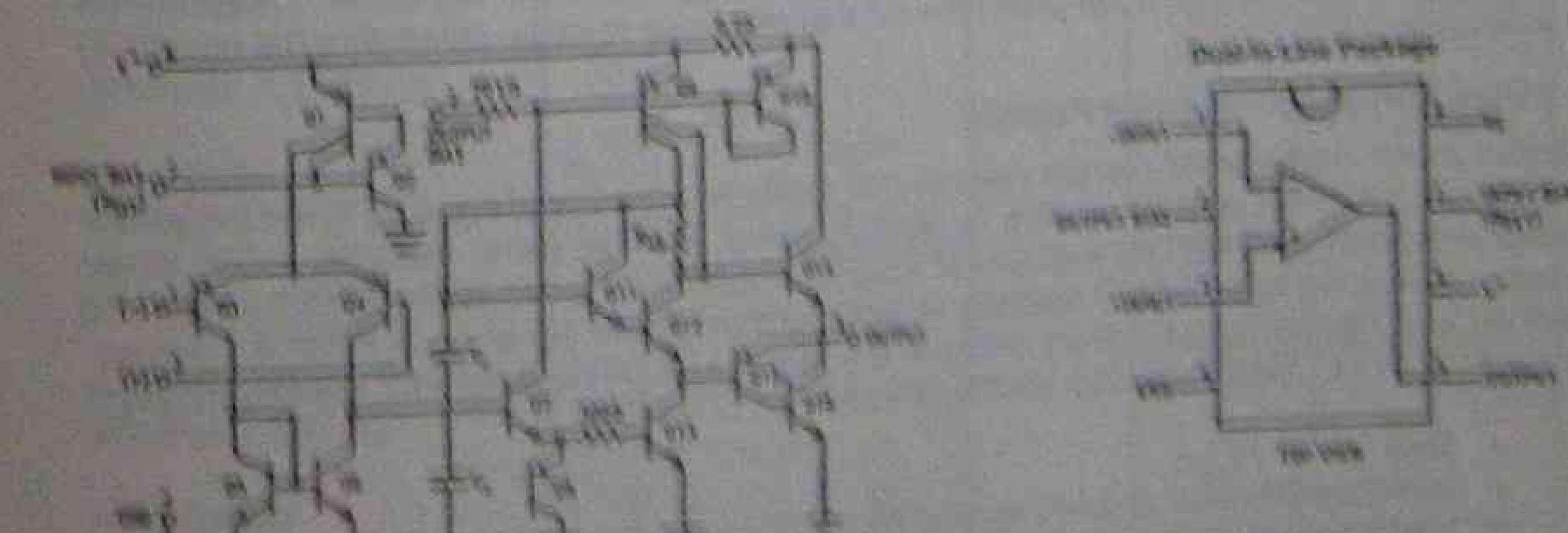
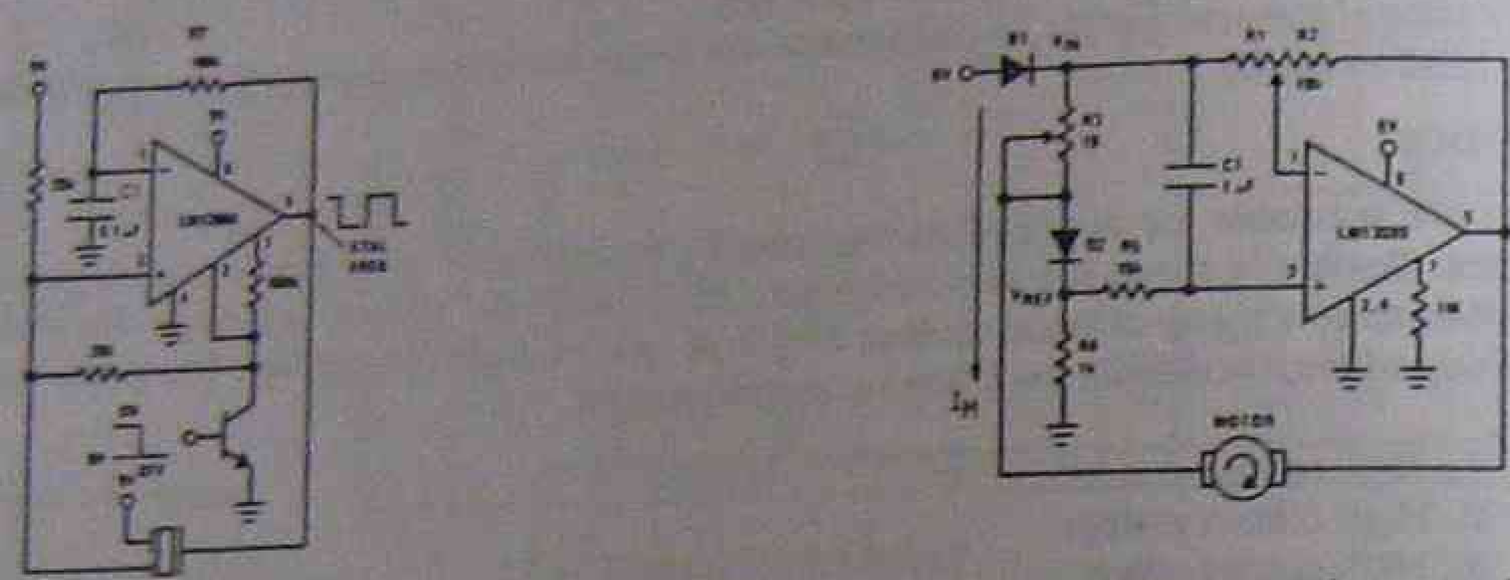


Fig1: *Internal schematic and case outline of the LM 13080*

1.2 APPLICATIONS OF THE LM13080

Fig.2(a) shows a circuit for a piezo-electric alarm, which uses a 3-terminal piezo transducer (Gulton 101FB or equivalent) to produce an 80dB alarm. The op amp operates as the oscillator and output driver for the load. The alarm transducer has a feedback terminal which is connected to the non-inverting input of the op amp, in a positive feedback path, causing oscillation to occur at the resonant frequency of the piezo-electric alarm. Transistor Q1 is the switch for the alarm, and if the transistor is turned on by applying 0.6V to its base, the alarm will sound.

R_F and C_1 are used to provide a reference voltage at the inverting input and keep the duty cycle of the oscillator close to 50%. The RC time constant of these two components should be at least ten times greater than the period of oscillation.



(a) Piezo-electric alarm. *(b) Motor speed control.*

Fig.2

The LM13080 can also be used to control the speed of a small DC motor that requires less than 0.5A start current. In Fig.2(b), R3, D2 and R4 are used to provide a reference voltage at the non-inverting input of the op amp. This voltage is summed together with a feedback voltage caused by the current flowing through the motor and R3. The op amp acts as a difference amplifier, amplifying the difference between V_{REF} and V_{IN}, as V_{IN} is also applied to R1, and appears as a common mode voltage to both inputs. The 10k speed adjust potentiometer, referred to as R1/R2, is used for both the input resistor and the negative feedback resistor. The voltage across the motor will be:

$$V_{MOTOR} = \frac{R2 \times (V_{D2} + I3R3)}{R1} + V_{D1}$$

Positive feedback is developed as a change in voltage across R3, due to the change in the motor current caused by a variation in the load driven by the motor. Potentiometer R3 adjusts the amount of feedback, controlling the speed of the motor. If the load on the motor increases, the speed will fall and the motor current increase. This will increase the feedback voltage drop across R3, which is then amplified to increase the voltage applied to the motor. Capacitor C1 and resistor R5 are used as a low pass filter for the reference voltage, preventing the possibility of oscillation.

1.3 HIGH CURRENT: MEDIUM POWER

Medium power, high output current devices are available with output currents in the range of 1 to 5 Amps. This category of op amp is used in areas such as alarm systems, small motor control and audio systems. An example of this type of op amp is the Siemens TCA 2365, dual op amp, shown in Fig.3. This IC consists of two identical power op amps in a SIP-9 package. Each op amp has an output current rating of 2.5 amps, and can operate with supplies ranging from ±4V to ±13V. Features of this device include:

- Slew rate of 4V/µs.
- Internal protection circuitry against short circuits to the positive or negative supplies.
- Internal protection circuitry against thermal overload.

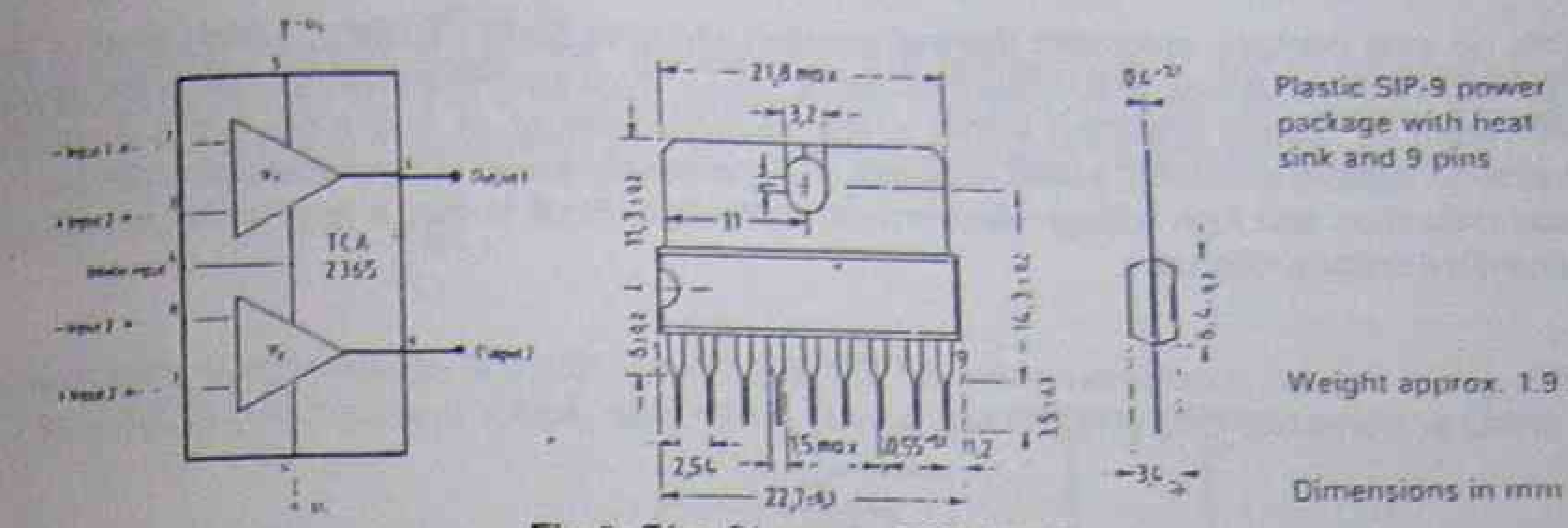
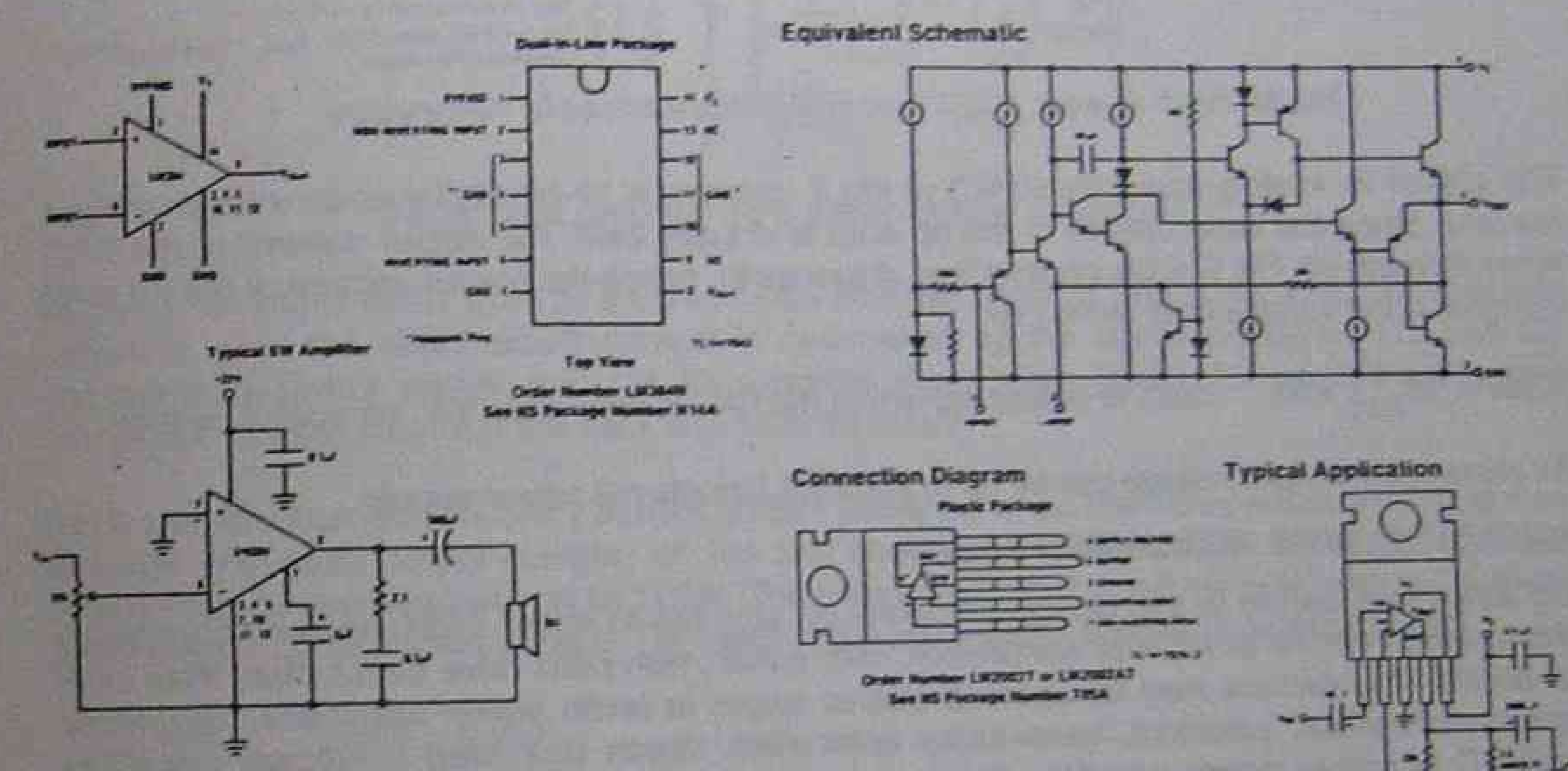


Fig.3: *The Siemens TCA 2365*

National Semiconductor make a range of audio power amplifiers, in power ratings from 1W to 8W. Fig.4 shows a 5W and 8W audio power amp, both designed to supply peak currents of about 3.5A with high peak-to-peak voltage swings across low impedance loads. Internal current limiting and thermal overload circuitry is included. Sometimes, voltage gain can be set by external resistors, as with general purpose op amps.



(a) LM384, 5W op amp. (b) LM2002, 8W op amp

Fig.4: *Audio power amplifiers*

2. POWER OP AMPS - ENHANCED VOLTAGE

Burr-Brown components, distributed by Kenelec make a range of high voltage op amps, such as the 3584 shown in Fig.5. This IC is rated for output voltage swings of ±145V with 50% overload.

- JFET input
- Slew rate = 150V/µs
- Input bias current = 20pA
- CMRR = -110dB
- Gain-bandwidth product = 50MHz
- Short circuit current limit = ±25mA

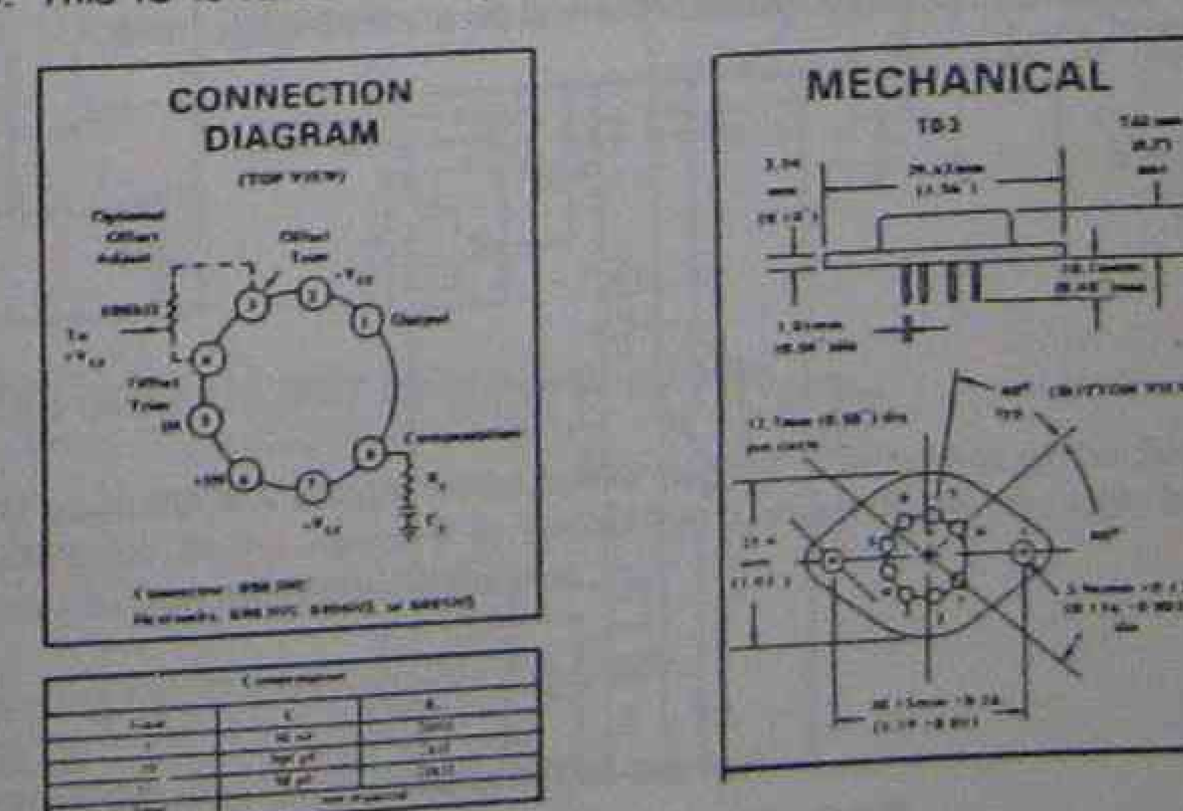
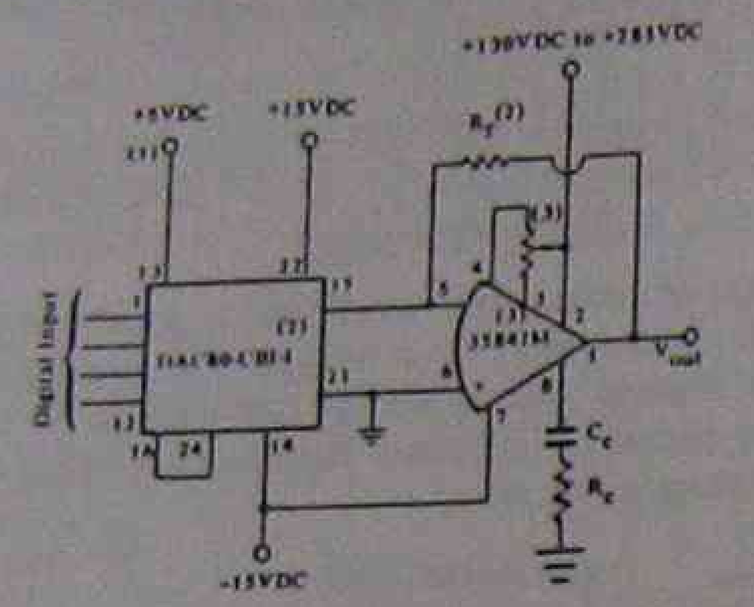


Fig.5: *Pin connections of Burr-Brown 3584*

This op amp contains automatic thermal sensing and shut-down circuitry, which comes into operation if the substrate temperature exceeds 150°C. Once an overload is removed the device will return to normal operation within a few seconds.Devices of this type find use in DC precision digitally controlled power supplies, programmable waveform generators, cathode ray tube deflection and high voltage electrostatic deflection. Fig.6 shows a high voltage digitally controlled voltage reference.



NOTES:
1. May be connected to +15VDC.
2. Use for DAC gain adjust. $V_{out} = (I_{DAC\ OUT})(R_f)$.
3. Optional offset adjust.

Fig.6: *High speed, high voltage programmable reference*

The digital to Analog converter (DAC) of Fig.6 converts a 12-bit digital code into an output current. Since the input current of the op amp is virtually zero, the output current of the DAC flows through RF. Pin 5 of the op amp is at virtual earth, hence the output voltage of the op amp is equal to the voltage drop across RF.

$V_{OUT} = IO_{DAC} \times R_F$ thus the output current of the DAC is converted into an output voltage.

As shown, the output voltage can be up to 230V for a single rail power supply.

3. POWER OP AMPS - ENHANCED POWER

Op amps that belong to this category operate on supplies to about ±40V, and are capable of dissipating from 10 to 100W of continuous RMS power, with peak drive capabilities from 50W to 800W. The devices may be used as output stages in audio power amplifiers, DC motor speed controllers, actuators, servo-motor controllers, floppy disk head positioner amplifiers and DC precision power supplies. Enhanced power op amps generally require an external resistor to program the output current limit to the load. Manufacturers specifications usually provide an equation to to determine the value of the current limit resistor.

3.1 APPLICATIONS OF THE LH0101 POWER OP AMP

Fig.7 shows a JFET input National Semiconductor power op amp type LH0101, a device which can dissipate up to 60W when mounted on a suitable heat sink.

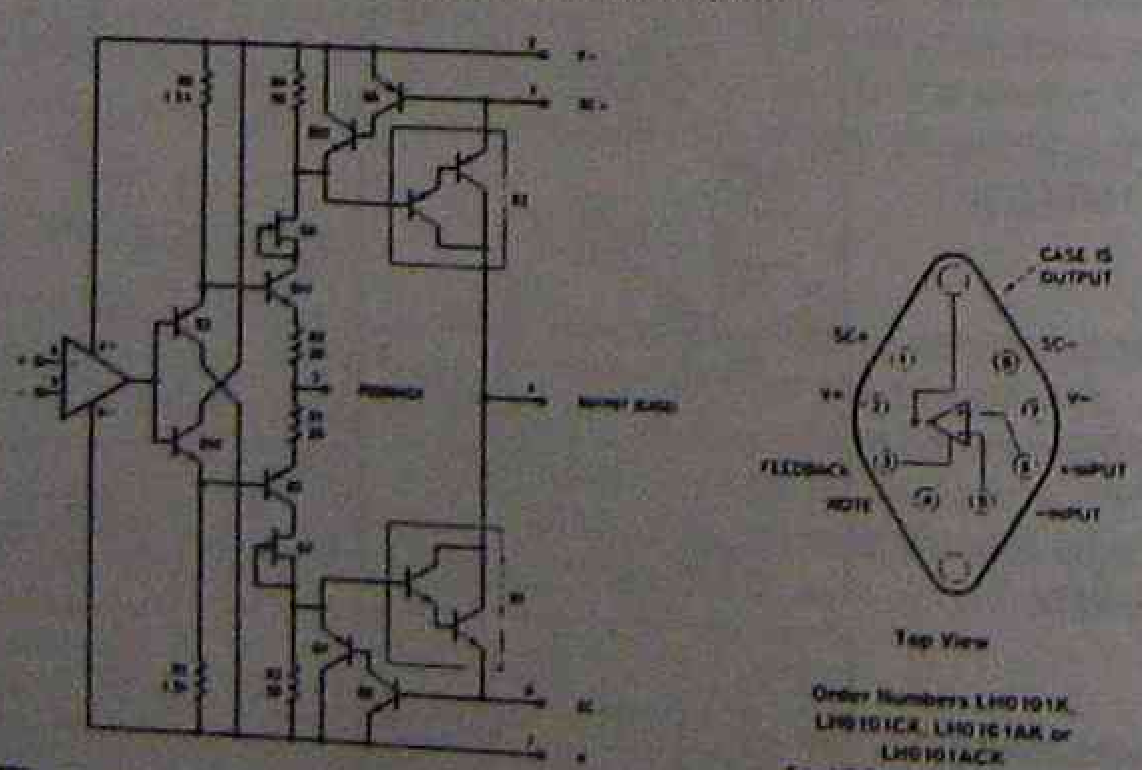


Fig.7: *The NS, JFET input, LH0101 power op amp*

Two external resistors are USED TO set the current limit, one between pins 1 and 2, (in parallel with internal transistor Q6), and a second between pins 7 and 8 (in parallel with internal transistor Q9). Each resistor is in series with the load current. The value of the resistor is calculated by:

$$R_{SC} = \frac{0.6V}{I_L}$$

Maximum supply voltage is ±22V, peak current is 5A with a continuous current rating of 2A. Input bias current is about 300pA, slew rate is 10V/μs and the unity gain-bandwidth is 5MHz.

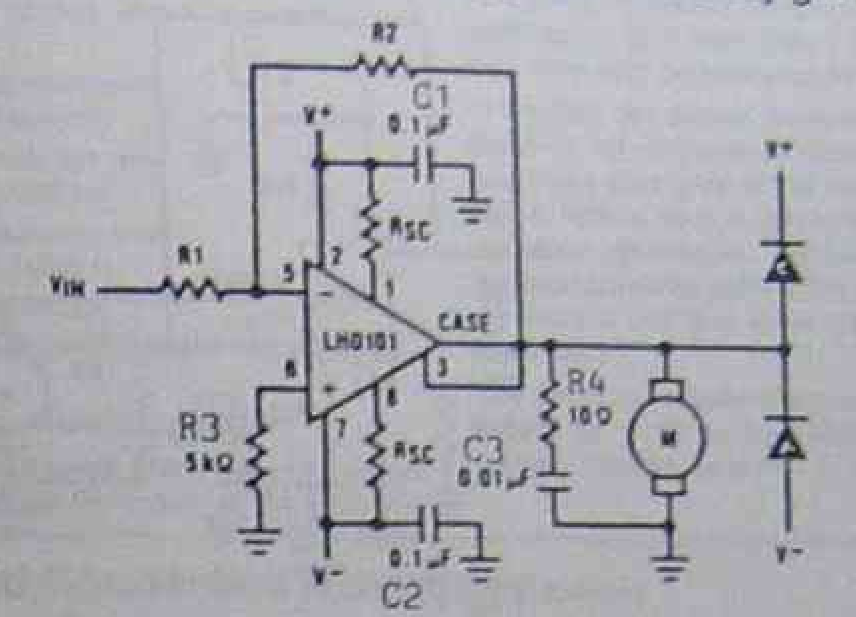


Fig.8: *The LH0101 as a servo-motor amplifier*

Fig.8 shows the LH0101 used as a servo-motor amplifier. Voltage gain is set as for an inverting amplifier, and the short circuit current is determined by the above equation. C1 and C2 decouple the power supply, and C3, R4 suppress motor noise. Diodes D1 and D2 are typical 1A diodes used to suppress the back emf from the motor.

Fig.9 shows a Burr-Brown, JFET input, OPA541 power op amp. This IC operates from supplies of ±40V with peak output currents of 10A and continuous currents of 5A. With correct heat sinking this IC can dissipate up to 125W. The load current limit is set via one external current limiting resistor, as shown in Fig.9. The value of the current limit resistor is determine by;

$$R_{SC} = \frac{0.809}{I_{LIM}} - 0.057$$

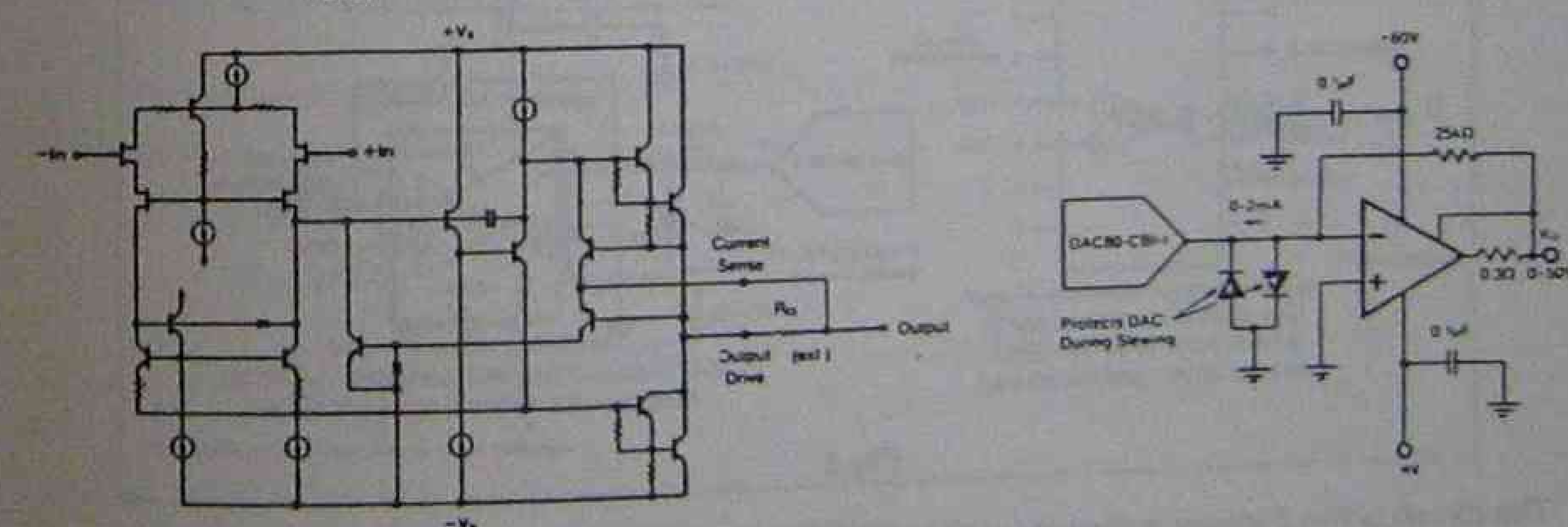


Fig.9: *The OPA541, JFET input power op amp.*

Fig.9 also shows the OPA541 as a digitally programmed power supply. The output of the DAC is a digitally encoded current value in the range 0 to 2mA. This current flows through the feedback resistor of the op amp so that the output voltage is:

$V_{OUT} = I_F R_F$ causing VOUT to range from 0 to +50V.

In this circuit RSC = 0.3Ω, setting the short circuit current limit at 2.27A.

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

TUTORIAL WEEK 16

$$R_{SC} = \frac{0.6V}{I_L}$$

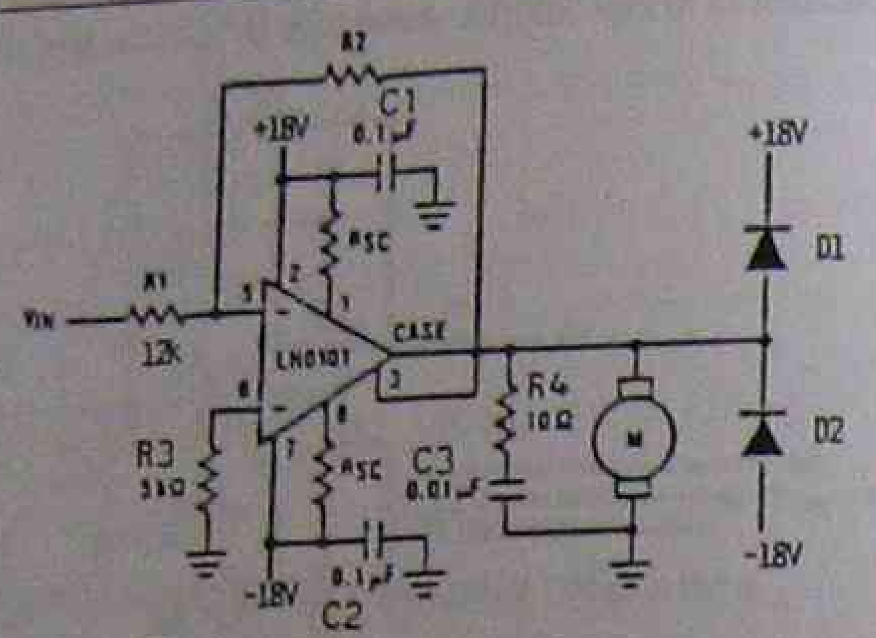


Fig.1

1. The circuit of Fig.1 is used to control the speed of a small DC motor. The motor has a maximum rating of 12V at 2A.

   (a) Calculate the value of R_{SC} to limit the current to 2.1A.

   (b) Determine the value of R_2 if an input voltage of 1V is to cause the motor to run at full speed.

   (c) Calculate the total power dissipated in the motor and the power op amp at maximum ratings of the motor.

$$R_{SC} = \frac{0.809}{I_{LIM}} - 0.057$$

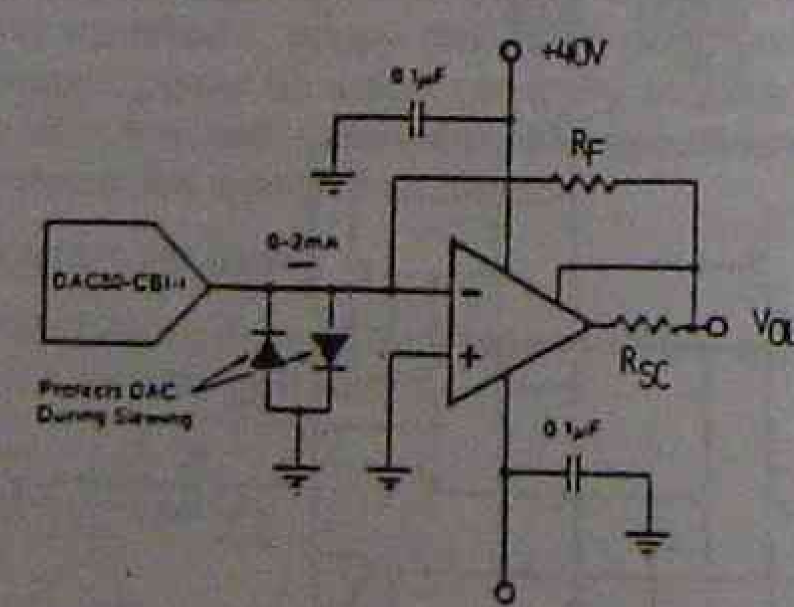


Fig.2

2. The circuit of Fig.2 shows a digitally controlled power supply. In this circuit, a binary number applied to the input of the DAC determines the current flowing into the output terminal of the DAC. If the output current range of the DAC is 0 to 2mA:

   (a) Calculate the value of R_F required to give an output voltage range of 0 to 30V.

   (b) Calculate the value of R_{SC} required to limit the current to 4A.

****

National Semiconductor Corporation

LM101A/LM201A/LM301A Operational Amplifiers

General Description

The LM101A series are general purpose operational amplifiers which feature improved performance over industry standards like the LM709. Advanced processing techniques make possible an order of magnitude reduction in input currents, and a redesign of the biasing circuitry reduces the temperature drift of input current. Improved specifications include:

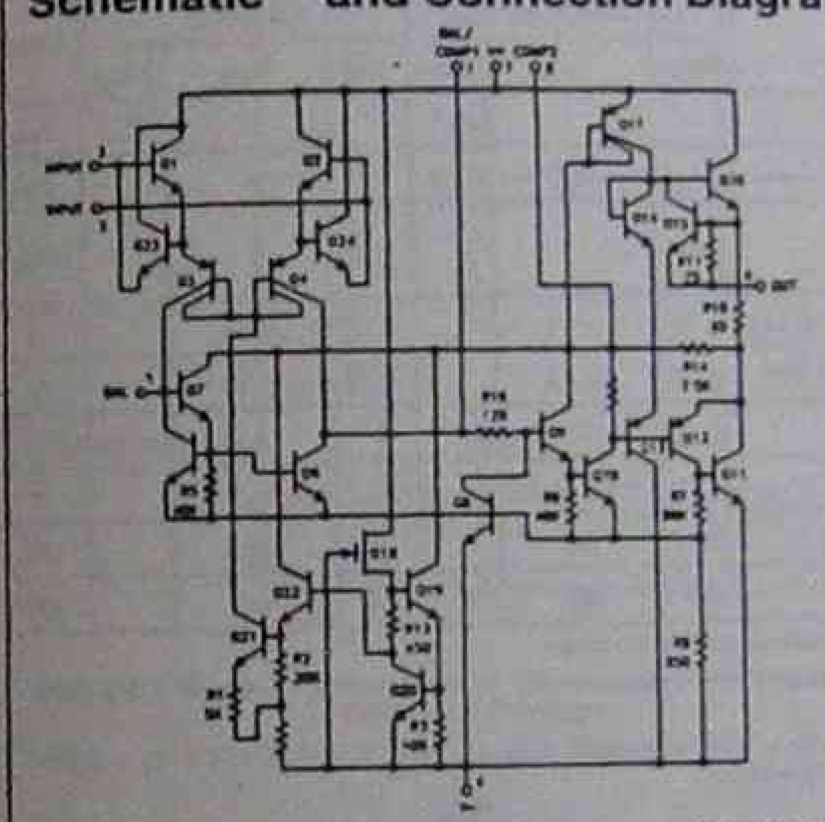
- Offset voltage 3 mV maximum over temperature (LM101A/LM201A)
- Input current 100 nA maximum over temperature (LM101A/LM201A)
- Offset current 20 nA maximum over temperature (LM101A/LM201A)
- Guaranteed drift characteristics
- Offsets guaranteed over entire common mode and supply voltage ranges
- Slew rate of 10V/µs as a summing amplifier

This amplifier offers many features which make its application nearly foolproof: overload protection on the input and output, no latch-up when the common mode range is exceeded, and freedom from oscillations and compensation with a single 30 pF capacitor. It has advantages over internally compensated amplifiers in that the frequency compensation can be tailored to the particular application. For example, in low frequency circuits it can be overcompensated for increased stability margin. Or the compensation can be optimized to give more than a factor of ten improvement in high frequency performance for most applications.

In addition, the device provides better accuracy and lower noise in high impedance circuitry. The low input currents also make it particularly well suited for long interval integrators or timers, sample and hold circuits and low frequency waveform generators. Further, replacing circuits where matched transistor pairs buffer the inputs of conventional IC op amps, it can give lower offset voltage and a drift at a lower cost.

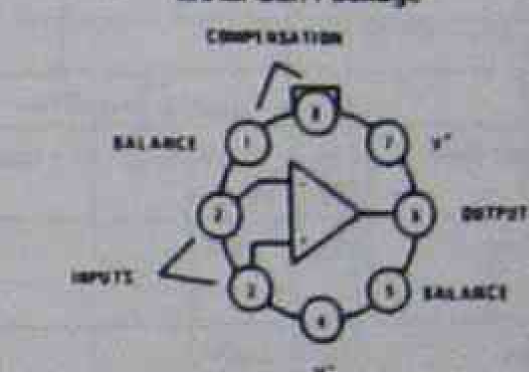
The LM101A is guaranteed over a temperature range of −55°C to +125°C, the LM201A from −25°C to +85°C, and the LM301A from 0°C to +70°C.

Schematic** and Connection Diagrams (Top View)



TL/H/7752-1

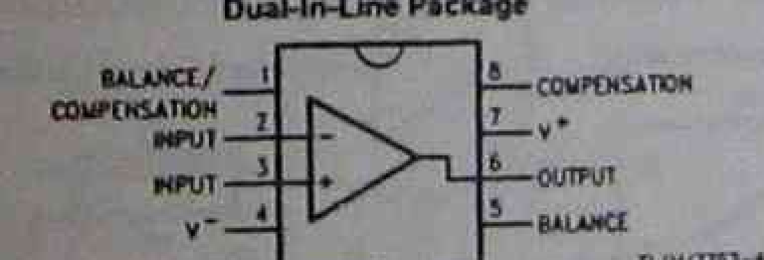
Metal Can Package



TL/H/7752-2

Order Number LM101AH, LM201AH or LM301AH
See NS Package Number H08C

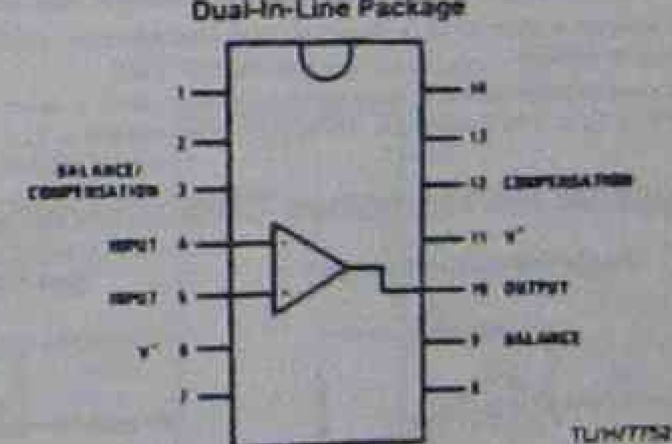
Dual-In-Line Package



TL/H/7752-4

Order Number LM101AJ, LM201AJ, LM301AJ, LM201AN or LM301AN
See NS Package Number J08A or N08A

Dual-In-Line Package



TL/H/7752-3

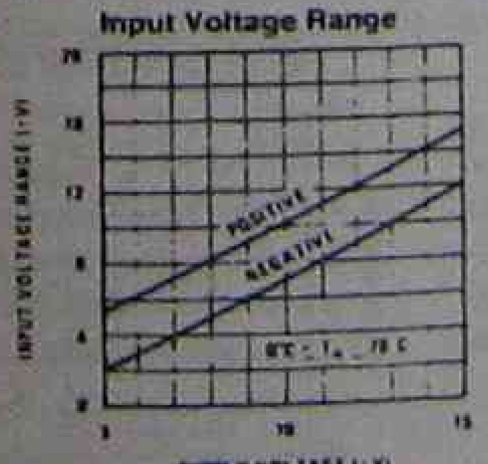
Note: Pin 6 connected to bottom of package.

Order Number LM101AJ-14A, LM201AJ-14 or LM301AJ-14
See NS Package Number J14A

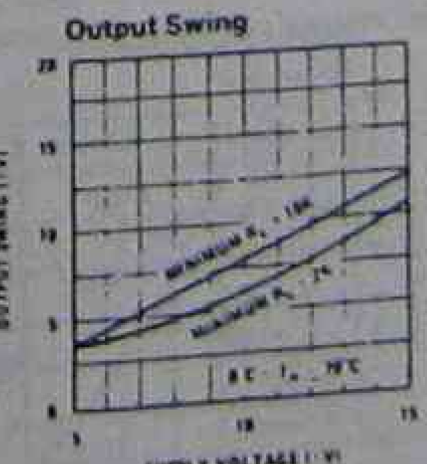
**Pin connections shown are for 8-pin packages.

Guaranteed Performance Characteristics LM301A

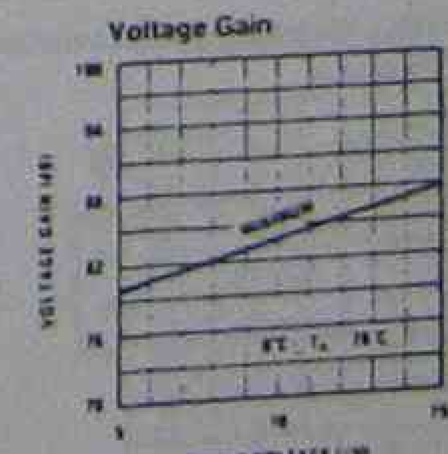
Input Voltage Range



Output Swing



Voltage Gain



Electrical Characteristics (Note 3) $T_A = T_J$

Parameter	Conditions		LM101A/LM201A Min	LM101A/LM201A Typ	LM101A/LM201A Max	LM301A Min	LM301A Typ	LM301A Max	Units
Input Offset Voltage	$T_A = 25°C, R_S \le 50\ k\Omega$			0.7	2.0		2.0	7.5	mV
Input Offset Current	$T_A = 25°C$			1.5	10		3.0	50	nA
Input Bias Current	$T_A = 25°C$			30	75		70	250	nA
Input Resistance	$T_A = 25°C$		1.5	4.0		0.5	2.0		MΩ
Supply Current	$T_A = 25°C$	$V_S = \pm 20V$		1.8	3.0				mA
		$V_S = \pm 15V$					1.8	3.0	mA
Large Signal Voltage Gain	$T_A = 25°C, V_S = \pm 15V$, $V_{OUT} = \pm 10V, R_L \ge 2\ k\Omega$		50	160		25	160		V/mV
Input Offset Voltage	$R_S \le 50\ k\Omega$				3.0			10	mV
Average Temperature Coefficient of Input Offset Voltage	$R_S \le 50\ k\Omega$			3.0	15		6.0	30	μV/°C
Input Offset Current					20			70	nA
Average Temperature Coefficient of Input Offset Current	$25°C \le T_A \le T_{MAX}$			0.01	0.1		0.01	0.3	nA/°C
	$T_{MIN} \le T_A \le 25°C$			0.02	0.2		0.02	0.6	nA/°C

Electrical Characteristics (Note 3) $T_A = T_J$ (Continued)

Parameter	Conditions		LM101A/LM201A Min	LM101A/LM201A Typ	LM101A/LM201A Max	LM301A Min	LM301A Typ	LM301A Max	Units
Input Bias Current					0.1			0.3	μA
Supply Current	$T_A = T_{MAX}, V_S = \pm 20V$			1.2	2.5				mA
Large Signal Voltage Gain	$V_S = \pm 15V, V_{OUT} = \pm 10V$, $R_L \ge 2k$		25			15			V/mV
Output Voltage Swing	$V_S = \pm 15V$	$R_L = 10\ k\Omega$	±12	±14		±12	±14		V
		$R_L = 2\ k\Omega$	±10	±13		±10	±13		V
Input Voltage Range	$V_S = \pm 20V$		±15						V
	$V_S = \pm 15V$			±15, −13		±12	+15, −13		V
Common-Mode Rejection Ratio	$R_S \le 50\ k\Omega$		80	96		70	90		dB
Supply Voltage Rejection Ratio	$R_S \le 50\ k\Omega$		80	96		70	96		dB

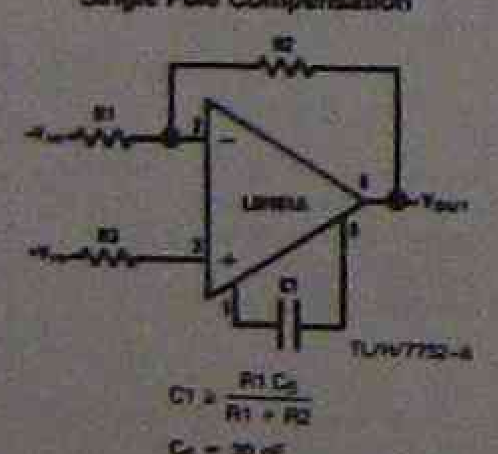
Note 1: For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.

Note 2: Continuous short circuit is allowed for case temperatures to 125°C and ambient temperatures to 75°C for LM101A/LM201A, and 70°C and 55°C respectively for LM301A.

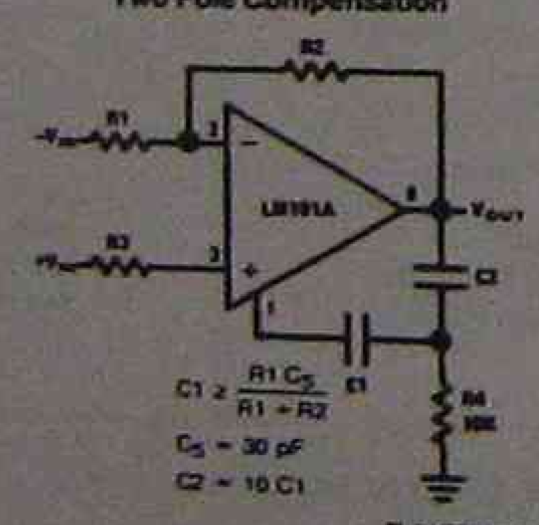
Note 3: Unless otherwise specified, these specifications apply for C1 = 30 pF, $\pm 5V \le V_S \le \pm 20V$ and $-55°C \le T_A \le +125°C$ (LM101A), $\pm 5V \le V_S \le \pm 20V$ and $-25°C \le T_A \le +85°C$ (LM201A), $\pm 5V \le V_S \le \pm 15V$ and $0°C \le T_A \le +70°C$ (LM301A).

Typical Performance Characteristics for Various Compensation Circuits**

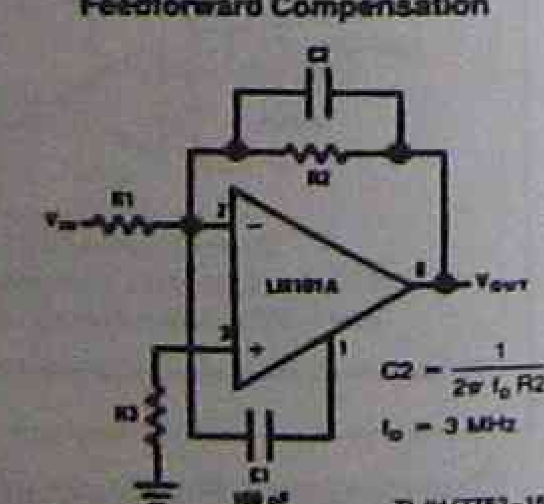
Single Pole Compensation



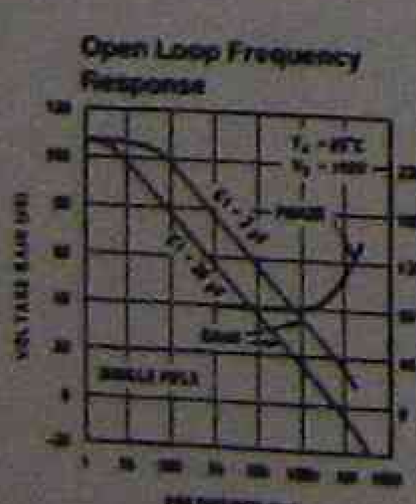
Two Pole Compensation



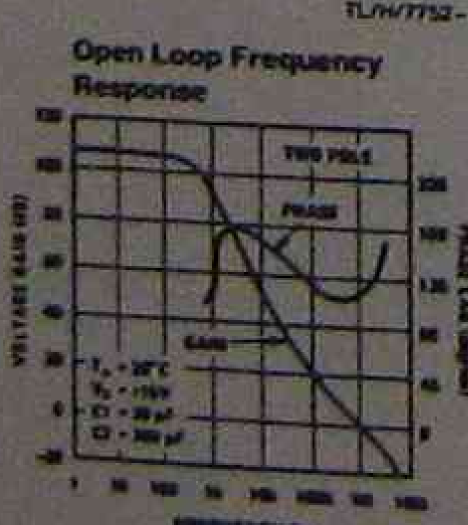
Feedforward Compensation



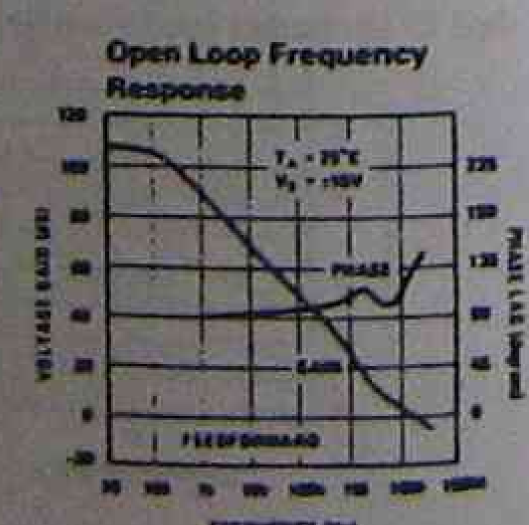
Open Loop Frequency Response



Open Loop Frequency Response



Open Loop Frequency Response



National Semiconductor Corporation

LM741/LM741A/LM741C/LM741E Operational Amplifier

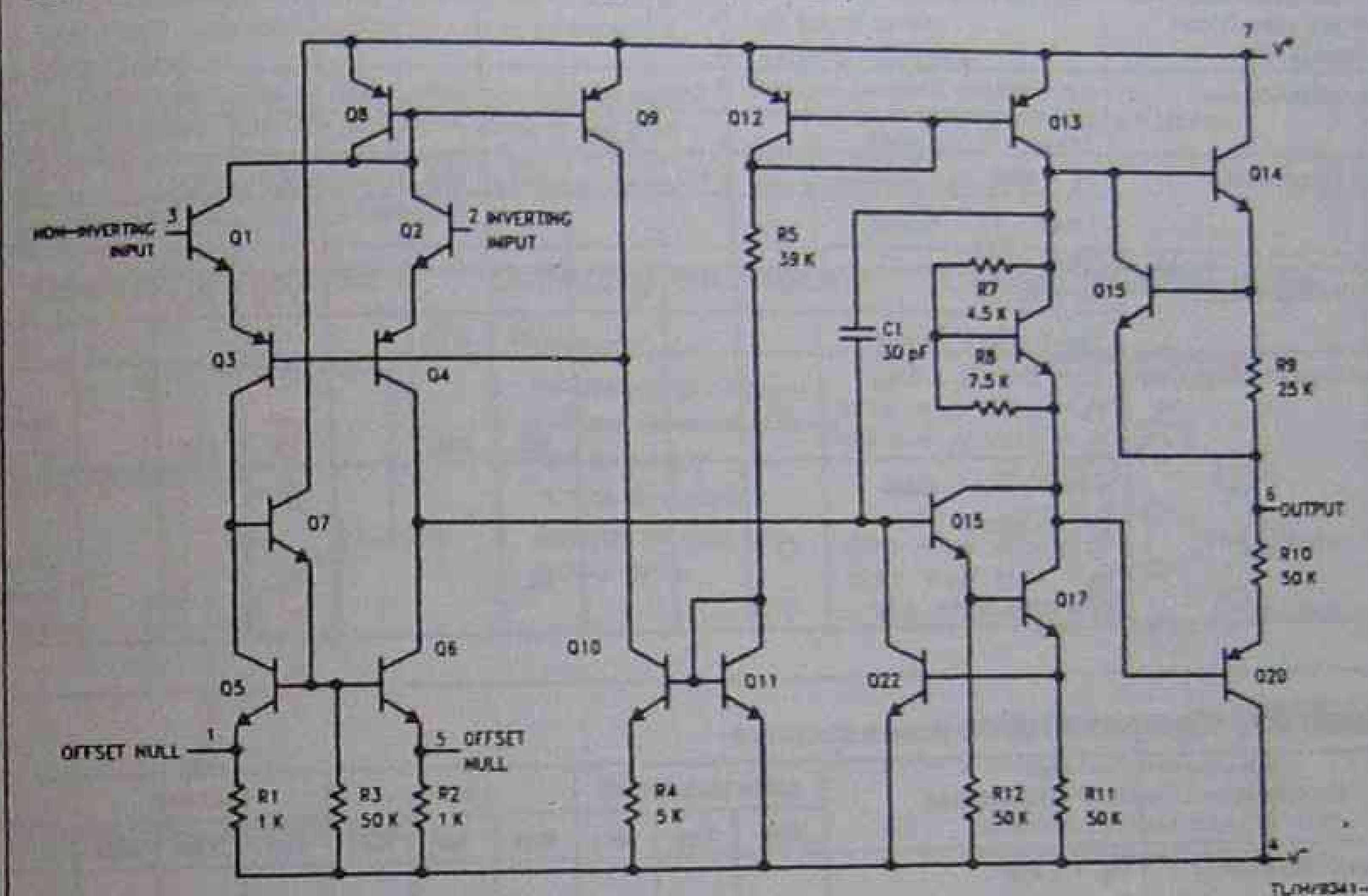
General Description

The LM741 series are general purpose operational amplifiers which feature improved performance over industry standards like the LM709. They are direct, plug-in replacements for the 709C, LM201, MC1439 and 748 in most applications.

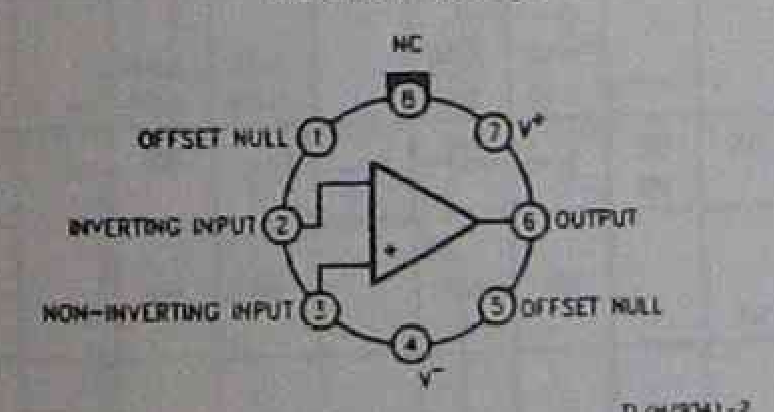
The amplifiers offer many features which make their application nearly foolproof: overload protection on the input and output, no latch-up when the common mode range is exceeded, as well as freedom from oscillations.

The LM741C/LM741E are identical to the LM741/LM741A except that the LM741C/LM741E have their performance guaranteed over a 0°C to +70°C temperature range, instead of −55°C to +125°C.

Schematic and Connection Diagrams (Top Views)

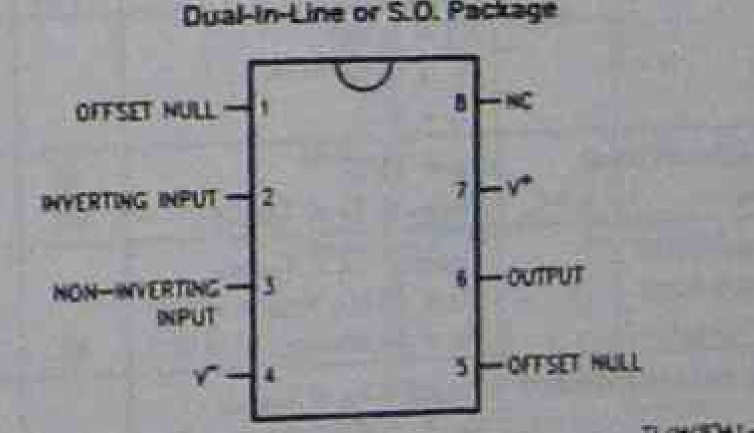


Metal Can Package



Order Number LM741H, LM741AH, LM741CH or LM741EH
See NS Package Number H08C

Dual-In-Line or S.O. Package



Order Number LM741CJ, LM741CM, LM741CN or LM741EN
See NS Package Number J08A, M08A or N08E

Absolute Maximum Ratings

If Military/Aerospace specified devices are required, contact the National Semiconductor Sales Office/Distributors for availability and specifications.
(Note 5)

	LM741A	LM741E	LM741	LM741C
Supply Voltage	±22V	±22V	±22V	±18V
Power Dissipation (Note 1)	500 mW	500 mW	500 mW	500 mW
Differential Input Voltage	±30V	±30V	±30V	±30V
Input Voltage (Note 2)	±15V	±15V	±15V	±15V
Output Short Circuit Duration	Indefinite	Indefinite	Indefinite	Indefinite

Electrical Characteristics (Note 3) LM741/LM741A/LM741C/LM741E

Parameter	Conditions	LM741A/LM741E Min	LM741A/LM741E Typ	LM741A/LM741E Max	LM741 Min	LM741 Typ	LM741 Max	LM741C Min	LM741C Typ	LM741C Max	Units
Input Offset Voltage	$T_A = 25°C$, $R_S \leq 10\ k\Omega$					1.0	5.0		2.0	6.0	mV
	$R_S \leq 50\Omega$		0.8	3.0							mV
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $R_S \leq 50\Omega$			4.0							mV
	$R_S \leq 10\ k\Omega$						6.0			7.5	mV
Average Input Offset Voltage Drift				15							µV/°C
Input Offset Voltage Adjustment Range	$T_A = 25°C$, $V_S = \pm 20V$	±10				±15			±15		mV
Input Offset Current	$T_A = 25°C$		3.0	30		20	200		20	200	nA
	$T_{AMIN} \leq T_A \leq T_{AMAX}$			70		85	500			300	nA
Average Input Offset Current Drift				0.5							nA/°C
Input Bias Current	$T_A = 25°C$		30	80		80	500		80	500	nA
	$T_{AMIN} \leq T_A \leq T_{AMAX}$			0.210			1.5			0.8	µA
Input Resistance	$T_A = 25°C$, $V_S = \pm 20V$	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $V_S = \pm 20V$	0.5									MΩ
Input Voltage Range	$T_A = 25°C$							±12	±13		V
	$T_{AMIN} \leq T_A \leq T_{AMAX}$				±12	±13					V
Large Signal Voltage Gain	$T_A = 25°C$, $R_L \geq 2\ k\Omega$, $V_S = \pm 20V$, $V_O = \pm 15V$	50									V/mV
	$V_S = \pm 15V$, $V_O = \pm 10V$				50	200		20	200		V/mV
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $R_L \geq 2\ k\Omega$, $V_S = \pm 20V$, $V_O = \pm 15V$	32									V/mV
	$V_S = \pm 15V$, $V_O = \pm 10V$				25			15			V/mV
	$V_S = \pm 5V$, $V_O = \pm 2V$	10									V/mV

Electrical Characteristics (Note 3) (Continued)

Parameter	Conditions	LM741A/LM741E Min	LM741A/LM741E Typ	LM741A/LM741E Max	LM741 Min	LM741 Typ	LM741 Max	LM741C Min	LM741C Typ	LM741C Max	Units
Output Voltage Swing	$V_S = \pm 20V$, $R_L \geq 10\ k\Omega$	±16									V
	$R_L \geq 2\ k\Omega$	±15									V
	$V_S = \pm 15V$, $R_L \geq 10\ k\Omega$				±12	±14		±12	±14		V
	$R_L \geq 2\ k\Omega$				±10	±13		±10	±13		V
Output Short Circuit Current	$T_A = 25°C$	10	25	35		25			25		mA
	$T_{AMIN} \leq T_A \leq T_{AMAX}$	10		40							mA
Common-Mode Rejection Ratio	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $R_S \leq 10\ k\Omega$, $V_{CM} = \pm 12V$				70	90		70	90		dB
	$R_S \leq 50\ k\Omega$, $V_{CM} = \pm 12V$	80	95								dB
Supply Voltage Rejection Ratio	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $V_S = \pm 20V$ to $V_S = \pm 5V$, $R_S \leq 50\Omega$	86	96								dB
	$R_S \leq 10\ k\Omega$				77	96		77	96		dB
Transient Response Rise Time	$T_A = 25°C$, Unity Gain		0.25	0.8		0.3			0.3		µs
Overshoot			6.0	20		5			5		%
Bandwidth (Note 4)	$T_A = 25°C$	0.437	1.5								MHz
Slew Rate	$T_A = 25°C$, Unity Gain	0.3	0.7			0.5			0.5		V/µs
Supply Current	$T_A = 25°C$					1.7	2.8		1.7	2.8	mA

Note 1: For operation at elevated temperatures, these devices must be derated based on thermal resistance, and T_j max. (listed under "Absolute Maximum Ratings"). $T_j = T_A + (\theta_{jA} P_D)$.

Thermal Resistance	Cerdip (J)	DIP (N)	TO-5 (H)	SO-8 (M)
θ_{jA} (Junction to Ambient)	100°C/W	100°C/W	150°C/W	195°C/W
θ_{jC} (Junction to Case)	N/A	N/A	80°C/W	N/A

Note 2: For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.

Note 3: Unless otherwise specified ...

National Semiconductor Corporation

LF411A/LF411 Low Offset, Low Drift JFET Input Operational Amplifier

General Description

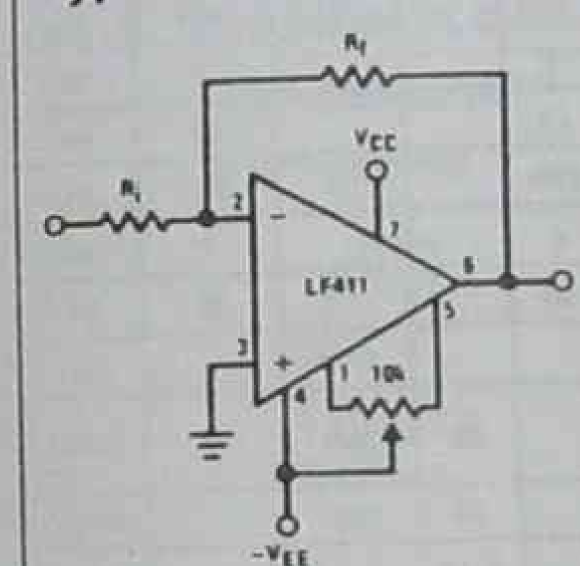
These devices are low cost, high speed, JFET input operational amplifiers with very low input offset voltage and guaranteed input offset voltage drift. They require low supply current yet maintain a large gain bandwidth product and fast slew rate. In addition, well matched high voltage JFET input devices provide very low input bias and offset currents. The LF411 is pin compatible with the standard LM741 allowing designers to immediately upgrade the overall performance of existing designs.

These amplifiers may be used in applications such as high speed integrators, fast D/A converters, sample and hold circuits and many other circuits requiring low input offset voltage and drift, low input bias current, high input impedance, high slew rate and wide bandwidth.

Features

- Internally trimmed offset voltage — 0.5 mV(max)
- Input offset voltage drift — 10 µV/°C(max)
- Low input bias current — 50 pA
- Low input noise current — 0.01 pA/$\sqrt{Hz}$
- Wide gain bandwidth — 3 MHz(min)
- High slew rate — 10V/µs(min)
- Low supply current — 1.8 mA
- High input impedance — $10^{12}\Omega$
- Low total harmonic distortion $A_V = 10$, $R_L = 10k$, $V_O = 20$ Vp-p, BW = 20 Hz – 20 kHz — <0.02%
- Low 1/f noise corner — 50 Hz
- Fast settling time to 0.01% — 2 µs

Typical Connection



TL/H/5655–1

Ordering Information

LF411XYZ

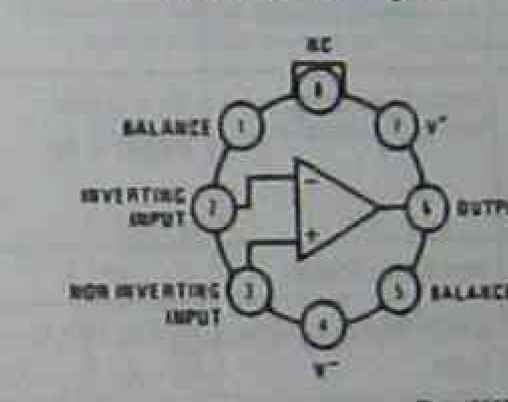
X indicates electrical grade

Y indicates temperature range
"M" for military
"C" for commercial

Z indicates package type
"H" or "N"

Connection Diagrams

Metal Can Package



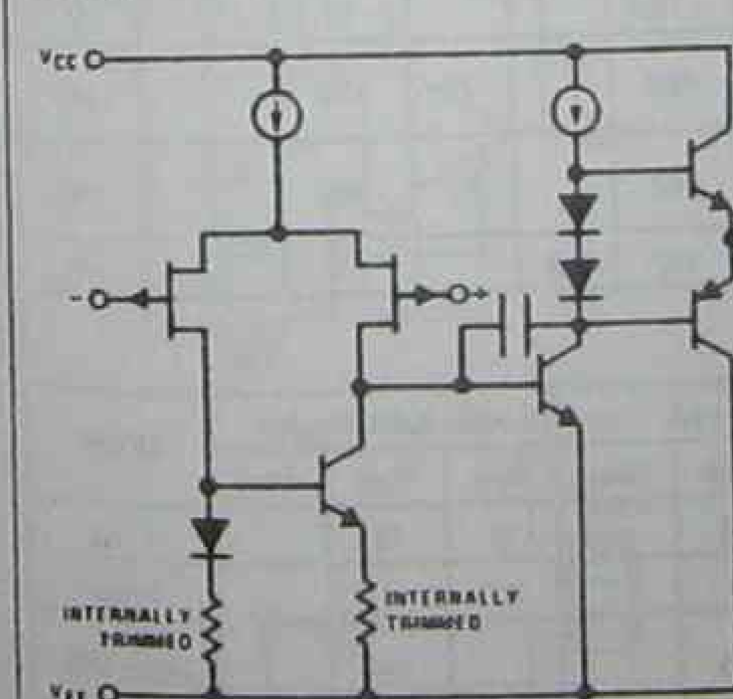
TL/H/5655–5

Top View

Note: Pin 4 connected to case

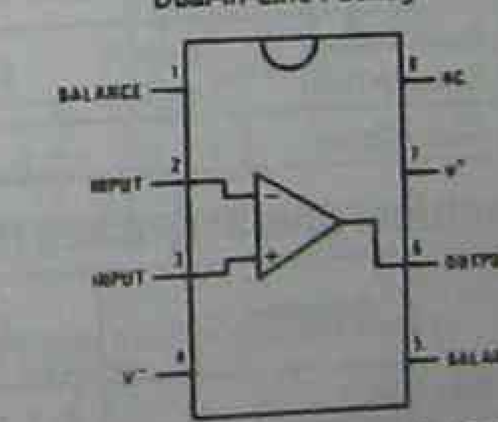
Order Number LF411AMH, LF411MH, LF411ACH or LF411CH
See NS Package Number H08B

Simplified Schematic



TL/H/5655–6

Dual-In-Line Package



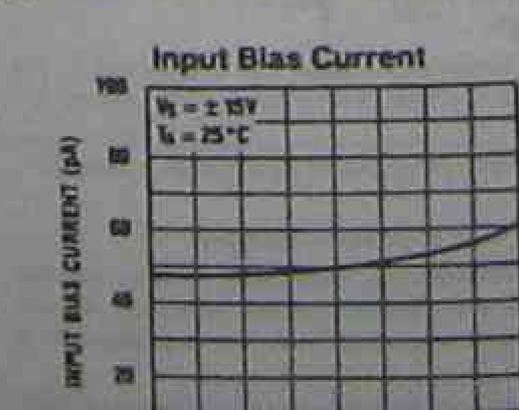
TL/H/5655–7

Top View

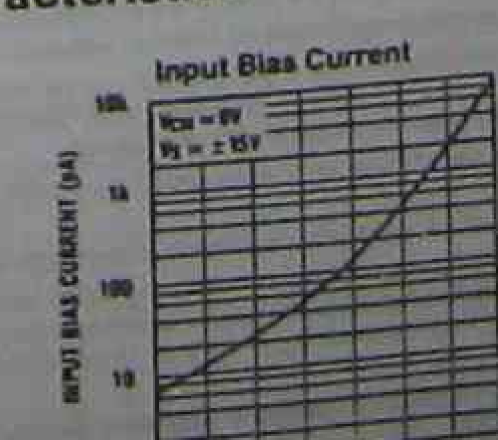
Order Number LF411ACN or LF411CN
See NS Package Number N08E

Typical Performance Characteristics

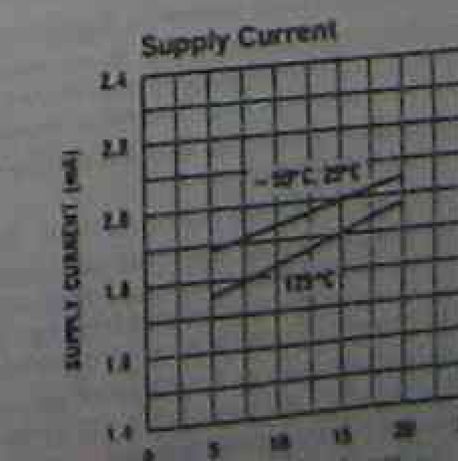
Input Bias Current



Input Bias Current



Supply Current



LM741/LM741A/LM741C/LM741E

Electrical Characteristics (Note 3)

Parameter	Conditions	LM741A/LM741E Min	LM741A/LM741E Typ	LM741A/LM741E Max	LM741 Min	LM741 Typ	LM741 Max	LM741C Min	LM741C Typ	LM741C Max	Units
Input Offset Voltage	$T_A = 25°C$, $R_S \le 10 k\Omega$					1.0	5.0		2.0	6.0	mV
	$R_S \le 50\Omega$		0.8	3.0							mV
	$T_{AMIN} \le T_A \le T_{AMAX}$, $R_S \le 50\Omega$			4.0							mV
	$R_S \le 10 k\Omega$						6.0			7.5	mV
Average Input Offset Voltage Drift				15							μV/°C
Input Offset Voltage Adjustment Range	$T_A = 25°C$, $V_S = \pm 20V$	±10				±15			±15		mV
Input Offset Current	$T_A = 25°C$		3.0	30		20	200		20	200	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			70		85	500			300	nA
Average Input Offset Current Drift				0.5							nA/°C
Input Bias Current	$T_A = 25°C$		30	80		80	500		80	500	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			0.210			1.5			0.8	μA
Input Resistance	$T_A = 25°C$, $V_S = \pm 20V$	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \le T_A \le T_{AMAX}$, $V_S = \pm 20V$	0.5									MΩ
Input Voltage Range	$T_A = 25°C$							±12	±13		V
	$T_{AMIN} \le T_A \le T_{AMAX}$				±12	±13					V
Large Signal Voltage Gain	$T_A = 25°C$, $R_L \ge 2 k\Omega$, $V_S = \pm 20V$, $V_O = \pm 15V$	50									V/mV
	$V_S = \pm 15V$, $V_O = \pm 10V$				50	200		20	200		V/mV
	$T_{AMIN} \le T_A \le T_{AMAX}$, $R_L \ge 2 k\Omega$, $V_S = \pm 20V$, $V_O = \pm 15V$	32									V/mV
	$V_S = \pm 15V$, $V_O = \pm 10V$				25			15			V/mV
	$V_S = \pm 5V$, $V_O = \pm 2V$	10									V/mV

Electrical Characteristics (Note 3) (Continued)

Parameter	Conditions	LM741A/LM741E Min	LM741A/LM741E Typ	LM741A/LM741E Max	LM741 Min	LM741 Typ	LM741 Max	LM741C Min	LM741C Typ	LM741C Max	Units
Output Voltage Swing	$V_S = \pm 20V$, $R_L \ge 10 k\Omega$	±16									V
	$R_L \ge 2 k\Omega$	±15									V
	$V_S = \pm 15V$, $R_L \ge 10 k\Omega$				±12	±14		±12	±14		V
	$R_L \ge 2 k\Omega$				±10	±13		±10	±13		V
Output Short Circuit Current	$T_A = 25°C$	10	25	35		25			25		mA
	$T_{AMIN} \le T_A \le T_{AMAX}$	10		40							mA
Common-Mode Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$, $R_S \le 10 k\Omega$, $V_{CM} = \pm 12V$				70	90		70	90		dB
	$R_S \le 50 k\Omega$, $V_{CM} = \pm 12V$	80	95								dB
Supply Voltage Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$, $V_S = \pm 20V$ to $V_S = \pm 5V$, $R_S \le 50\Omega$	86	96								dB
	$R_S \le 10 k\Omega$				77	96		77	96		dB
Transient Response Rise Time	$T_A = 25°C$, Unity Gain		0.25	0.8		0.3			0.3		μs
Overshoot			6.0	20		5			5		%
Bandwidth (Note 4)	$T_A = 25°C$	0.437	1.5								MHz
Slew Rate	$T_A = 25°C$, Unity Gain	0.3	0.7			0.5			0.5		V/μs
Supply Current	$T_A = 25°C$					1.7	2.8		1.7	2.8	mA

Note 1: For operation at elevated temperatures, these devices must be derated based on thermal resistance, and T_j max. (listed under "Absolute Maximum Ratings"). $T_j = T_A + (\theta_{jA} P_D)$.

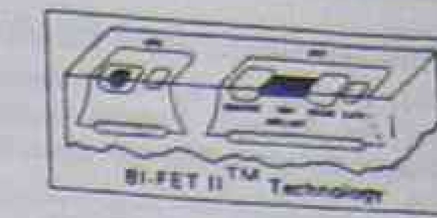
Thermal Resistance	Cerdip (J)	DIP (N)	TO-5 (H)	SO-8 (M)
θ_{jA} (Junction to Ambient)	100°C/W	100°C/W	150°C/W	195°C/W
θ_{jC} (Junction to Case)	N/A	N/A	80°C/W	N/A

Note 2: For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.

Note 3: Unless otherwise specified, these specifications apply for $V_S = \pm 15V$, $-55°C \le T_A \le +125°C$ (LM741/LM741A). For the LM741C/LM741E, these specifications are limited to $0°C \le T_A \le +70°C$.

Note 4: Calculated value from: BW (MHz) = 0.35/Rise Time(μs).

National Semiconductor Corporation



LF411A/LF411 Low Offset, Low Drift JFET Input Operational Amplifier

General Description

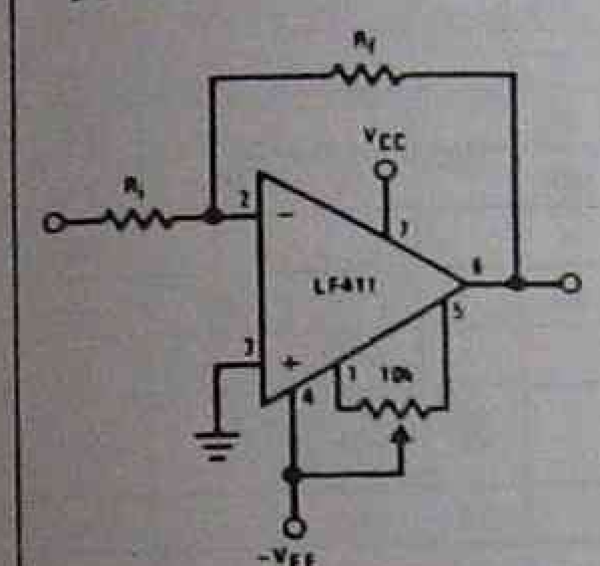
These devices are low cost, high speed, JFET input operational amplifiers with very low input offset voltage and guaranteed input offset voltage drift. They require low supply current yet maintain a large gain bandwidth product and fast slew rate. In addition, well matched high voltage JFET input devices provide very low input bias and offset currents. The LF411 is pin compatible with the standard LM741 allowing designers to immediately upgrade the overall performance of existing designs.

These amplifiers may be used in applications such as high speed integrators, fast D/A converters, sample and hold circuits and many other circuits requiring low input offset voltage and drift, low input bias current, high input impedance, high slew rate and wide bandwidth.

Features

- Internally trimmed offset voltage — 0.5 mV(max)
- Input offset voltage drift — 10 μV/°C(max)
- Low input bias current — 50 pA
- Low input noise current — 0.01 pA/$\sqrt{Hz}$
- Wide gain bandwidth — 3 MHz(min)
- High slew rate — 10V/μs(min)
- Low supply current — 1.8 mA
- High input impedance — $10^{12}\Omega$
- Low total harmonic distortion $A_V = 10$, $R_L = 10k$, $V_O = 20$ Vp-p, BW = 20 Hz−20 kHz — <0.02%
- Low 1/f noise corner — 50 Hz
- Fast settling time to 0.01% — 2 μs

Typical Connection



TL/H/5655–1

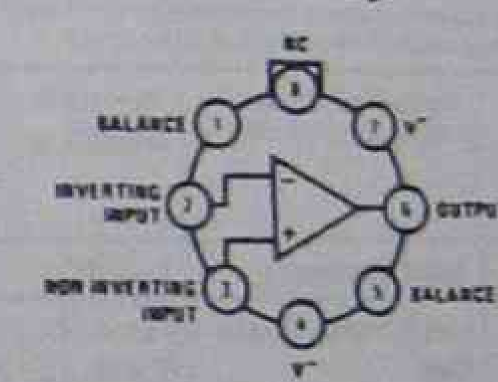
Ordering Information

LF411XYZ

X indicates electrical grade
Y indicates temperature range
"M" for military
"C" for commercial
Z indicates package type
"H" or "N"

Connection Diagrams

Metal Can Package



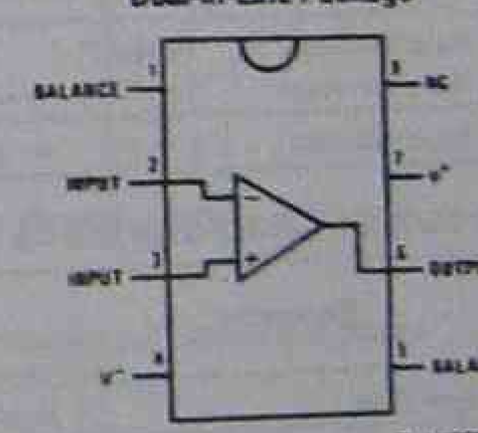
TL/H/5655–5

Top View

Note: Pin 4 connected to case.

Order Number LF411AMH, LF411MH, LF411ACH or LF411CH
See NS Package Number H08B

Dual-In-Line Package

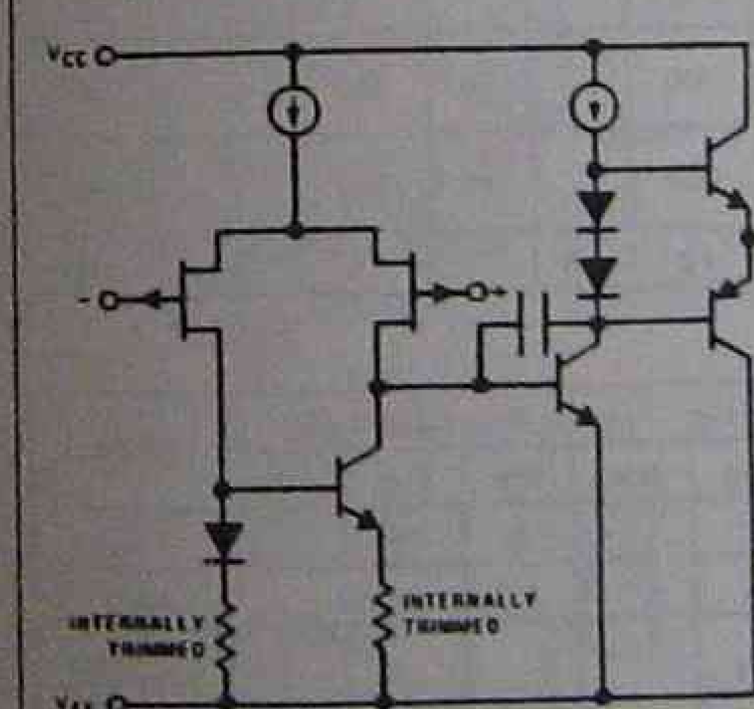


TL/H/5655–7

Top View

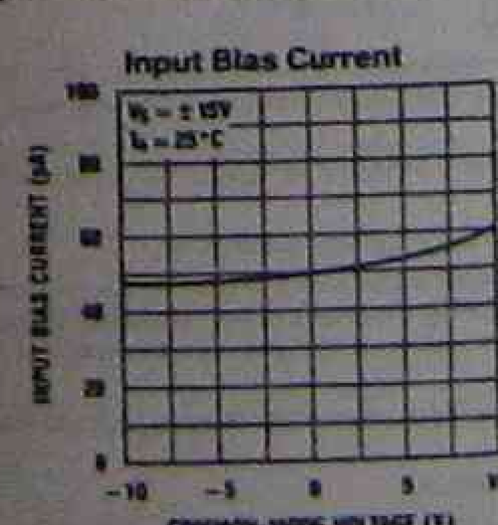
Order Number LF411ACN or LF411CN
See NS Package Number N08E

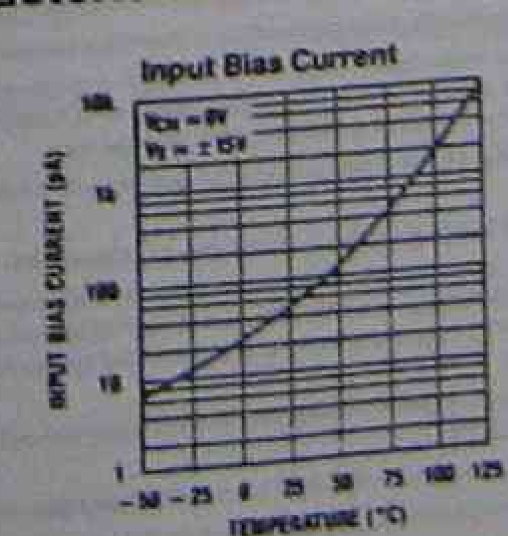
Simplified Schematic

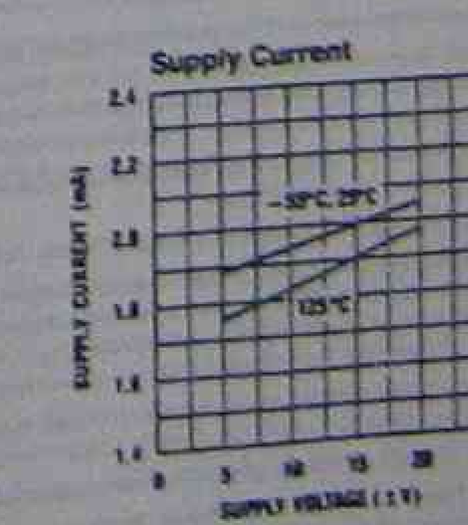


TL/H/5655–6

Typical Performance Characteristics







Absolute Maximum Ratings

If Military/Aerospace specified devices are required, contact the National Semiconductor Sales Office/Distributors for availability and specifications. (Note 8)

	LF411A	LF411
Supply Voltage	±22V	±18V
Differential Input Voltage	±38V	±30V
Input Voltage Range (Note 1)	±19V	±15V
Output Short Circuit Duration	Continuous	Continuous

LF411A/LF411	H Package	N Package
Power Dissipation (Notes 2 and 9)	670 mW	670 mW
$T_j max$	150°C	115°C
θ_jA	225°C/W (Still Air) 160°C/W (400 LF/min Air Flow)	120°C/W
θ_jC	25°C/W	
Operating Temp. Range	(Note 3)	(Note 3)
Storage Temp. Range	$-65°C \le T_A \le 150°C$	$-65°C \le T_A \le 150°C$
Lead Temp. (Soldering, 10 sec.)	260°C	260°C

ESD rating to be determined.

DC Electrical Characteristics (Note 4)

Symbol	Parameter	Conditions		LF411A Min	LF411A Typ	LF411A Max	LF411 Min	LF411 Typ	LF411 Max	Units
V_{OS}	Input Offset Voltage	$R_S = 10\ k\Omega$, $T_A = 25°C$			0.3	0.5		0.8	2.0	mV
$\Delta V_{OS}/\Delta T$	Average TC of Input Offset Voltage	$R_S = 10\ k\Omega$ (Note 5)			7	10		7	20 (Note 5)	μV/°C
I_{OS}	Input Offset Current	$V_S = \pm 15V$ (Notes 4, 6)	$T_j = 25°C$		25	100		25	100	pA
			$T_j = 70°C$			2			2	nA
			$T_j = 125°C$			25			25	nA
I_B	Input Bias Current	$V_S = \pm 15V$ (Notes 4, 6)	$T_j = 25°C$		50	200		50	200	pA
			$T_j = 70°C$			4			4	nA
			$T_j = 125°C$			50			50	nA
R_{IN}	Input Resistance	$T_j = 25°C$			10^{12}			10^{12}		Ω
A_{VOL}	Large Signal Voltage Gain	$V_S = \pm 15V$, $V_O = \pm 10V$, $R_L = 2k$, $T_A = 25°C$		50	200		25	200		V/mV
		Over Temperature		25	200		15	200		V/mV
V_O	Output Voltage Swing	$V_S = \pm 15V$, $R_L = 10k$		±12	±13.5		±12	±13.5		V
V_{CM}	Input Common-Mode Voltage Range			±16	+19.5		±11	+14.5		V
					−16.5			−11.5		V
CMRR	Common-Mode Rejection Ratio	$R_S \le 10k$		80	100		70	100		dB
PSRR	Supply Voltage Rejection Ratio	(Note 7)		80	100		70	100		dB
I_S	Supply Current				1.8	2.8		1.8	3.4	mA

AC Electrical Characteristics (Note 4)

Symbol	Parameter	Conditions	LF411A Min	LF411A Typ	LF411A Max	LF411 Min	LF411 Typ	LF411 Max	Units
SR	Slew Rate	$V_S = \pm 15V$, $T_A = 25°C$	10	15		8	15		V/μs
GBW	Gain-Bandwidth Product	$V_S = \pm 15V$, $T_A = 25°C$	3	4		2.7	4		MHz
e_n	Equivalent Input Noise Voltage	$T_A = 25°C$, $R_S = 100\Omega$, $f = 1\ kHz$		25			25		$nV/\sqrt{Hz}$
i_n	Equivalent Input Noise Current	$T_A = 25°C$, $f = 1\ kHz$		0.01			0.01		$pA/\sqrt{Hz}$

Note 1: Unless otherwise specified the absolute maximum negative input voltage is equal to the negative power supply voltage.

Note 2: For operating at elevated temperature, these devices must be derated based on a thermal resistance of θ_jA.

Note 3: These devices are available in both the commercial temperature range $0°C \le T_A \le 70°C$ and the military temperature range $-55°C \le T_A \le 125°C$. The temperature range is designated by the position just before the package type in the device number. A "C" indicates the commercial temperature range and an "M" indicates the military temperature range. The military temperature range is available in "H" package only.

Note 4: Unless otherwise specified, the specifications apply over the full temperature range and for $V_S = \pm 20V$ for the LF411A and for $V_S = \pm 15V$ for the LF411. V_{OS}, I_B, and I_{OS} are measured at $V_{CM} = 0$.

Note 5: The LF411A is 100% tested to this specification. The LF411 is sample tested to insure at least 80% of the units meet this specification.

Note 6: The input bias currents are junction leakage currents which approximately double for every 10°C increase in the junction temperature, T_j. Due to limited production test time, the input bias currents measured are correlated to junction temperature. In normal operation the junction temperature rises above the ambient temperature as a result of internal power dissipation, P_D. $T_j = T_A + \theta_{jA} P_D$ where θ_{jA} is the thermal resistance from junction to ambient. Use of a heat sink is recommended if input bias current is to be kept to a minimum.

Note 7: Supply voltage rejection ratio is measured for both supply magnitudes increasing or decreasing simultaneously in accordance with common practice, from ±15V to ±5V for the LF411 and from ±20V to ±5V for the LF411A.

Note 8: Refer to RETS 411AX for LF411AMH military specifications and to RETS 411X for LF411MH military specifications.

Note 9: Max. Power Dissipation is defined by the package characteristics. Operating the part near the Max. Power Dissipation may cause the part to operate outside guaranteed limits.

National Semiconductor Corporation

LM111/LM211/LM311 Voltage Comparator

General Description

The LM111, LM211 and LM311 are voltage comparators that have input currents nearly a thousand times lower than devices like the LM106 or LM710. They are also designed to operate over a wider range of supply voltages: from standard ±15V op amp supplies down to the single 5V supply used for IC logic. Their output is compatible with RTL, DTL and TTL as well as MOS circuits. Further, they can drive lamps or relays, switching voltages up to 50V at currents as high as 50 mA.

Both the inputs and the outputs of the LM111, LM211 or the LM311 can be isolated from system ground, and the output can drive loads referred to ground, the positive supply or the negative supply. Offset balancing and strobe capability are provided and outputs can be wire OR'ed. Although slower than the LM106 and LM710 (200 ns response time vs 40 ns) the devices are also much less prone to spurious oscillations. The LM111 has the same pin configuration as the LM106 and LM710.

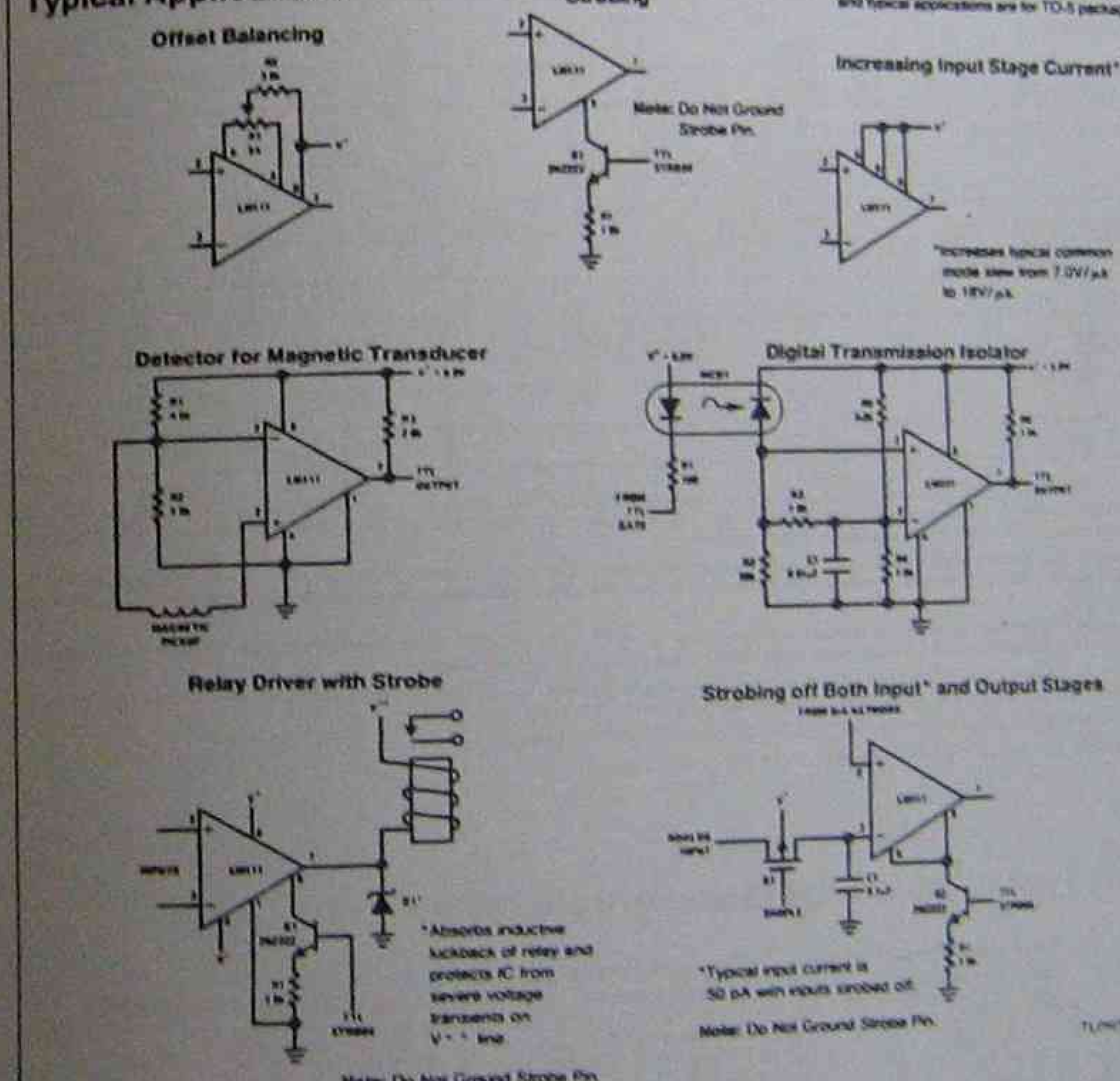
The LM211 is identical to the LM111, except that its performance is specified over a −25°C to +85°C temperature range instead of −55°C to +125°C. The LM311 has a temperature range of 0°C to +70°C.

Features

- Operates from single 5V supply
- Input current: 150 nA max. over temperature
- Offset current: 20 nA max. over temperature
- Differential input voltage range: ±30V
- Power consumption: 135 mW at ±15V

Typical Applications**

**Note: Pin connections shown on schematic diagram and typical applications are for TO-5 package.



Absolute Maximum Ratings for the LM311

If Military/Aerospace specified devices are required, contact the National Semiconductor Sales Office/Distributors for availability and specifications.

Total Supply Voltage (V_{84})	36V
Output to Negative Supply Voltage (V_{74})	40V
Ground to Negative Supply Voltage (V_{14})	30V
Differential Input Voltage	±30V
Input Voltage (Note 1)	±15V
Power Dissipation (Note 2)	500 mW
Output Short Circuit Duration	10 sec
Operating Temperature Range	0° to 70°C
Storage Temperature Range	−65°C to 150°C
Lead Temperature (soldering, 10 sec)	260°C
Voltage at Strobe Pin	V^+ −5V

Soldering Information
- Dual-In-Line Package
  - Soldering (10 seconds) .... 260°C
- Small Outline Package
  - Vapor Phase (60 seconds) .... 215°C
  - Infrared (15 seconds) .... 220°C

See AN-450 "Surface Mounting Methods and Their Effect on Product Reliability" for other methods of soldering surface mount devices.

Electrical Characteristics for the LM311 (Note 3)

Parameter	Conditions	Min	Typ	Max	Units
Input Offset Voltage (Note 4)	$T_A = 25°C$, $R_S \le 50k$		2.0	7.5	mV
Input Offset Current (Note 4)	$T_A = 25°C$		6.0	50	nA
Input Bias Current	$T_A = 25°C$		100	250	nA
Voltage Gain	$T_A = 25°C$	40	200		V/mV
Response Time (Note 5)	$T_A = 25°C$		200		ns
Saturation Voltage	$V_{IN} \le -10$ mV, $I_{OUT} = 50$ mA $T_A = 25°C$		0.75	1.5	V
Strobe ON Current	$T_A = 25°C$	1.5	3.0		mA
Output Leakage Current	$V_{IN} \ge 10$ mV, $V_{OUT} = 35V$ $T_A = 25°C$, $I_{STROBE} = 3$ mA $V^- = V_{GRND} = -5V$		0.2	50	nA
Input Offset Voltage (Note 4)	$R_S \le 50K$			10	mV
Input Offset Current (Note 4)				70	nA
Input Bias Current				300	nA
Input Voltage Range		−14.5	13.8, −14.7	13.0	V
Saturation Voltage	$V^+ \ge 4.5V$, $V^- = 0$ $V_{IN} \le -10$ mV, $I_{SINK} \le 8$ mA		0.23	0.4	V
Positive Supply Current	$T_A = 25°C$		5.1	7.5	mA
Negative Supply Current	$T_A = 25°C$		4.1	5.0	mA

Note 1: This rating applies for ±15V supplies. The positive input voltage limit is 30V above the negative supply. The negative input voltage limit is equal to the negative supply voltage or 30V below the positive supply, whichever is less.

Note 2: The maximum junction temperature of the LM311 is 110°C. For operating at elevated temperature, devices in the TO-5 package must be derated based on a thermal resistance of 150°C/W, junction to ambient, or 45°C/W, junction to case. The thermal resistance of the dual-in-line package is 100°C/W, junction to ambient.

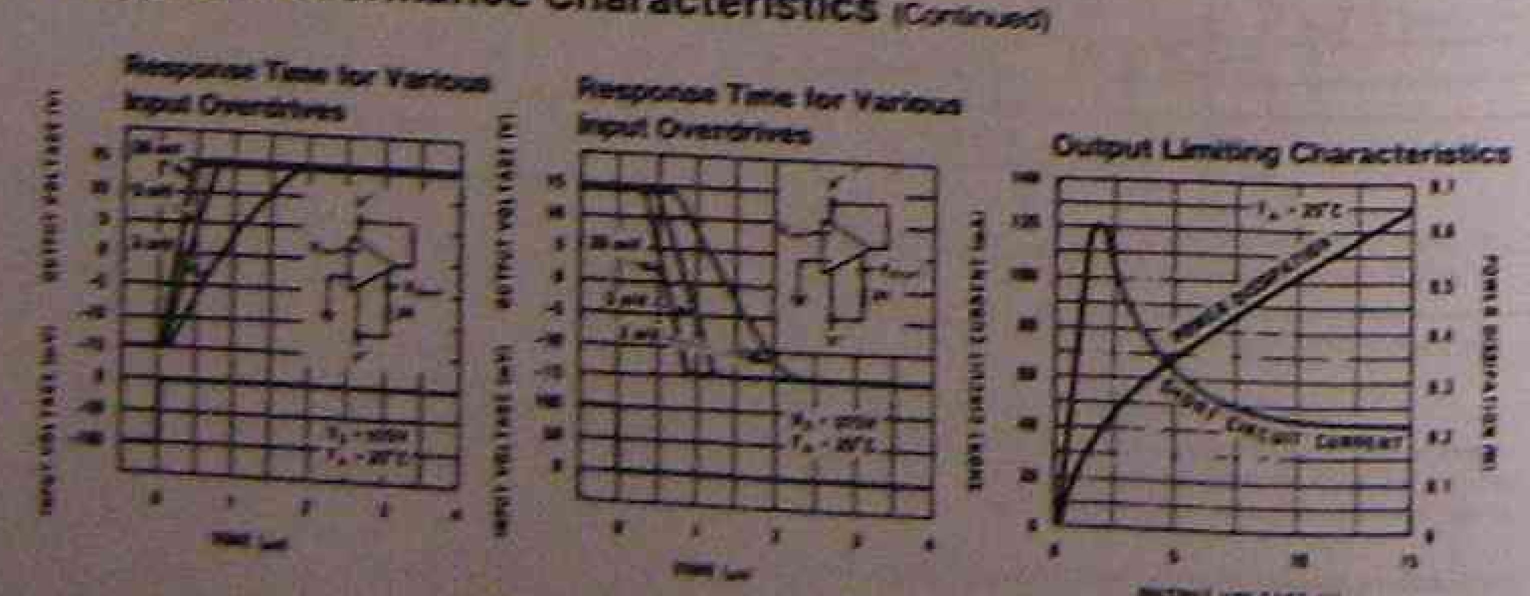
Note 3: These specifications apply for $V_S = \pm 15V$ and the Ground pin at ground, and 0°C < T_A < +70°C, unless otherwise specified. The offset voltage, offset current and bias current specifications apply for any supply voltage from a single 5V supply up to ±15V supplies.

Note 4: The offset voltages and offset currents given are the maximum values required to drive the output within a volt of either supply with 1 mA load. Thus, these parameters define an error band and take into account the worst-case effects of voltage gain and input impedance.

Note 5: The response time specified (see definitions) is for a 100 mV input step with 5 mV overdrive.

Note 6: Do not short the strobe pin to ground; it should be current driven at 3 to 5 mA.

LM311 Typical Performance Characteristics (Continued)

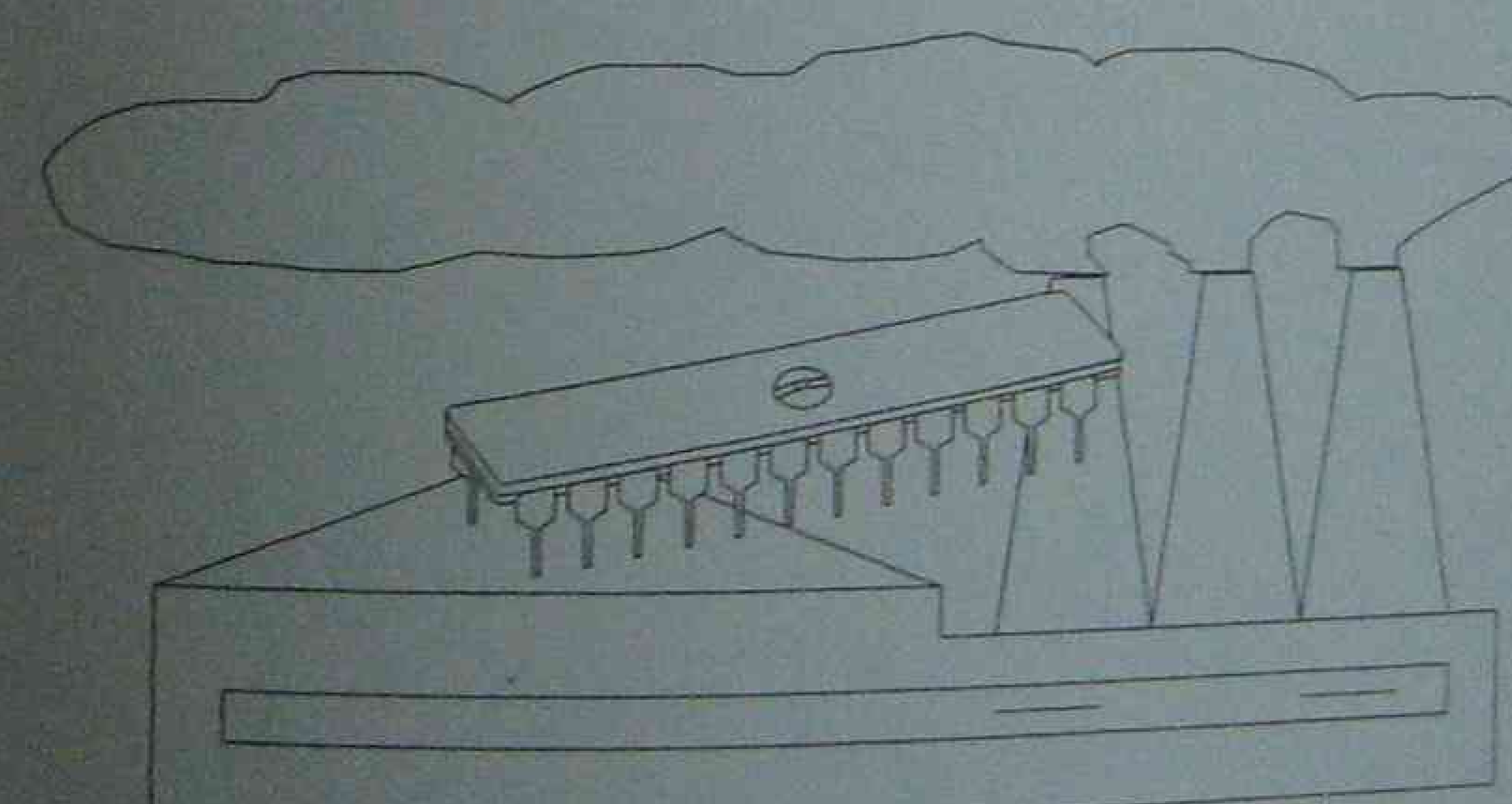


ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

OPERATIONAL AMPLIFIERS

SUBJECT NO. 6016C

PRACTICAL MANUAL



ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS 6016

OPERATIONAL AMPLIFIERS 6016C

PRACTICAL MANUAL

by

Graeme Odgers and Barry Maloney

ACKNOWLEDGEMENTS

The assignments in this manual have been written by Graeme Odgers and Barry Maloney. Editing and layout has been done by Peter Phillips. The assignments have all been trialled and corrections made where appropriate. However, despite the best of efforts, some errors may still remain. We therefore apologise for these in advance.

TO THE STUDENT

The practical assignments in this manual rely in some cases on information contained in the companion theory notes for this subject. As well, some of the questions that follow each assignment may require research from manufacturers' data books or from a text book. To assist you, this manual is arranged so that all data and waveforms can be recorded in the spaces provided. If there is insufficient space, attach additional pages to the assignment. The two data sheets in this manual provide a summary of typical op amps and are included for your reference.

TO THE TEACHER

Most of the practical assignments in this manual can be constructed on a breadboard. However, assignment 15 requires a dedicated PCB and assignments 1 and 2 are best done using a dedicated panel. Also, assignment 16 may require a dedicated panel. These assignments are identical to those originally distributed, although corrections have been made and the order rearranged to suit the revised theory manual. The emphasis may vary between teaching centres and locally relevant material can be included. Each practical session should occupy around 2 hours. Manual produced 1991.

CONTENTS

WEEK	TITLE	PAGE
1	The Differential Amplifier	1
2	Common Mode Rejection	6
3	Comparators	11
4	Timers	16
5	Operational Amplifier Circuits - 1	22
6	====== *Revision & Class Test* ======	
7	Operational Amplifier Circuits - 2	27
8	Operational Amplifier Metering Circuits	31
9	Operational Amplifier Characteristics	36
10	Operational Amplifier Diode Circuits	40
11	Sine and Square Wave Oscillators	45
12	====== *Revision and Class Test* ======	
13	Operational Amplifier RC Circuits	50
14	Active Filters	55
15	Multistage Operational Amplifier Circuits	59
16	Power Operational Amplifiers	63
17	====== *Revision and Class Test* ======	
18	---------- ***Final Examination*** ----------	
	Data Sheets: (Reprinted with the kind permission of Dick Smith Electronics and Electronics Australia	68

oooo0000oooo

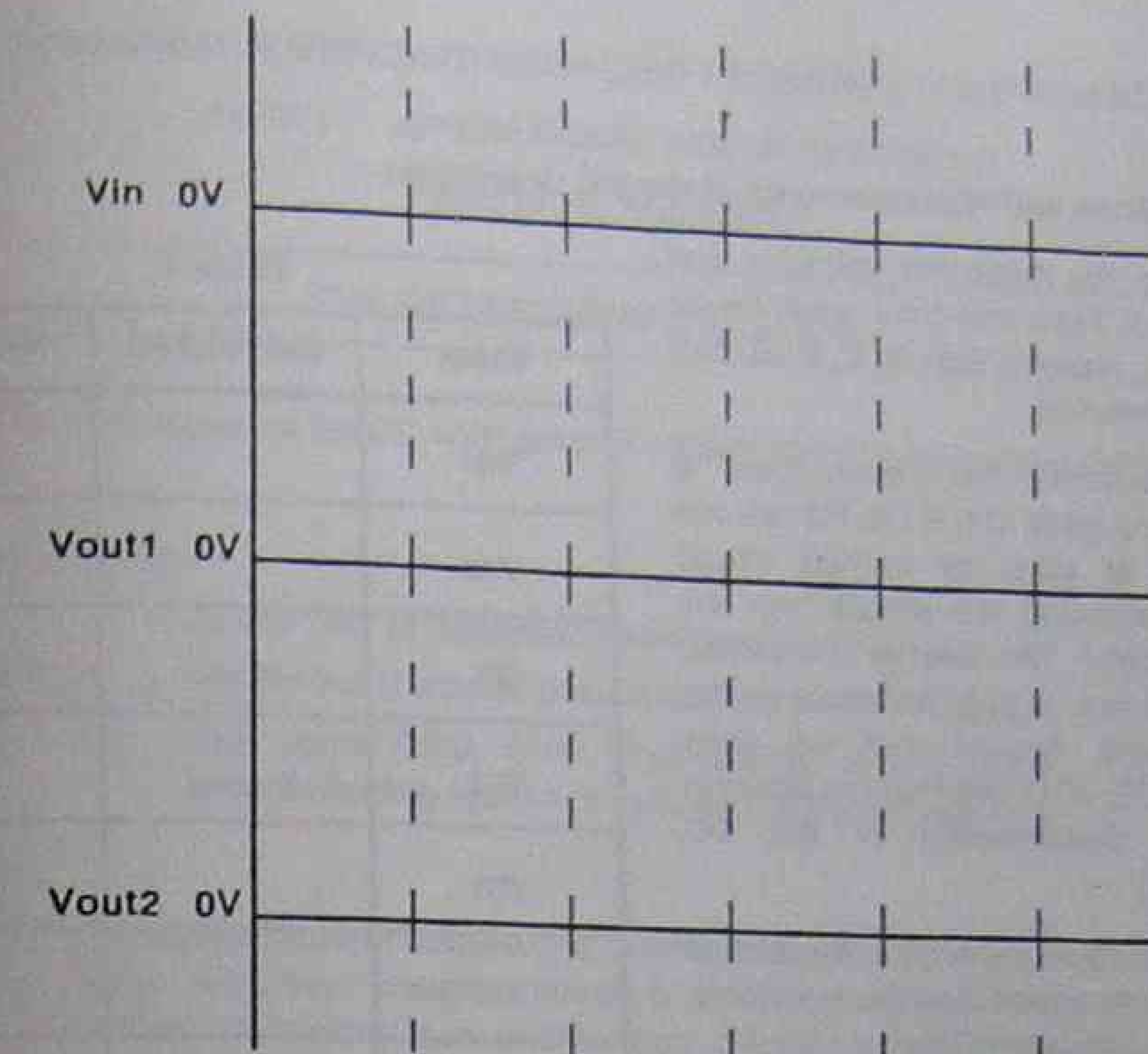


Table 2

Signal	Value (Vp-p)	Voltage gain (measured)
Vin		
Vout1		
Vout2		

- Using the results of Table 2 determine the voltage gain where Av = Vout/Vin. Record these values in Table 2.
- From the waveforms, determine the peak to peak output voltage between Vout1 and Vout2. Using this value, determine the voltage gain of the circuit for a balanced output.

Av(bal) = ______

- Calculate the theoretical voltage gains of the circuit for the unbalanced and balanced output connections where:

$Av\ (unbalanced) = \frac{R_C}{2r_e}$ $\quad Av(balanced) = \frac{R_C}{r_e}$ $\quad$ (r_e equals value from Table 1)

Calculated gains

Av (unbalanced) = ______ $\quad$ Av (balanced) = ______

PROCEDURE

Step 1: Calculations and Measurements of the DC Conditions

- Calculate the quiescent voltages and currents of Fig.2 and enter your results in Table 1. Assume links B, C, F, G, and H are connected.
- Either construct the circuit onto a breadboard (note Q3, R1 to R3 are not required) or plug the printed circuit board containing the circuit into the mother board. The teacher will advise how the power supply connects to the motherboard. Ensure that the links labelled B, C, F, G and H are in position to allow measurement of the DC conditions.
- Apply ±10V (dual polarity) to the circuit then measure and record the quiescent VOLTAGES shown in Table 1.
- Use the measured quiescent voltages of Table 1 to determine the measured values of current and r_e.
- Answer the following question on these values:

Table 1

Value	Calculated	Measured
V_{B1}		
V_{B2}		
V_E		
V_{C1}		
V_{C2}		
I_{C1}		
I_{C2}		
I_{EE}		
r_e		

QUESTION

Describe how these quiescent voltage values show that the differential amplifier is biased on, even though the bases of each transistor are grounded.

Step 2: AC Conditions for a Single Ended Input

In this section you will determine the AC conditions of the circuit under two conditions:

(a) with a single ended input (unbalanced) and an unbalanced output.

(b) with an unbalanced input and balanced output, by using the values measured for the configuration of (a).

- With the DC supply OFF, remove link B and connect link A. (All other links remain as previously connected). This configuration now sets the circuit to an unbalanced input.
- Connect a sinewave generator (set to 1kHz) to the input. Connect a CRO to monitor Vin (channel 1) and Vout1 (channel 2).
- Apply power to the circuit and adjust the output level of the signal generator so that the signal voltage at Vout1 equals 3Vp-p.
- On the axes shown on the next page sketch the waveforms present at Vin, Vout1 and Vout2. Ensure the waveforms are drawn to show their phase relationships. Also record the peak-peak voltage values on the waveforms and in Table 2.

THE DIFFERENTIAL AMPLIFIER

AIM: To calculate and measure the DC voltages and voltage gain of a discrete component differential amplifier.

EQUIPMENT REQUIRED:

Resistors: 3 x 4k7 ¼W (if dedicated panel not available)

Semiconductors: LM394 matched pair (if dedicated panel not available)

Equipment: dual trace CRO, DVM, dual polarity power supply, audio oscillator, breadboarding system (if dedicated panel not available)

INTRODUCTION

The biasing of DC coupled discrete component circuits depends on the internal parameters of the active devices. When the direct coupled circuit is a differential pair, slight differences in the parameters of the two transistors can create large differences in Ic, and hence Vc. Internal parameters can be made almost identical if the two transistors are formed on the same integrated circuit. The following specifications are for an LM394, super-matched pair. Constructing a differential amplifier with this device will give a DC coupled circuit with well balanced quiescent currents and voltages.

The LM394 is a junction isolated ultra well matched monolithic NPN transistor pair with an order of magnitude improvement in matching over conventional transistor pairs. This was accomplished by advanced linear processing and a unique device structure.

Electrical characteristics of these devices such as drift versus initial offset voltage, noise and the exponential relationship of base-emitter voltage to collector current closely approach those of a theoretical transistor. Extrinsic emitter and base resistances are much lower than presently available pairs, either monolithic or discrete, giving extremely low noise and theoretical operation over a wide current range. Most parameters are guaranteed over a current range of 1µA to 1mA and 0V up to 40V collector-base voltage, ensuring superior performance in nearly all applications.

To guarantee long term stability of matching parameters, internal clamp diodes have been added across the emitter-base junction of each transistor. These prevent degradation due to reverse biased emitter current - the most common cause of field failures in matched devices. The parasitic isolation junction formed by the diodes also clamps the substrate region to the most negative emitter to ensure complete isolation between devices.

FEATURES

- Emitter-base voltage matched to 50µV
- Offset voltage drift less than 0.1µV/°C
- Current gain matched to 2%
- Common-mode rejection ratio greater than 120dB (1,000,000)
- Parameters guaranteed over 1µA to 1mA collector current
- Extremely low noise
- Superior logging characteristic compared to conventional pairs
- Plug-in replacement for presently available devices

Absolute Maximum Ratings

Collector current	20mA
Collector-emitter voltage	40V
Base-emitter current	±10mA
Power dissipation	500mW

Connection Diagram
(metal can package)
top view

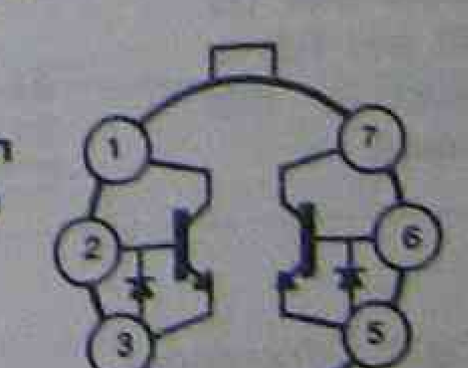


Fig.1: The LM394

Electrical Characteristics (T_J = 25°C) - unless stated, V_{CB} = 0V to VMAX

Parameter	Conditions	Min	Typ	Max	Units
Current gain (h_{FE})	I_C = 1mA	300	700		
	I_C = 100µA	250	550		
	I_C = 10µA	200	450		
	I_C = 1µA	150	300		
Current gain match (h_{FE} match)	I_C = 10µA to 1mA		0.5	4	%
	I_C = 1µA		1.0		%
Emitter-base offset voltage	I_C = 1µA to 1mA (V_{CB} = 0)		25	150	µV
Collector-emitter voltage (VMAX)				40	V
Collector-base leakage	V_{CB} = VMAX		0.05	0.5	nA
Collector-collector leakage	V_{CC} = VMAX		0.1	5	nA
Collector to emitter saturation voltage	I_C = 1mA, I_B = 10µA		0.2		V
	I_C = 1mA, I_B = 100µA		0.1		V

The LM394 is used in high precision circuits such as instrumentation amplifiers and some of its electrical characteristics are given above. From these determine the specifications listed on the right:

(a) h_{FE} for a collector current of 1mA Answer: ______

(b) The collector to collector leakage current at maximum rail voltage Answer: ______

(c) Maximum operating voltage Answer: ______

(d) Current gain match for a collector current of 1mA Answer: ______

(e) $V_{CE(SAT)}$ for a collector current of 1mA Answer: ______

Fig.2 shows the circuit for this assignment.

Ideally this circuit should be available on a panel, otherwise the circuit can be constructed on a prototyping board. The section associated with Q3 (R1, R2 and R3) is not used in this assignment.

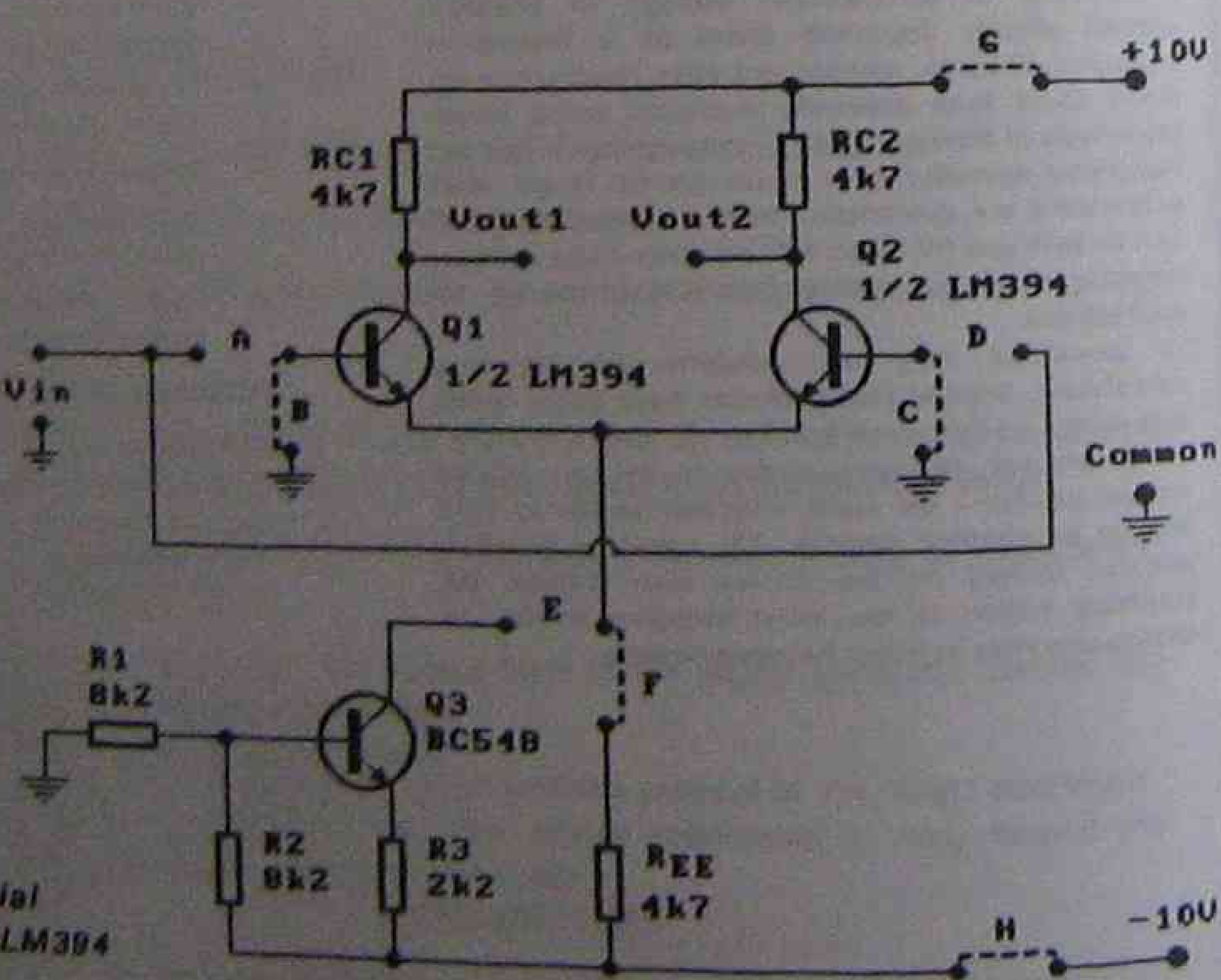


Fig.2: Differential amplifier using the LM394

QUESTIONS

(a) Compare the calculated and measured values for gain and give brief reasons for any differences.

______

______

______

(b) Identify the phase relationships between the signals at the input and the two outputs.

______

______

(c) Compare the voltage gain between the unbalanced and balanced output connections.

______

______

(d) State the main advantage of a direct coupled amplifier over an RC coupled amplifier.

______

(e) Determine the effect on the DC conditions for the circuit if R_{C1}, R_{C2} and R_{EE} were changed to 27k. What effect would this change have on the current gain (h_{FE}) of the transistors? (Hint: check manufacturers' specs and include in your explanation).

______

______

______

______

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1   OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 2

COMMON MODE REJECTION

AIM:

(a) To examine the operation of a basic constant current source.

(b) To determine the effect of a constant current source on the CMRR of a differential amplifier

(c) To observe the direction of the output current of an op amp for different polarities of the input voltage.

EQUIPMENT REQUIRED:

Resistors:	1 x 2k2, 3 x 4k7, 2 x 8k2 all ¼W (if dedicated panel not available)
Semiconductors:	LM394 matched pair, BC548 (if dedicated panel not available)
Equipment:	dual trace CRO, 2 x multimeters (or a 10mA ammeter and a DVM), dual polarity power supply, variable power supply, audio oscillator, breadboarding system

INTRODUCTION

Differential amplifiers can reject noise signals that are common to both inputs and amplify a difference voltage between the inputs. The differential amplifier circuit used in this assignment is the same as that used in the previous assignment, except the constant current source will now be used to show its ability to improve the common mode rejection ratio (CMRR) of a differential amplifier. Note that where this circuit is not available on a printed circuit, it will need to be constructed on a prototyping board.

The second part of this assignment examines the output current directions of an op amp. The type of op amp specified is a TL071 (or an LF411). The data for the LF411 is included in the Theory Manual. Other ICs such as the 741 can also be used, and the data for the 741 is also in the manual.

PROCEDURE

Step 1: The Constant Current Source

- Refer to Fig.1 on the next page and calculate the current that will flow through Q3 if link E is in place. Record this value below.

I_{EE} = I_{C3} = ____mA

- Either construct the circuit of Fig.1 onto a breadboard or plug the printed circuit board containing the circuit into the mother board. The teacher will advise how the power supply connects to the motherboard. Insert the links labelled B, C, G and H and connect a DC ammeter in place of link E. Note that link F is *not* connected.
- Apply ±10V (dual polarity) to power the circuit then measure and record the current flowing in Q3 (= I_{EE}). Record this value in Table 1 next to Vin = 0V (as set by links B and C).

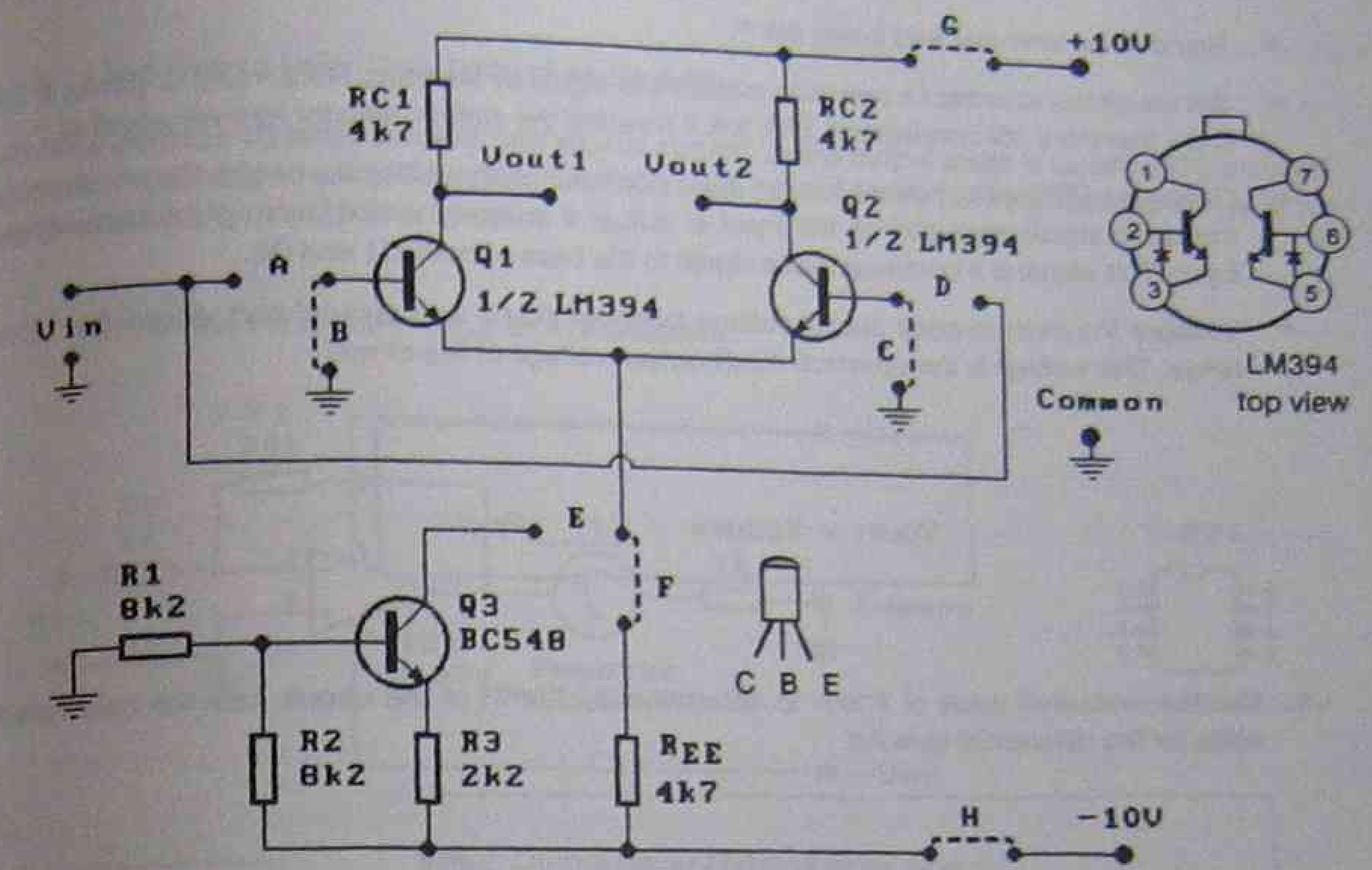


Fig.1: Differential amplifier and constant current source

- Remove link B and connect link A. This now sets the circuit to an unbalanced input.
- Apply the DC input voltages specified in Table 1, and record the value of the current I_{C3} for each voltage.

Table 1

Vin	I_{C3} = I_{EE}
0V	
-0.5V	
-0.2V	
+0.2V	
+0.5V	

Step 2: Measurement of Common Mode Rejection

- Calculate the CMRR of Fig.1 assuming R_{EE} equals 4k7. Record this value below.

CMRR = ________ (with R_{EE} = 4k7)

- Calculate the approximate voltage gain of the differential amplifier of Fig.1 assuming a balanced input and balanced output, the component values shown and link F connected.

A_D (balanced I/P and O/P) $= \frac{R_C}{r_e} = \frac{\_\_}{r_e} =$ ______

- Remove the ammeter and insert link F.
- Set the circuit to accept a common mode input signal by inserting links A and D. (Links B and C are therefore not connected). With link F inserted the emitter resistor REE will equal 4k7.
- Connect a CRO with channel 1 to the input (Vin) and channel 2 to the output VOUT1. Connect the audio signal generator to the input at adjust it to apply a 1kHz, 4Vp-p sinewave to the input. This signal is a common mode signal to the base of both Q1 and Q2.
- Measure the peak-to-peak output voltage between VOUT1 and ground and record this value below. This voltage is the common mode output voltage of the circuit.

VOUT1 = VOCM = ______Vp-p

- Use the measured value of VOUT1 to determine the CMRR of the circuit. Use the calculated value for the differential gain Ad.

ACM = Vout1 / Vin = ____ / 4Vp-p = ______

and CMRR = Ad / ACM = ____ = ______

Values for fixed emitter resistor

- Remove link F and connect the constant current source by inserting link E.
- Apply a 1kHz, 4Vp-p sinewave to the input as before. Measure the peak-to-peak output voltage between VOUT1 and ground and record this value below. This voltage is the common mode output voltage of the circuit, this time with the constant current source in place of the fixed value emitter resistor REE.

VOUT1 = VOCM = ______Vp-p

- As before, use the measured value of VOUT1 to determine the CMRR of the circuit. Use the calculated value for the differential gain Ad.

ACM = Vout1 / Vin = ____ / 4Vp-p = ______

and CMRR = Ad / ACM = ____ = ______

Values for constant current source

Step 3: The Output Drive Capability of an Op Amp

General purpose op amps can sink and source current, as the output stage is usually a DC coupled, complementary power amplifier. The maximum load current varies between devices and is in the range of 25mA for small power op amps.

- Construct the circuit of Fig.2 on a breadboard.

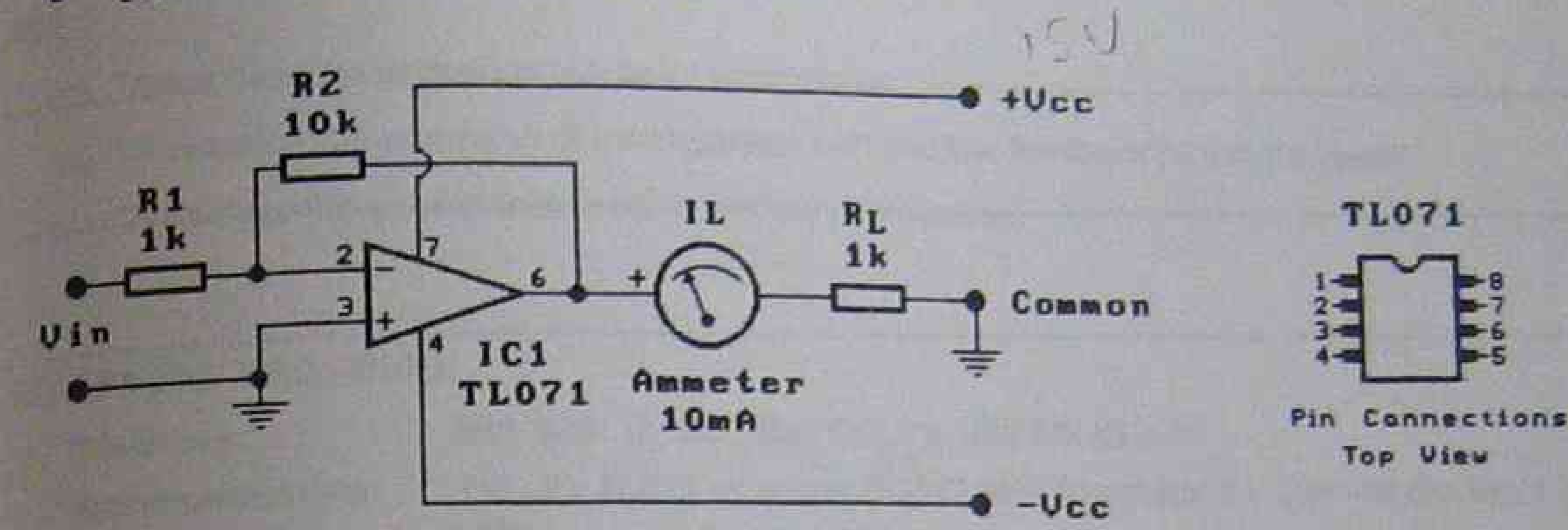


Fig.2: Output current drive of an op amp.

- Connect a dual polarity supply of ±15V to the circuit.
- Apply +1VDC between the input and common terminals. Record the direction (sink or source) and the value of the load current.

Direction of the load current = __________

Value of the current = _____mA

- Apply -1VDC between the input and common terminals. Record the direction (sink or source) and the value of the load current.

Direction of the load current = __________

Value of the current = _____mA

- See next page for conclusion questions...

QUESTIONS

(a) With respect to the constant current source in Fig.1 briefly explain how the value of the current is set.

(b) Compare the CMRR figures obtained for the differential amplifier when the constant current source was used compared to the fixed value resistor.

(c) Using the data on the LF411 op amp in the theory notes, determine the maximum output current rating of the IC.

(d) What circuit feature of an op amp allows bi-directional output current flow?

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 3

COMPARATORS

AIM:

(a) To examine the operation of a basic comparator.

(b) To examine the operation of a comparator with positive feedback (Schmitt trigger).

(c) To analyse the operation of a basic window comparator.

EQUIPMENT REQUIRED:

Resistors:	390, 820, 1k, 2k7, 3k3, 4k7, 7 x 10k, 47k all ¼W
Semiconductors:	741, 2 x TL071 op amps, BC547 NPN transistor, 2 x 1N4148 diodes, 1 x red LED
Equipment:	dual trace CRO, DVM, dual polarity power supply, variable power supply, breadboarding system.

INTRODUCTION

Comparator circuits are widely used in industrial control circuits as they can indicate when an analog input is above or below a set reference level. This assignment examines the operation of the basic comparator, the Schmitt trigger and a window comparator. The types of op amp being used are the TL071 (or an LF411) and the 741. The data for both ICs is the Theory Manual.

PROCEDURE

Step 1: The Basic Non-Inverting Comparator

- Calculate the reference voltage for the comparator of Fig.1. The value for +Vcc will depend on the available equipment and should be either ±12V or ±15V. Thus if the supply is ±12V, +Vcc = 12V.

V_{REF} = _______ volts

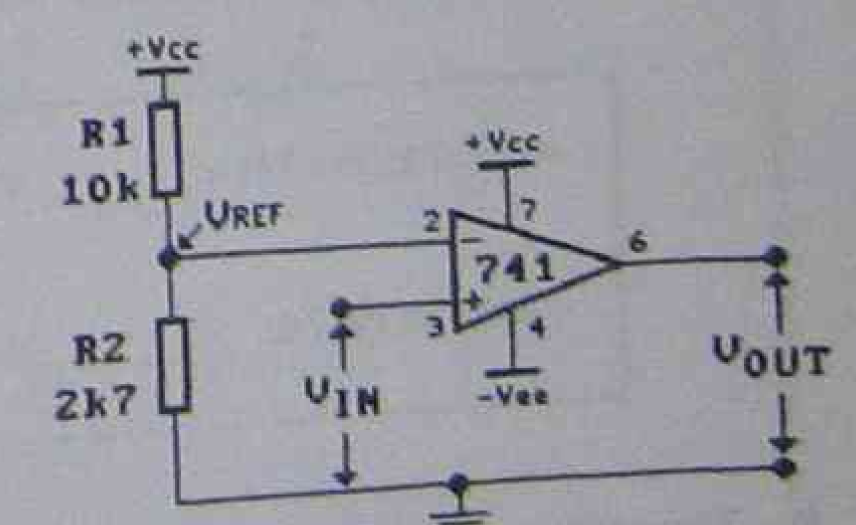


Fig.1: The basic comparator

- Construct the circuit of Fig.1 on a prototyping board. Apply a dual polarity supply of ±12V (or ±15V) to power the circuit.

- Connect a variable power supply to the input (V_{IN}), positive to pin 3 of the IC, and connect a DVM to measure V_{OUT}. Vary V_{IN} and for each value specified in Table 1, record the value and polarity of V_{OUT}. Measure to one decimal place.
- By varying the value of V_{IN}, determine the input voltage that causes the output to change state (the threshold voltage V_T). Record this value in Table 1.

Table 1

V_{IN}	Op amp type		
	741	TL071	TL071 - single supply
0V			
1V			
2V			
3V			
4V			
5V			
Measured value of V_T			

- Replace the 741 with a TL071 and repeat the previous two steps. Record the readings in Table 1.
- From the results of Table 1, determine the output saturation levels V_{OSAT} for both IC types.

$+V_{OSAT}$ for the 741 = ______V $+V_{OSAT}$ for the 071 = ______V

$-V_{OSAT}$ for the 741 = ______V $-V_{OSAT}$ for the 071 = ______V

- Turn off the power and disconnect the -ve supply from pin 4 of the IC. Reconnect pin 4 to the 0V (common) rail. The op amp is now working off a single-rail power supply. Vary V_{IN} as per Table 1 and record the output voltage levels and the threshold voltage.
- Vary V_{IN} slowly around the threshold level. Note that there is an area where the switching transition is poorly defined. Positive feedback, as used in the following circuit will overcome this limitation of the basic comparator.

Step 2: The Differential Comparator - With Voltage Hysteresis

- For the circuit of Fig.2, calculate the upper and lower threshold voltages (V_{UT} and V_{LT}). Refer to the theory notes for the equations.

V_{UT} = ______ V

V_{LT} = ______ V

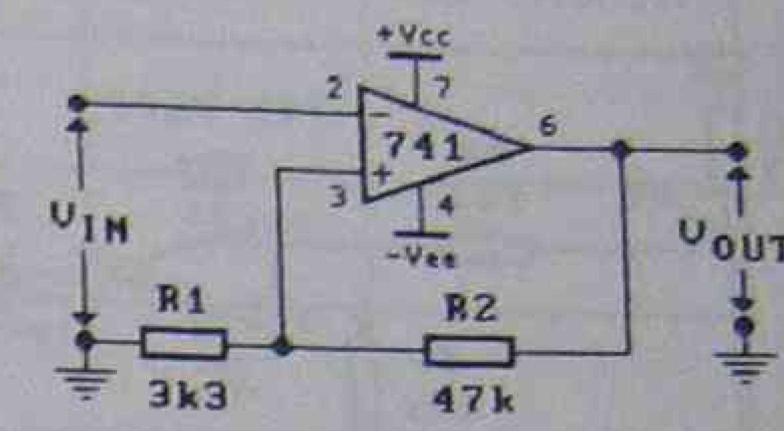
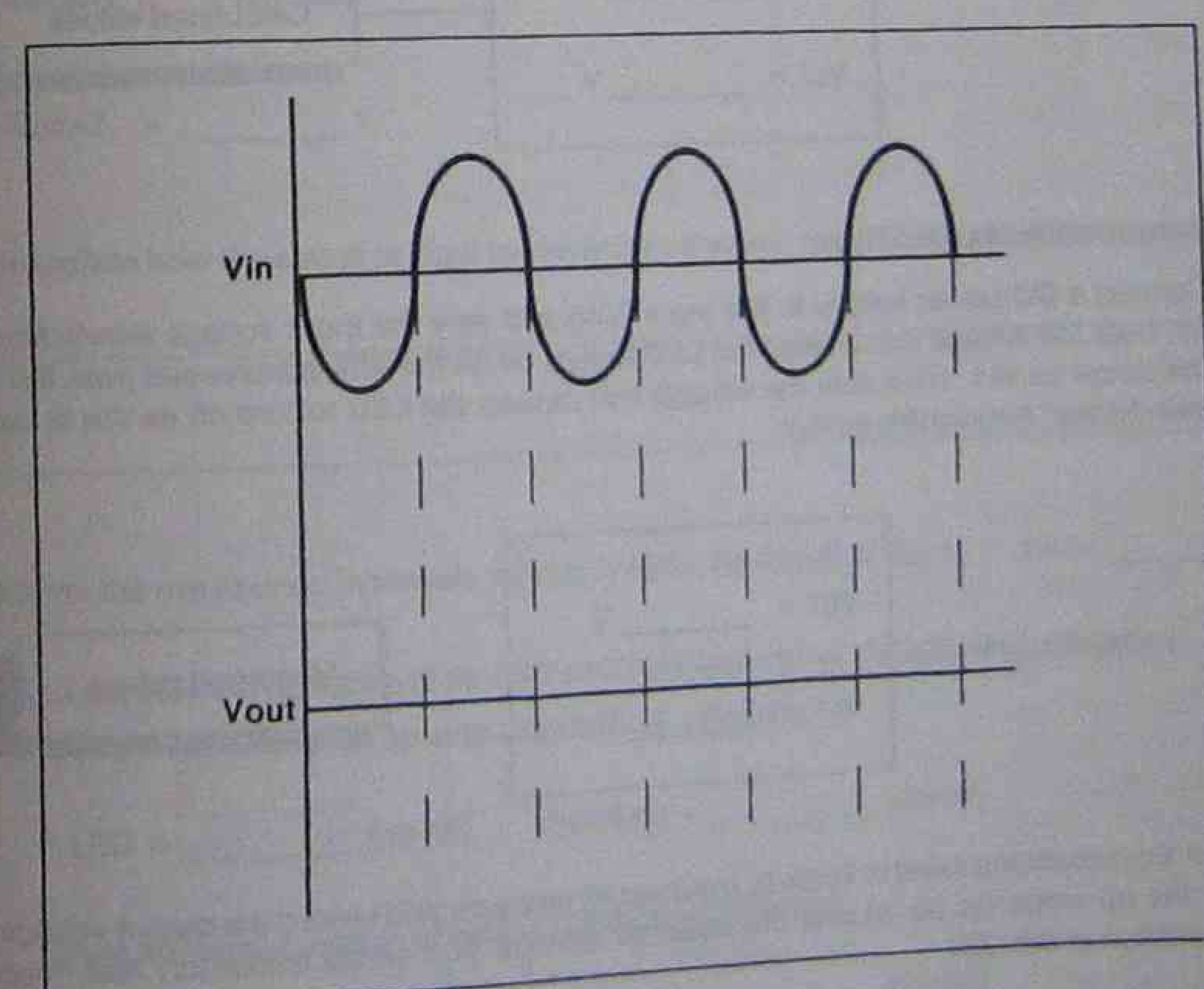


Fig.2: Comparator with positive feedback

- Construct the circuit of Fig.2 on the prototyping board.
- Connect a signal generator set to 10Vp-p, 1kHz sinewave to V_{IN}. Monitor V_{IN} with channel 1 of a CRO and V_{OUT} with channel 2.
- Sketch the output waveform on the axes below. Show the relationship of the output voltage to the input sinewave, indicating on the sinewave the voltage levels (V_{UT} and V_{LT}) that switching of the output occurs.



Step 3: The Window Comparator

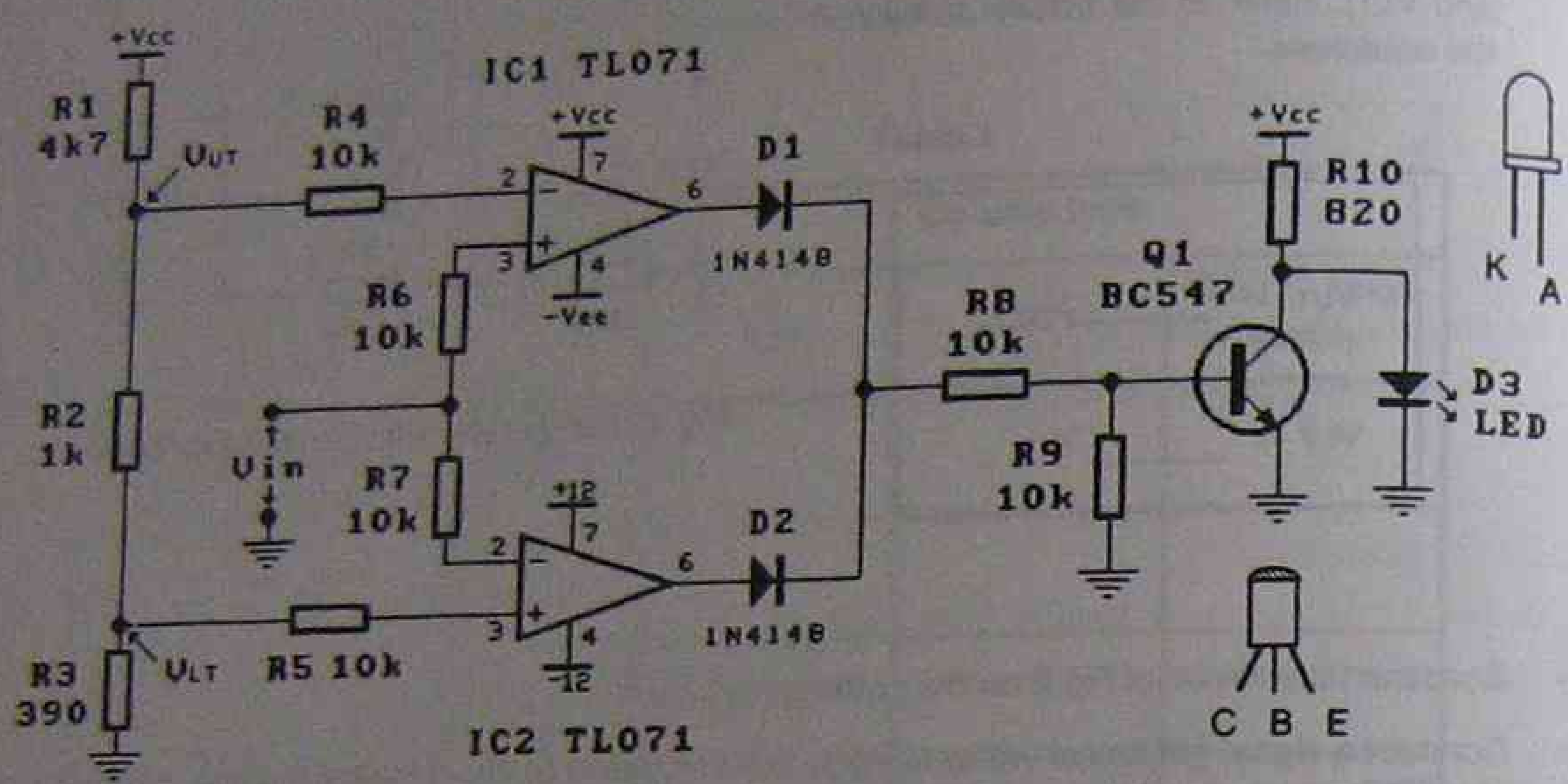


Fig.3: The window comparator

- For the circuit of Fig.3, calculate the upper (V_{UT}) and lower (V_{LT}) threshold voltages.

V_{UT} = ______ V	**Calculated values**
V_{LT} = ______ V	

- Construct the circuit of Fig.3.
- Connect a DC power supply to the input (V_{IN}) and vary the input voltage slowly from 0V to 5V. Note the voltage that causes the LED to turn on as the VIN is increased from 0V. Record this below as V_{LT}. Then note the voltage that causes the LED to turn off as V_{IN} is increased even further. Record this as V_{UT}.

V_{UT} = ______ V	**Measured values**
V_{LT} = ______ V	

- For the conditions listed in Table 2, (next page) measure and record the output voltage levels of the op amps (at pin 6) and the collector voltage V_{C1} of the transistor. Also record the condition of the LED.

Table 2

Condition	IC1: V_{out1}	IC2: V_{out2}	V_{C1}	LED: ON or OFF
Vin less than V_{LT}				
Vin greater than V_{LT} but less than V_{UT}				
Vin greater than V_{UT}				

QUESTIONS

(a) From the results of Table 1, state the typical difference between the supply voltage and the maximum output voltage (V_{OSAT}) for an op amp operated from a dual polarity power supply.

$+V_{OSAT}$ = +Vcc less ______ V

$-V_{OSAT}$ = -Vcc less ______ V

(b) From the results of Table 1, state the typical difference between the supply voltage and the maximum output voltage (V_{OSAT}) for an op amp operated from a single rail power supply.

$+V_{OSAT}$ = +Vcc less ______ V

$-V_{OSAT}$ = ______ V

(c) Briefly explain how the circuit of Fig.2 converts the sinewave input to the waveform obtained.

______

______

(d) Determine the *measured* hysteresis voltage (V_H) for the circuit of Fig.2 : V_H = ______ V

(e) For Fig.3, identify the condition of the LED and the transistor for the following conditions:

- when either one of the op amp outputs is at $+V_{O(SAT)}$.

  LED = ______ (on-off) Transistor = ______ (on-off)

- when the output voltage of both op amps is at $-V_{O(SAT)}$.

  LED = ______ (on-off) Transistor = ______ (on-off)

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 4

TIMERS

AIM:

(a) To examine the operation of the 555 timer IC as a monostable.

(b) To examine the operation of the 555 timer as an astable multivibrator.

EQUIPMENT REQUIRED:

Resistors:	1k, 10k, 15k, 2 x 56k, 560k, all ¼W
Capacitors:	1nF, 10nF, 15nF, 0.1μF (polyester), 4.7μF electrolytic
Semiconductors:	555 timer IC, BC547 NPN transistor, 1N4148 diode,
Equipment:	dual trace CRO, DVM, variable power supply, breadboarding system.

INTRODUCTION

The 555 timer finds considerable application in circuits that require time delays or continuous pulse waveforms. This IC can be used in a wide range of applications, and the two most basic are the monostable (timer) and the astable (free-running oscillator). These two configurations are examined in this assignment.

PROCEDURE

Step 1: The Monostable

- Calculate the period of the unstable state (time delay) for the circuit of Fig.1.

T = ______ mS

- Construct the circuit of Fig.1 on a prototyping board. The circuit can be powered from a single rail supply of +5V to +15V. The recommended supply voltage is +12V.

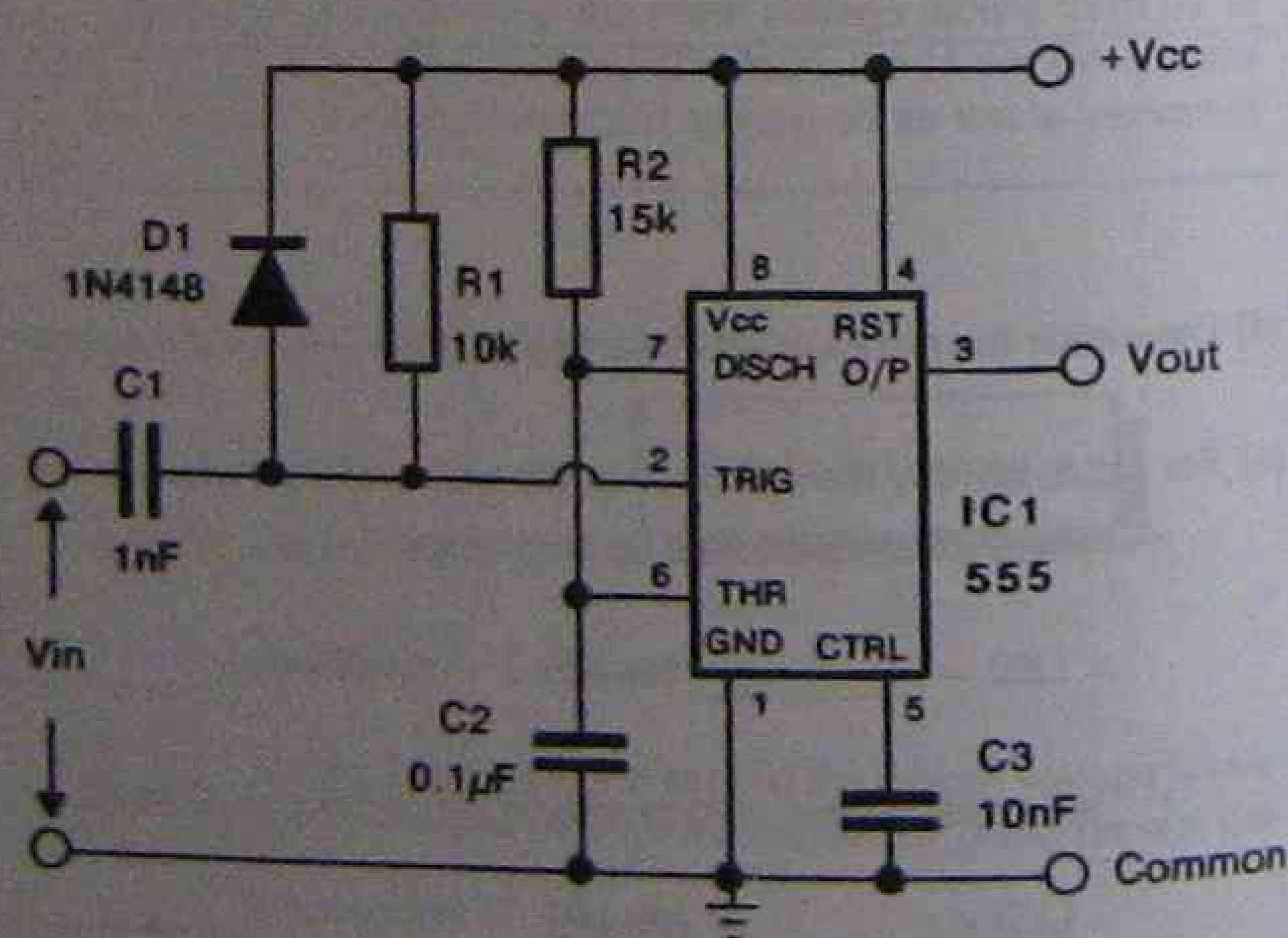
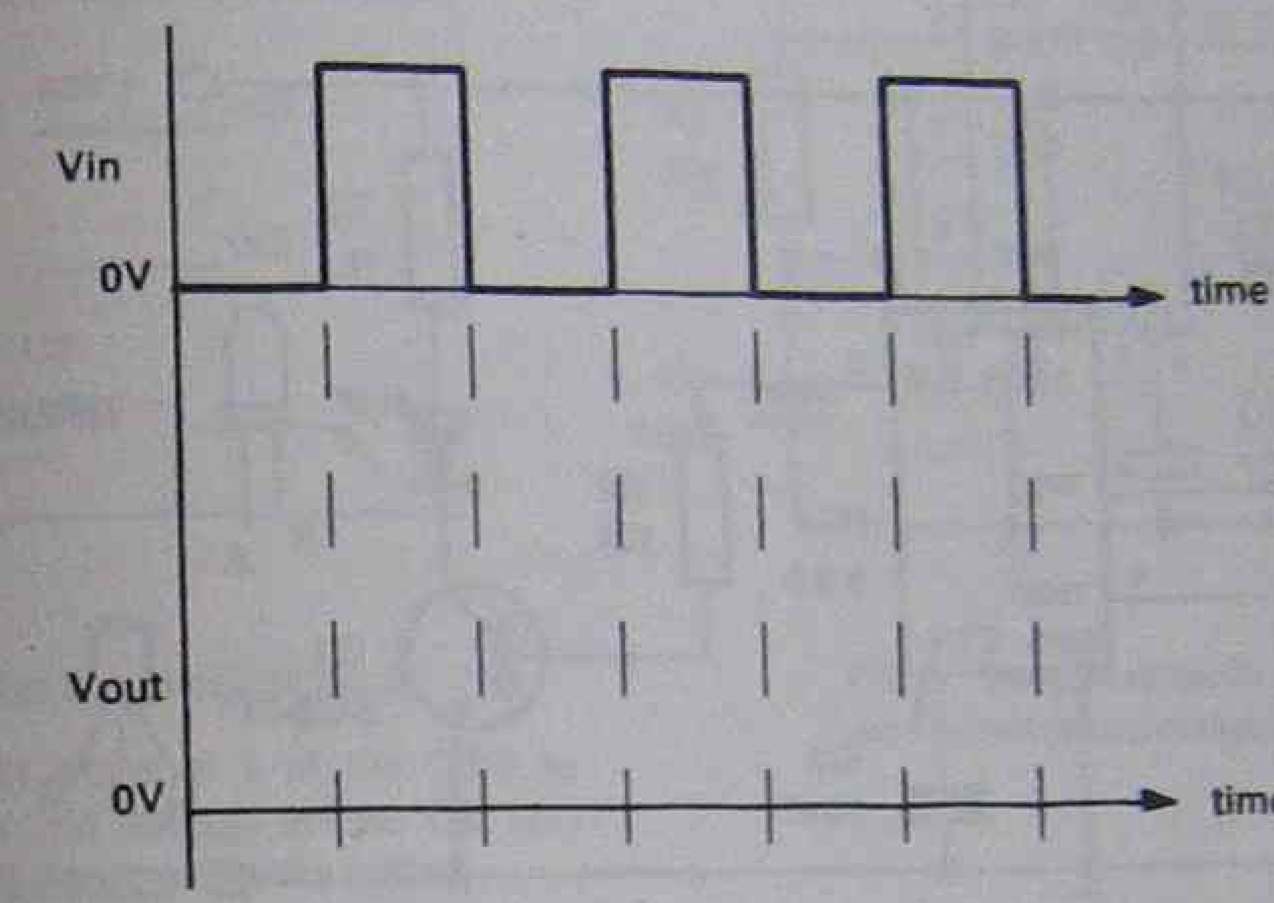


Fig.1: The 555 monostable

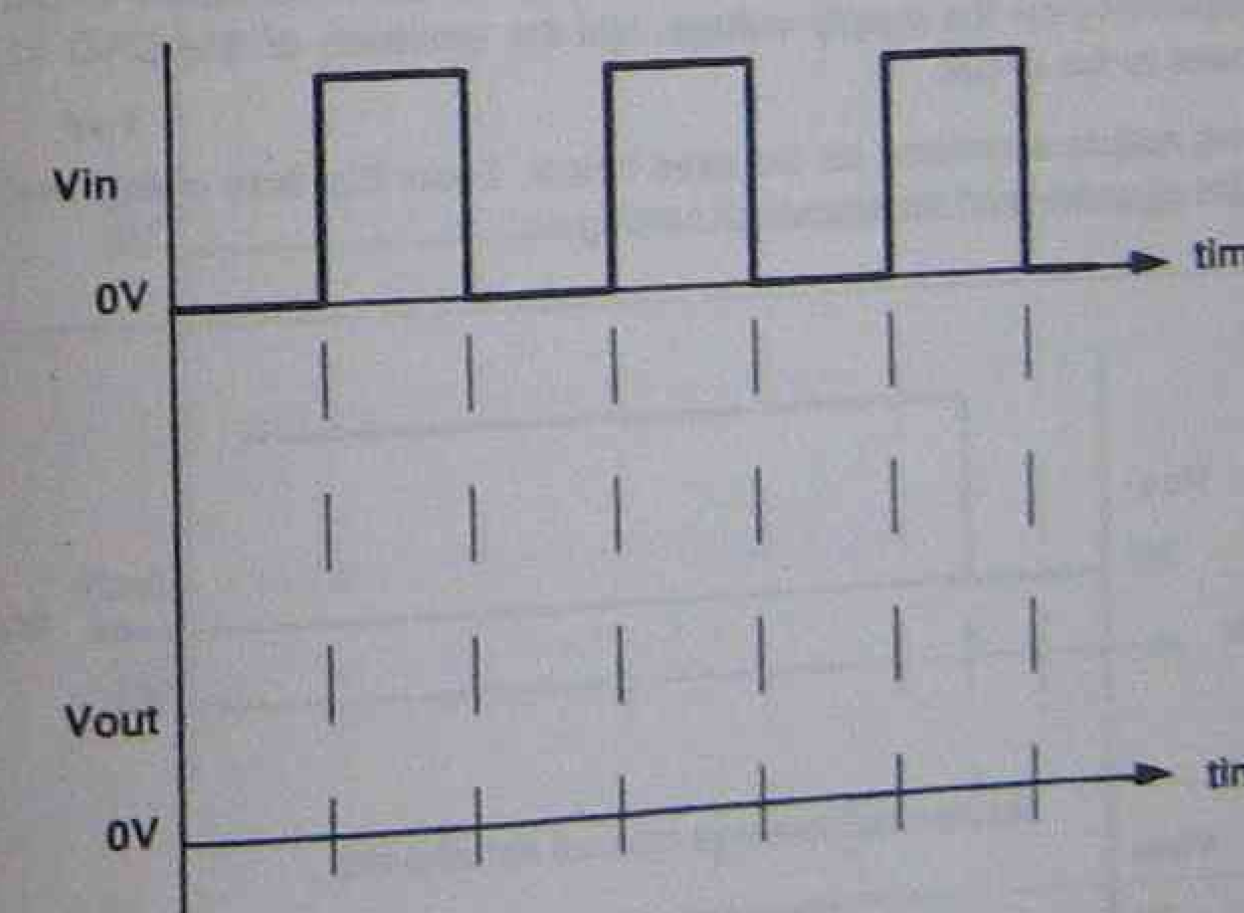
- Connect a CRO to the circuit with channel 1 to the input (VIN) and channel 2 to the output (VOUT). Also connect a signal generator to the input, with the generator set to 100Hz, square wave. Adjust the generator to give a 12Vp-p output if the supply voltage is +12V. Otherwise set the generator to give an output equal to the supply voltage.

- Record the output waveform on the axes below. Include the output voltage levels and the time periods.



Waveforms for VIN set to 100Hz

- Increase the frequency of the signal generator to 2kHz and record the output waveform on the axes below. Again include the output voltage levels and the time periods.



Waveforms for VIN set to 2kHz

Step 2: Power-On Time Delay

T = ______ seconds

- For the circuit of Fig.2, calculate the power-on time delay.
- Construct the circuit of Fig.2 on the prototyping board. If a 12V relay is available, connect it (in parallel with a diode as shown) in the collector circuit of the transistor instead of the LED and resistor R3.

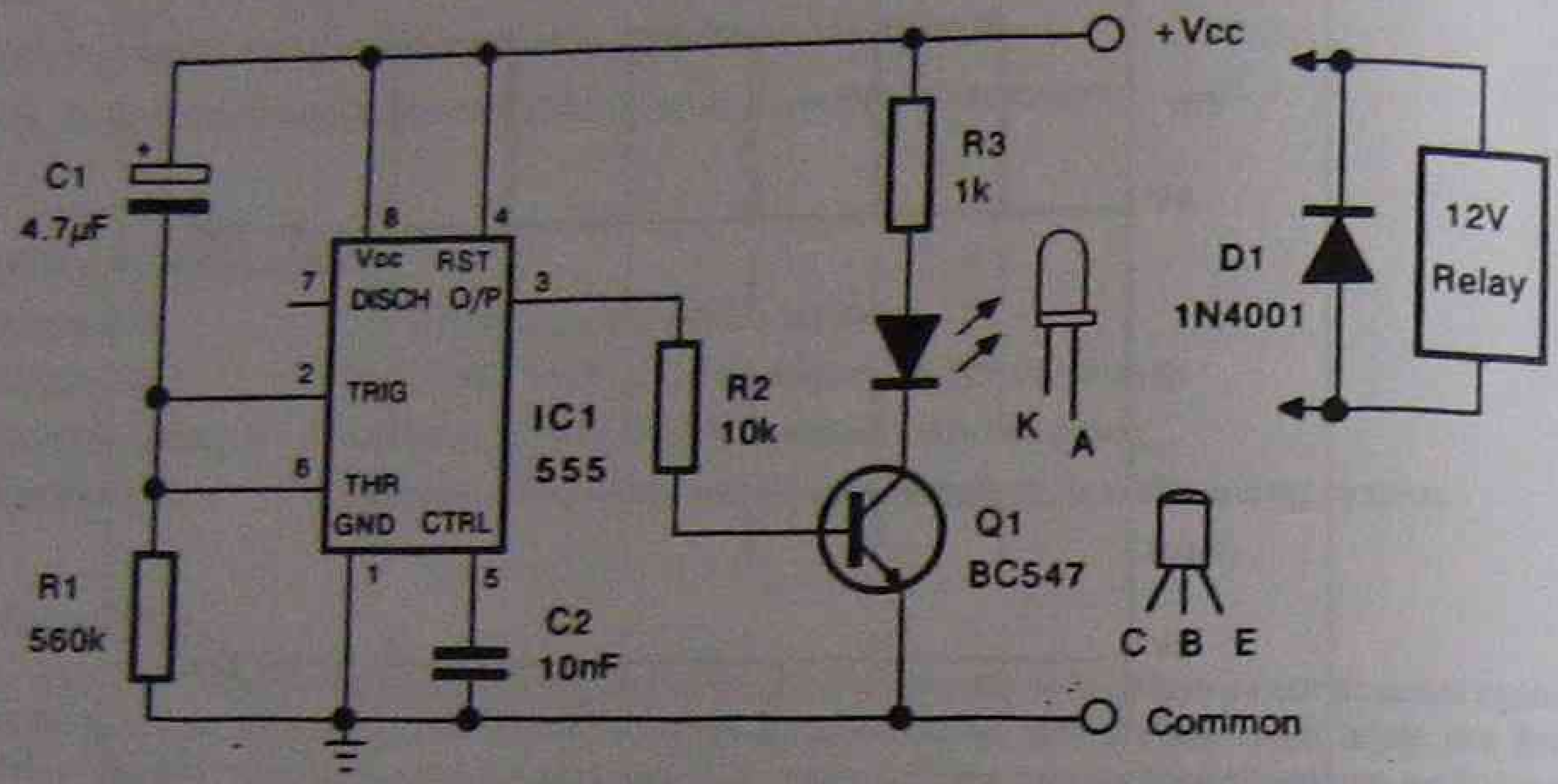
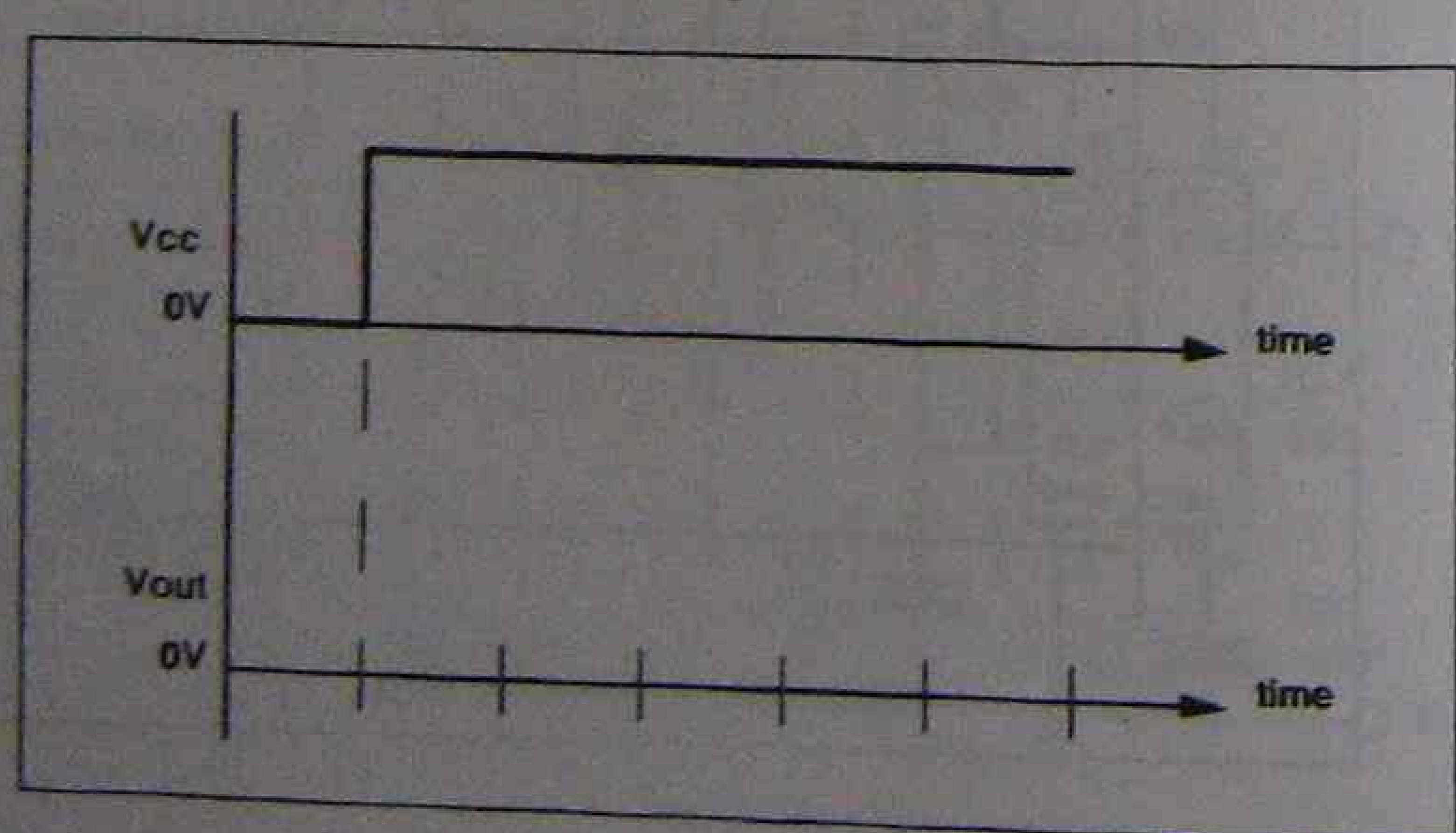


Fig.2: Power-on time delay

- This circuit should cause the LED to light (or the relay to operate) some time after power is first applied to the circuit. Apply power to the circuit and determine the time delay of the circuit by timing it with a watch.
- Turn off the power and connect a CRO with channel 1 to the supply terminal of the IC (pin 8) and channel 2 to the output (pin 3). You'll need to set the CRO inputs to DC and to a suitable V/div, depending on the supply voltage. Set the timebase of the CRO to 0.5sec/div, then apply power to the circuit.
- Sketch the output waveform on the axes below. Show the time delay that occurs between application of power and the transistor turning on.



Step 3: The 555 Astable; Non-Symmetrical Output

- For the circuit of Fig.3, calculate the time the output waveform is high (T_H) and the time it is low (T_L). Also calculate the frequency of the output waveform.

T_H = ______ mS

T_L = ______ mS

Frequency = ______ Hz

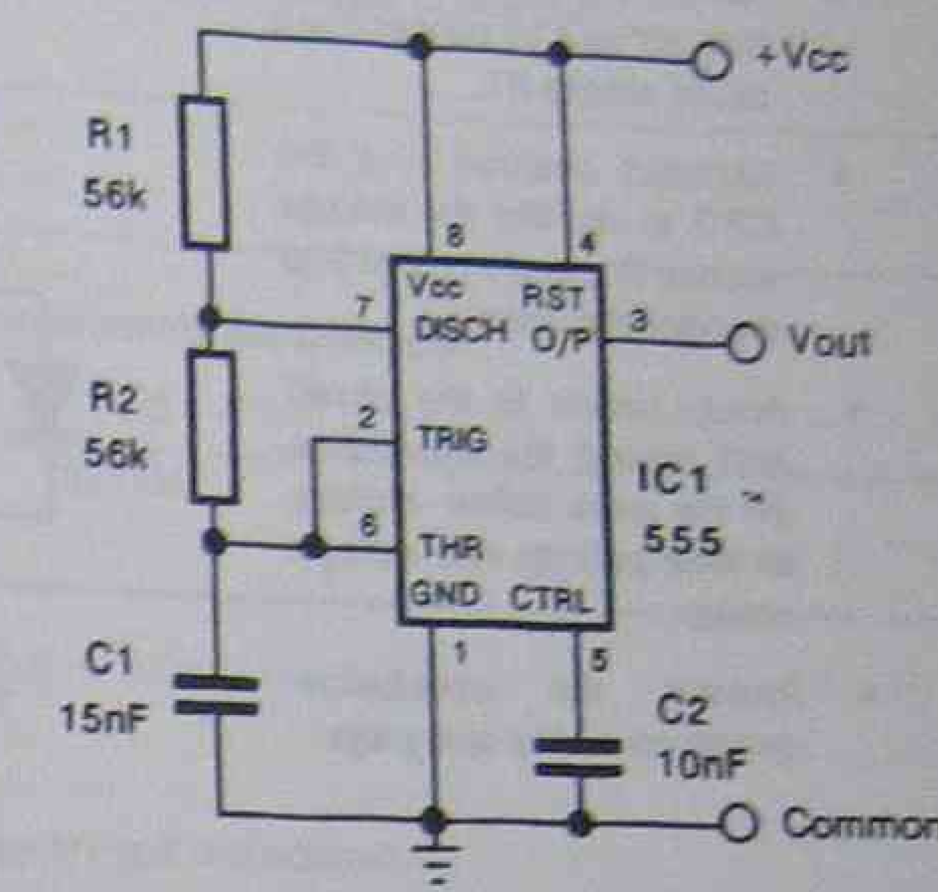
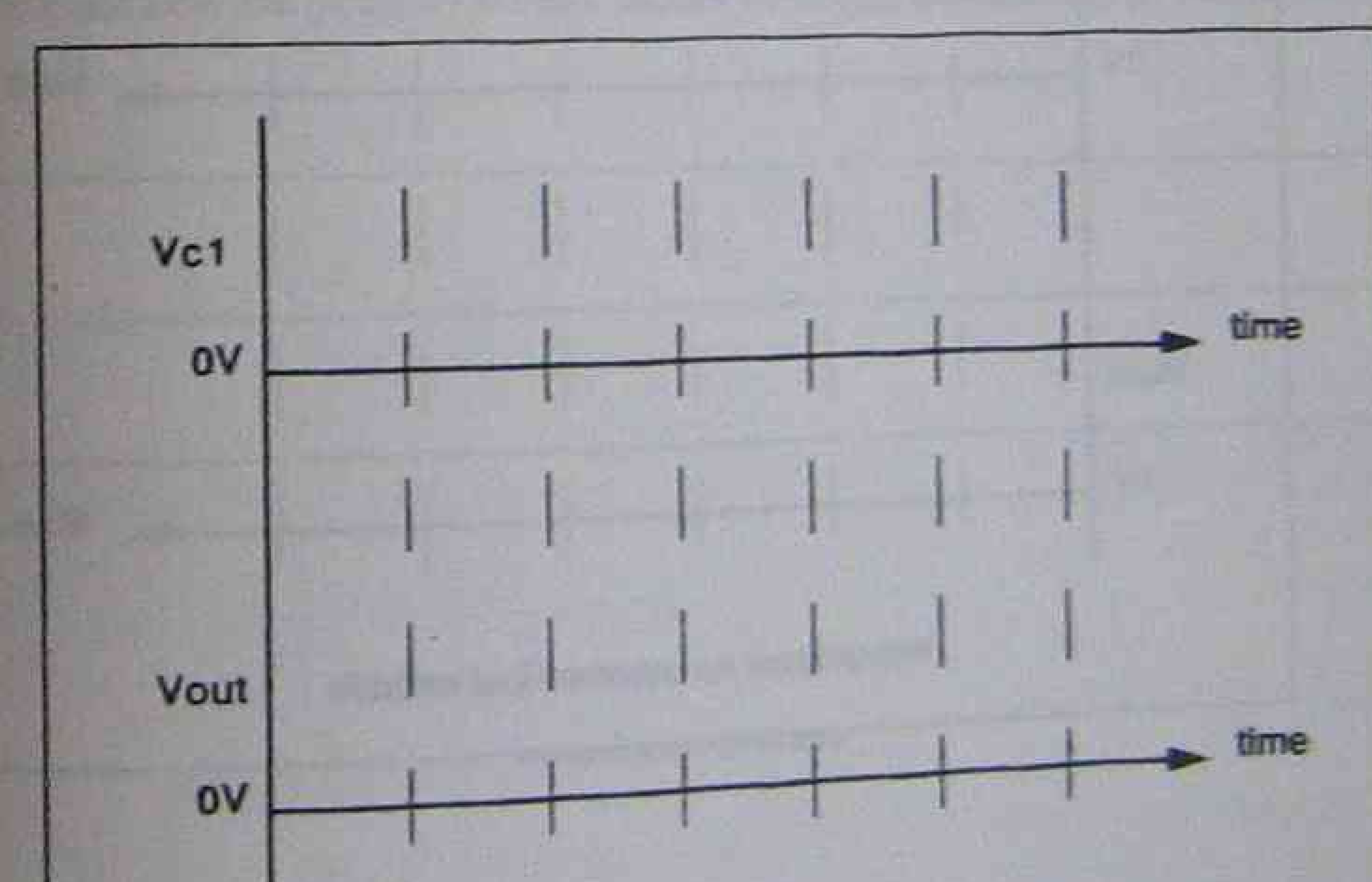


Fig.3: The 555 astable - non symmetrical output

- Construct the circuit of Fig.3.
- Connect channel 1 of the CRO to monitor the voltage across C1, and channel 2 to monitor the output.
- Apply power to the circuit and record the waveforms on the axes below. Include all time periods and voltage values.



Waveforms for non-symmetrical astable

Step 4: The 555 Astable; Symmetrical Output

- Modify the circuit of Fig.3 to that of Fig.4 by connecting a diode across R2.
- Connect channel 1 of the CRO to monitor the voltage across C1, and channel 2 to monitor the output.
- Apply power to the circuit and record the waveforms on the axes below. Include all time periods and voltage values.
- Answer the conclusion questions on the next page.

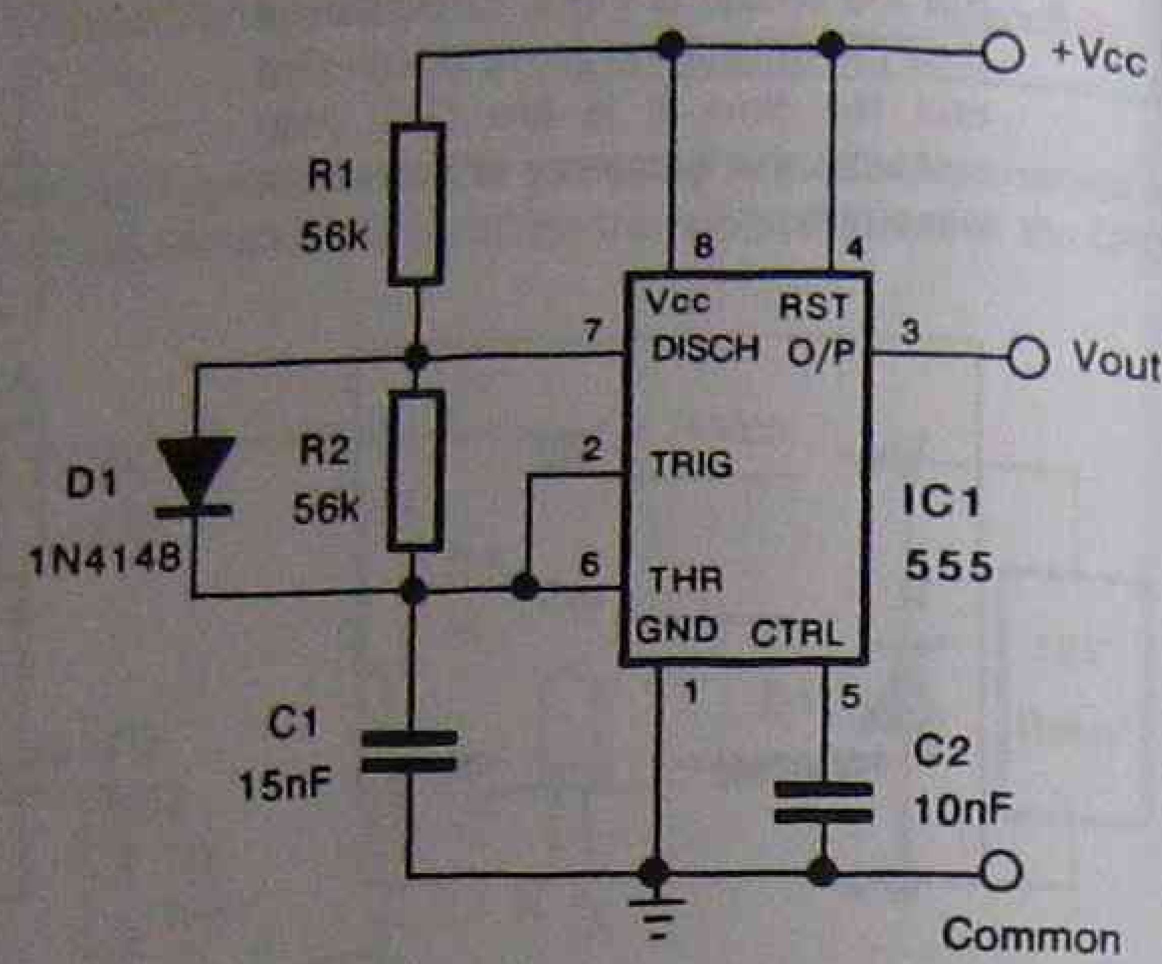
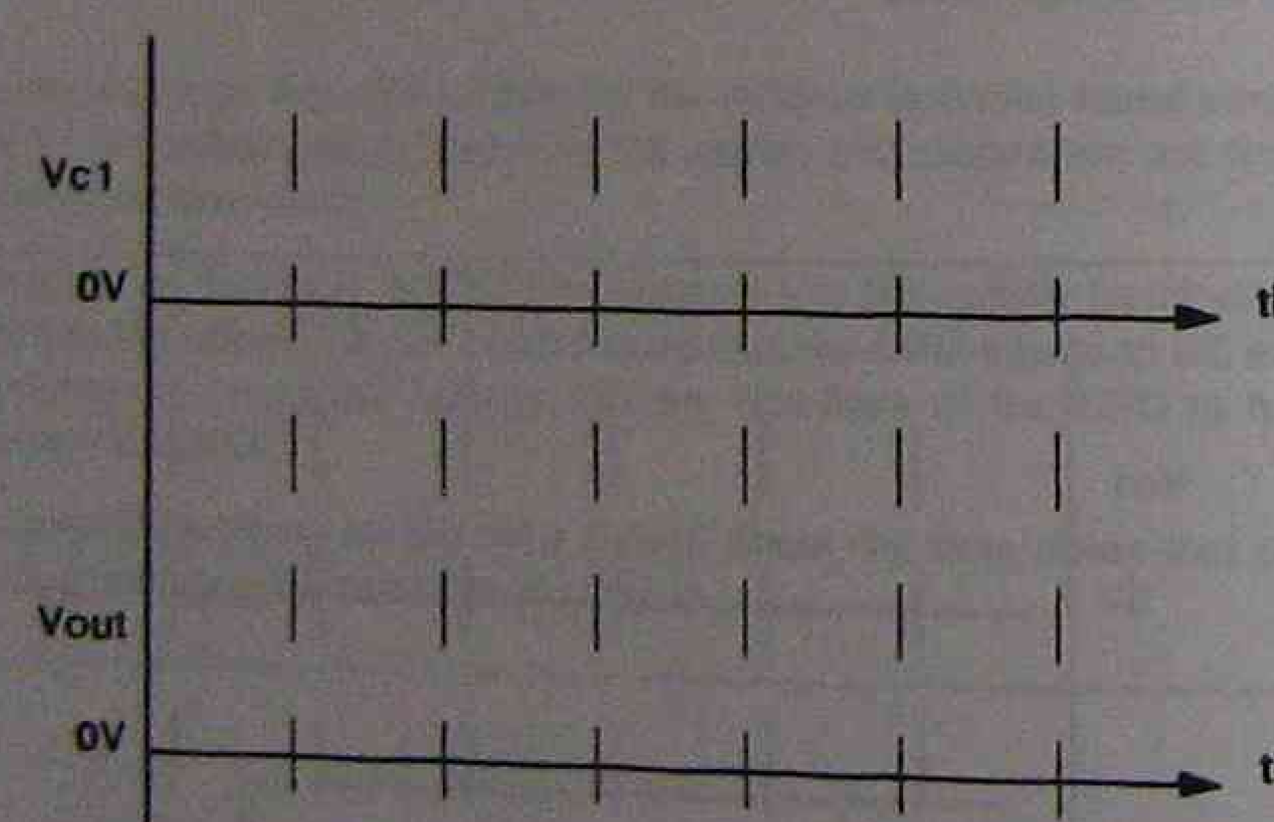


Fig.4: The 555 astable - symmetrical output



Waveforms for symmetrical astable

QUESTIONS

(a) For the waveforms of the monostable, describe the nature of the output signal when:

- the input trigger repeats slowly enough for the monostable to time-out.

____

____

- the input trigger repeats before the monostable has timed-out.

____

____

(b) Briefly explain how the power-on time delay of Fig.2 is produced.

____

____

____

(c) Briefly explain how the diode D1 in Fig.4 causes the output waveform to be symmetrical.

____

____

____

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 5

OPERATIONAL AMPLIFIER CIRCUITS - 1

AIM: To examine the operation of the inverting, non-inverting and voltage follower op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	22k, 10k, 68k, 100k, all ¼W
Semiconductors:	TL071 op amp IC
Equipment:	dual trace CRO, DVM, variable power supply, breadboarding system.

INTRODUCTION

The open loop gain of an op amp can be reduced to practical values by applying negative feedback. The two basic configurations are the inverting amplifier and the non-inverting amplifier. The voltage follower is a form of non-inverting amplifier, but with a gain of unity. These three circuits are examined in this assignment, in which the input and output signals are compared in regard to their phase difference and the overall gain of the circuit.

PROCEDURE

Step 1: The Inverting Amplifier

- For Fig.1, calculate the nearest preferred value of R_F that will achieve a gain of 8 for the circuit.

R_F = ______

- Calculate the output voltage (Vout) for the circuit if Vin equals +0.5VDC.

Vout = ______

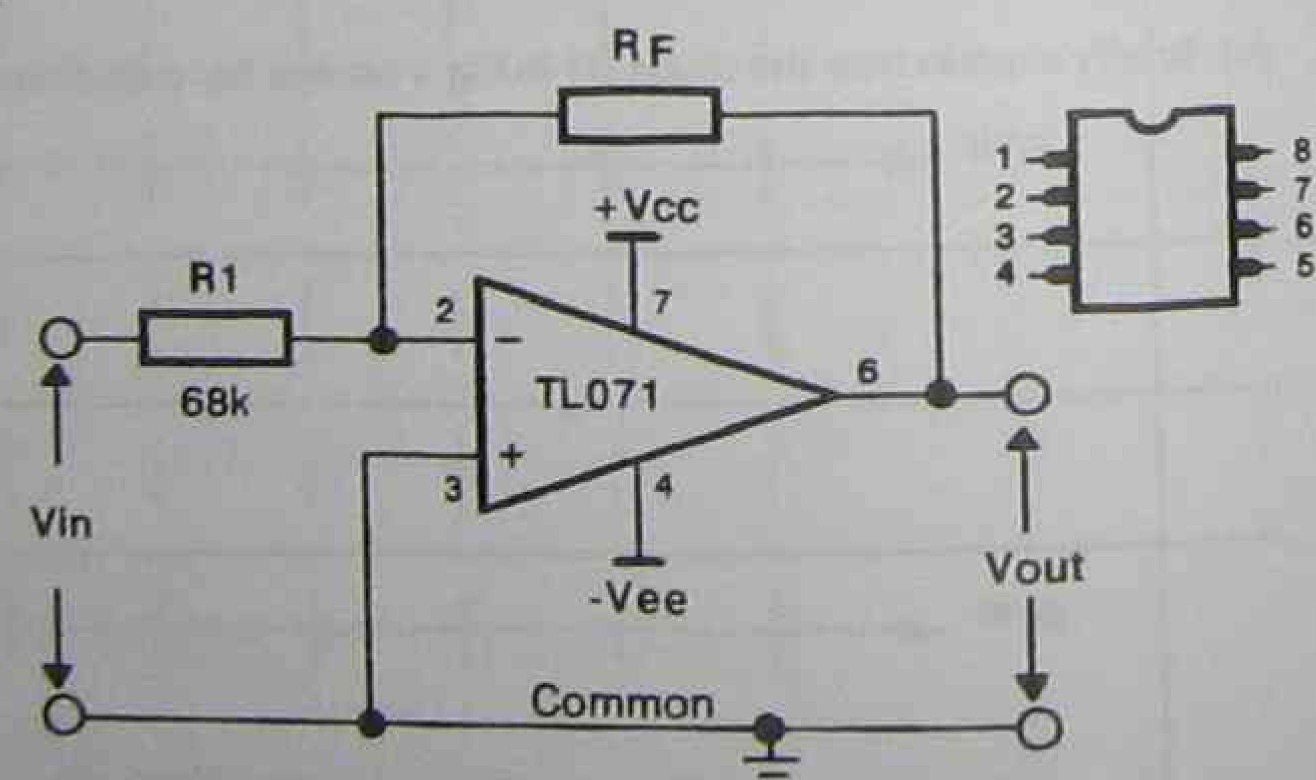


Fig.1: The inverting amplifier

- Using the calculated value of R_F, construct the circuit of Fig.1 on a prototyping board. The circuit is powered with a dual polarity power supply, either ±12V or ±15V.

- Using a variable DC power supply, apply an input signal (Vin) of +0.5VDC to the circuit. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of the circuit.

$V_{IN(MEAS)}$ = +0.5VDC; $V_{OUT(MEAS)}$ = ______; $A_{V(MEAS)}$ = ______

QUESTION:

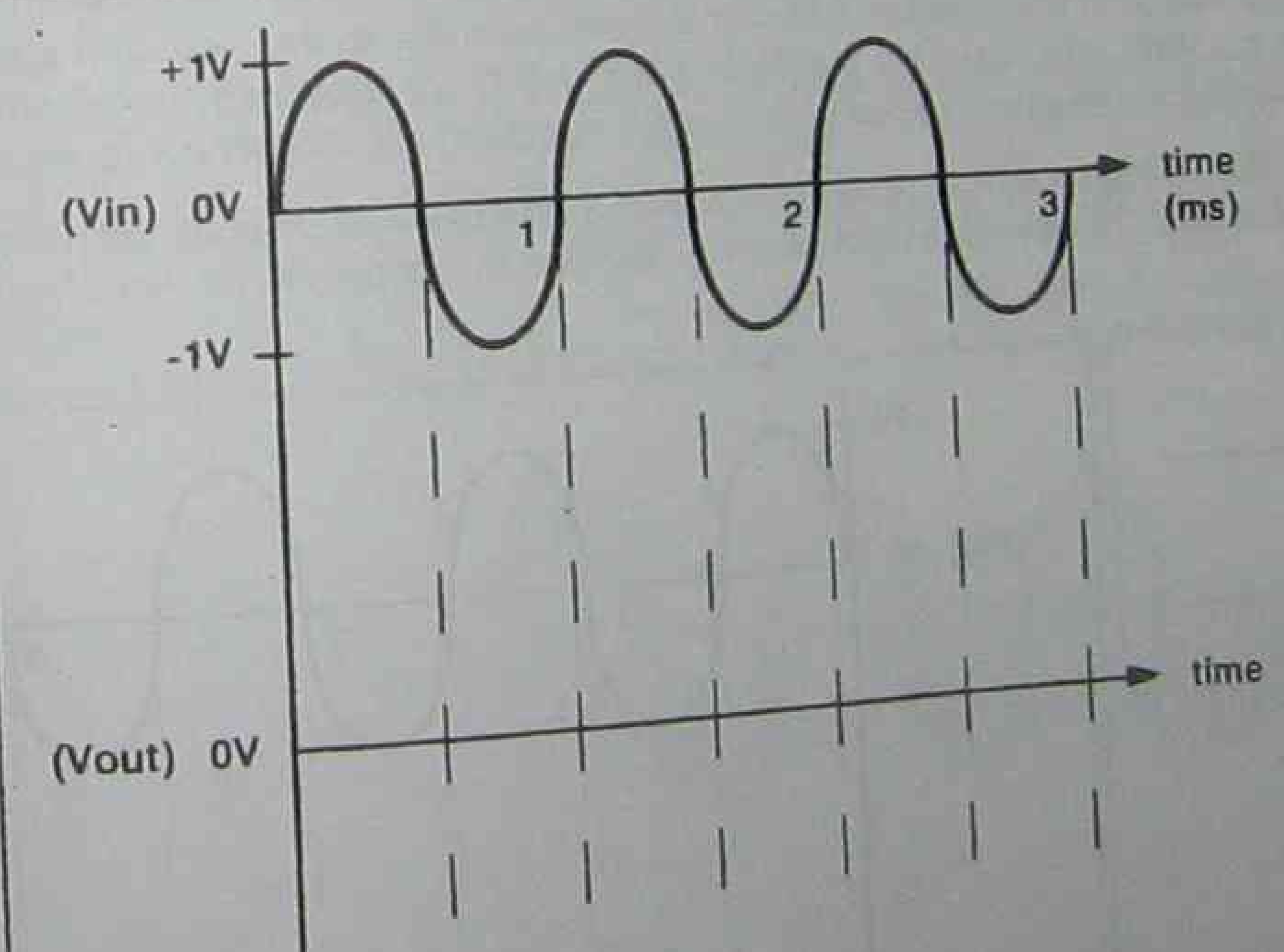
Briefly explain any difference between the calculated output voltage and the measured output voltage.

______

______

- Remove the DC voltage at Vin and connect a CRO with channel 1 to the input and channel 2 to the output of the amplifier. Connect a signal generator to the input and adjust it to give a 2Vp-p, 1kHz sine wave to the input (Vin). *NOTE: the input voltage must not contain any DC: if Minilabs are used for the signal source, adjust the DC offset so that the output voltage swings evenly around zero volts.*

- Record the input and output waveforms on the axes below.



Waveforms for VIN set to 1kHz, 2Vp-p - inverting amplifier

- Calculate the measured voltage gain (Av) and state the phase relationship (Ø) of the output to the input voltage.

Av = ______

Ø = ______

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 5

OPERATIONAL AMPLIFIER CIRCUITS - 1

AIM: To examine the operation of the inverting, non-inverting and voltage follower op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	22k, 10k, 68k, 100k, all ¼W
Semiconductors:	TL071 op amp IC
Equipment:	dual trace CRO, DVM, variable power supply, breadboarding system.

INTRODUCTION

The open loop gain of an op amp can be reduced to practical values by applying negative feedback. The two basic configurations are the inverting amplifier and the non-inverting amplifier. The voltage follower is a form of non-inverting amplifier, but with a gain of unity. These three circuits are examined in this assignment, in which the input and output signals are compared in regard to their phase difference and the overall gain of the circuit.

PROCEDURE

Step 1: The Inverting Amplifier

- For Fig.1, calculate the nearest preferred value of RF that will achieve a gain of 8 for the circuit.

RF = ______

- Calculate the output voltage (Vout) for the circuit if Vin equals +0.5VDC.

Vout = ______

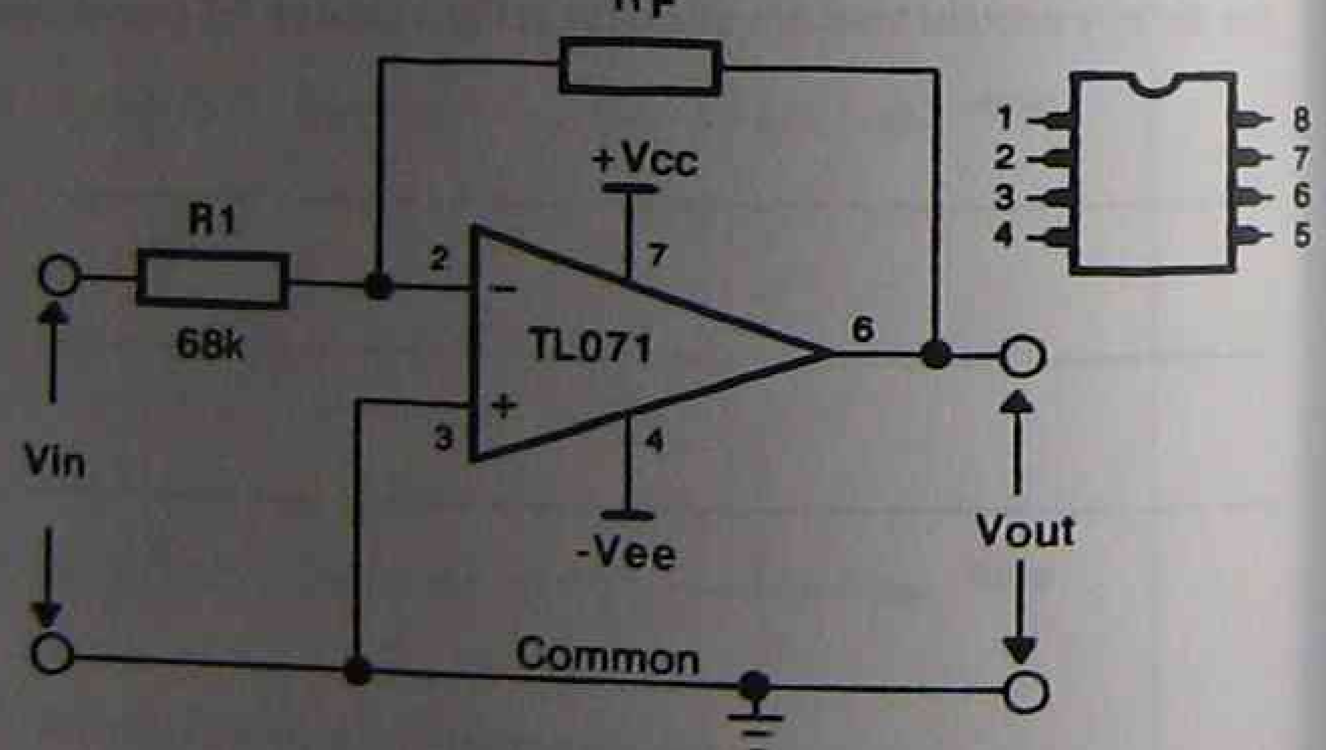


Fig.1: The inverting amplifier

- Using the calculated value of RF, construct the circuit of Fig.1 on a prototyping board. The circuit is powered with a dual polarity power supply, either ±12V or ±15V.

- Using a variable DC power supply, apply an input signal (Vin) of +0.5VDC to the circuit. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of the circuit.

VIN(MEAS) = +0.5VDC; VOUT(MEAS) = ______; AV(MEAS) = ______

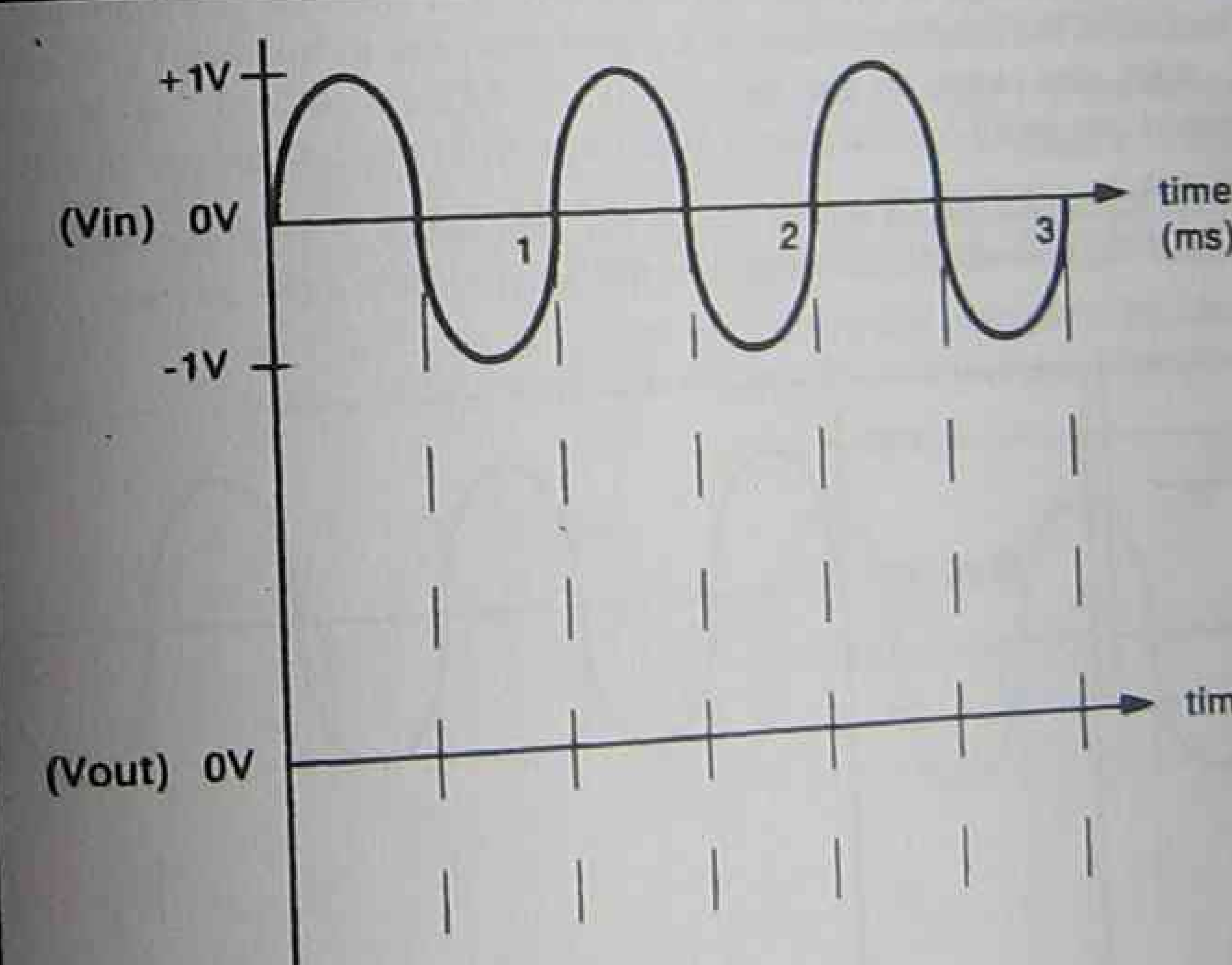
QUESTION:

Briefly explain any difference between the calculated output voltage and the measured output voltage.

______

______

- Remove the DC voltage at Vin and connect a CRO with channel 1 to the input and channel 2 to the output of the amplifier. Connect a signal generator to the input and adjust it to give a 2Vp-p, 1kHz sine wave to the input (Vin). *NOTE: the input voltage must not contain any DC: if Minilabs are used for the signal source, adjust the DC offset so that the output voltage swings evenly around zero volts.*
- Record the input and output waveforms on the axes below.



Waveforms for VIN set to 1kHz, 2Vp-p - inverting amplifier

- Calculate the measured voltage gain (Av) and state the phase relationship (Ø) of the output to the input voltage.

Av = ______

Ø = ______

Step 2: The Non-Inverting Amplifier

- For Fig.2, calculate the voltage gain of the circuit and the output voltage if Vin equals +0.5VDC

Av = ______

Vout = ______

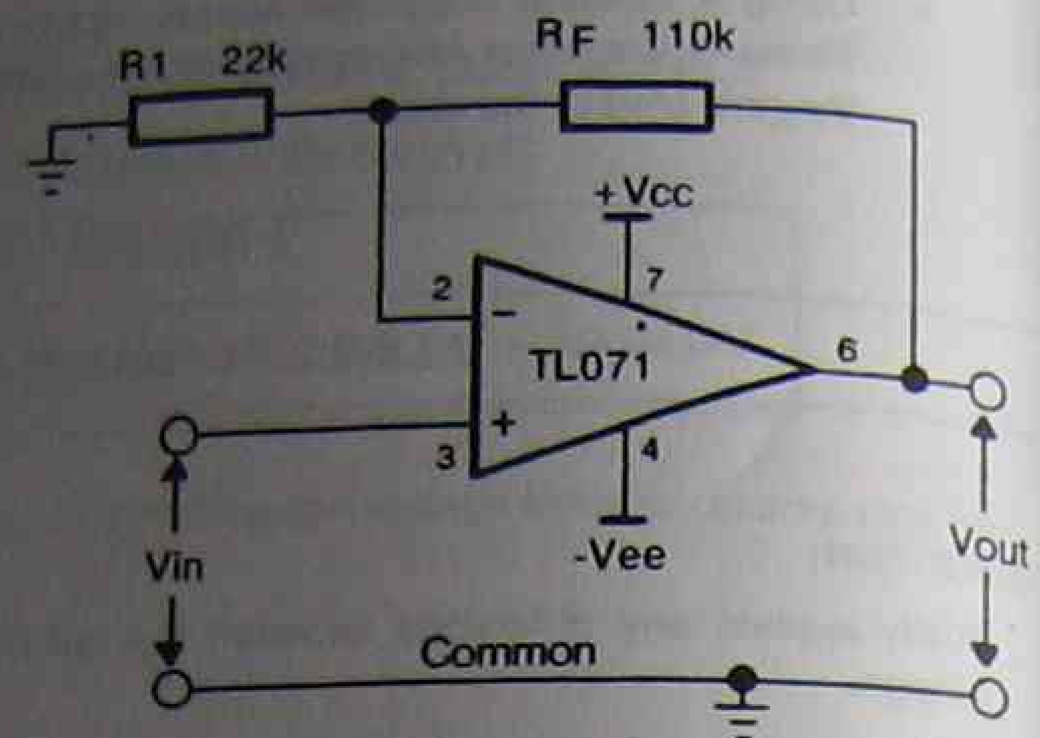


Fig.2: Non-inverting amplifier

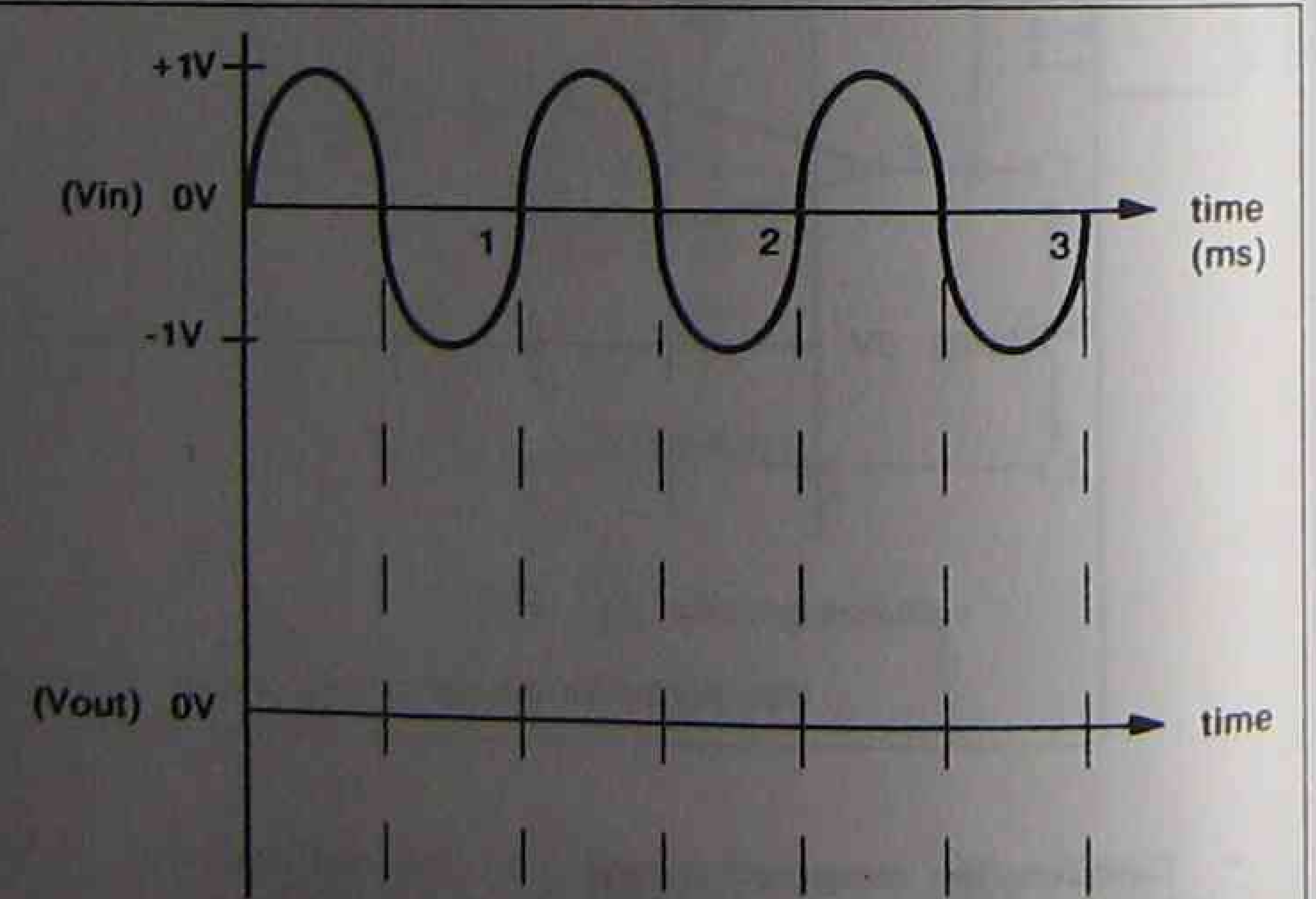
- Construct the circuit of Fig.2. Use a 100k and a 10k resistor connected in series for the 110k resistor if this value is not available.
- Apply an input signal (Vin) of +0.5VDC to the circuit. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of the circuit.

VIN(MEAS) = +0.5VDC; VOUT(MEAS) = ______; AV(MEAS) = ______

- Remove the DC voltage at Vin and connect a CRO with channel 1 to the input and channel 2 to the output of the amplifier. Connect a signal generator to the input and adjust it to give a 2Vp-p, 1kHz sine wave to the input (Vin). *NOTE: as before, if Minilabs or their equivalent are used for the signal source, adjust the DC offset to zero.*
- Record the input and output waveforms on the axes below. Also state the phase relationship (Ø) between the waveforms and determine the gain (Av) of the circuit (Vout/Vin).

Av = ______

Ø = ______



Waveforms for VIN set to 1kHz, 2Vp-p: non-inverting amplifier

Step 3: Voltage Follower

- Connect the circuit of Fig.3.
- Apply an input voltage of +2VDC to the non-inverting input (Vin).
- Measure the value and polarity of the output and determine the gain of the circuit (Av).

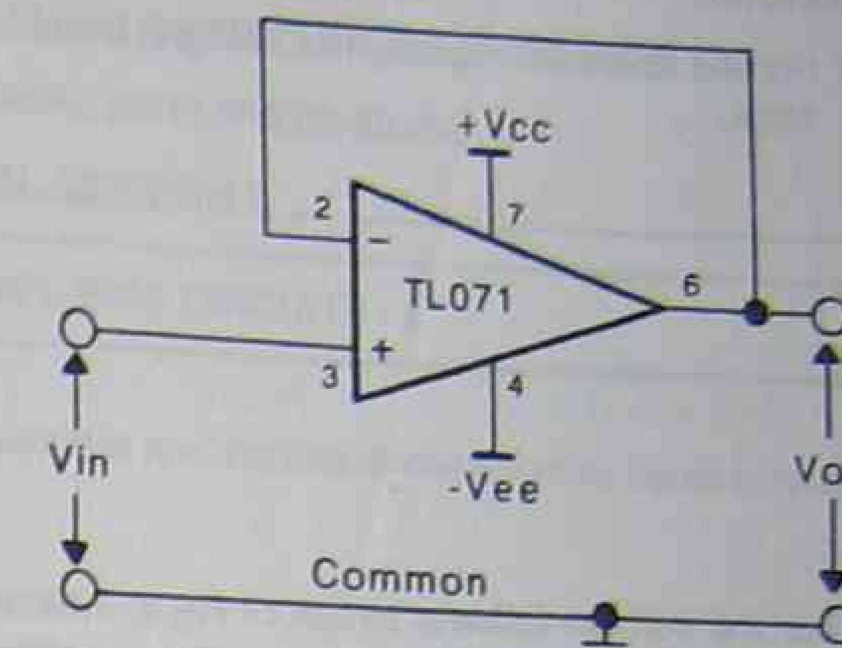


Fig.3: Voltage follower

VIN(MEAS) = +2VDC; VOUT(MEAS) = ______; AV(MEAS) = ______

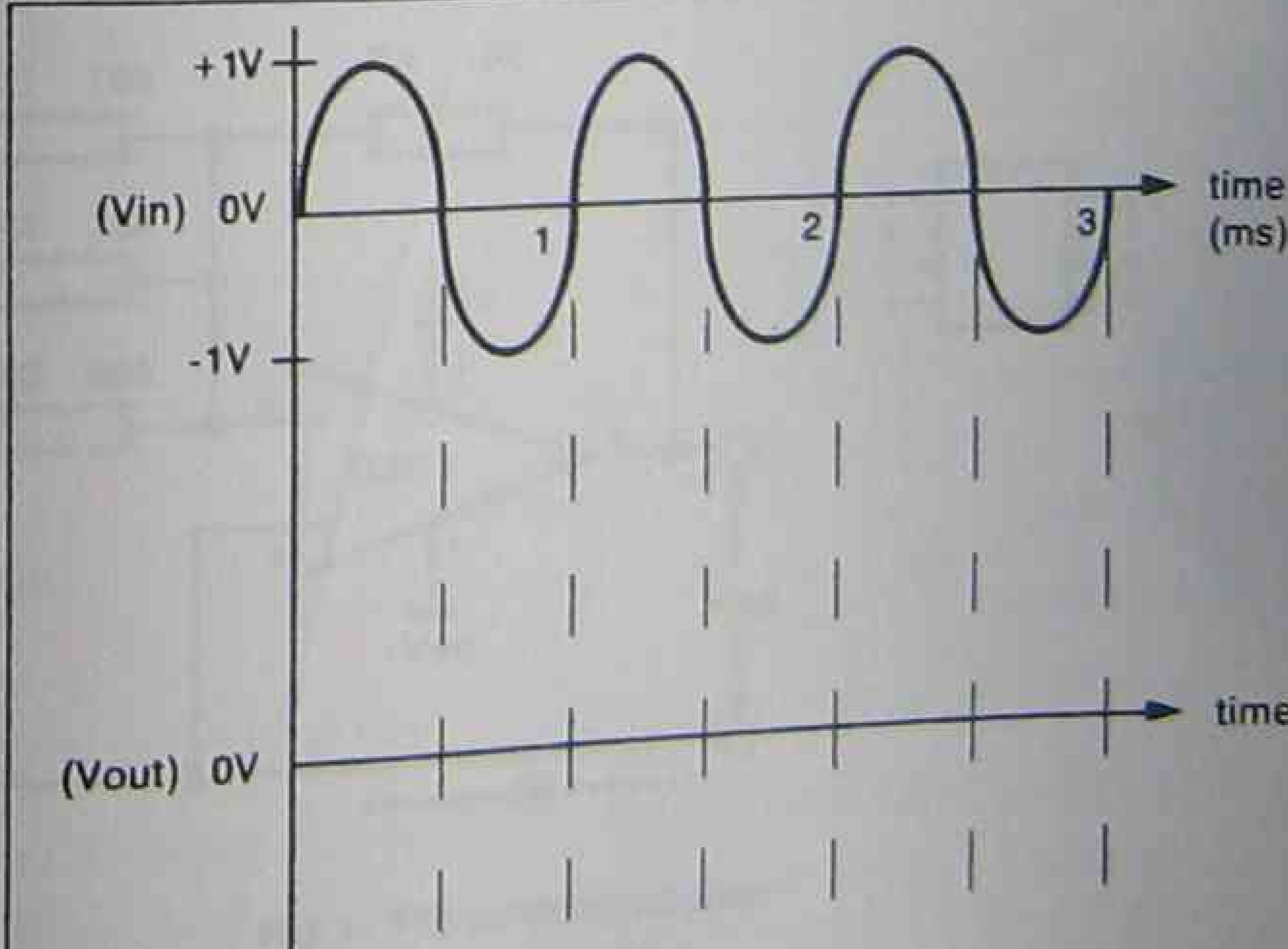
- Apply an input voltage of -2VDC to the non-inverting input. Measure the value and polarity of the output and determine the gain of the circuit (Av).

VIN(MEAS) = -2VDC; VOUT(MEAS) = ______; AV(MEAS) = ______

- Remove the DC voltage at Vin and connect a CRO with channel 1 to the input and channel 2 to the output of the amplifier. Connect a signal generator to the input and adjust it to give a 2Vp-p, 1kHz sine wave to the input (Vin). *NOTE: as before, if Minilabs or their equivalent are used for the signal source, adjust the DC offset to zero.*
- Record the input and output waveforms on the axes below. Also state the phase relationship (Ø) between the waveforms and determine the gain (Av) of the circuit (Vout/Vin).

Av = ______

Ø = ______



Waveforms for VIN set to 1kHz, 2Vp-p: voltage follower

QUESTIONS

(a) For the non-inverting amplifier of Fig.2, calculate the theoretical voltage gain (Av) if resistor R_F is 100k.

____

____

(b) For the voltage follower circuit of Fig.3, indicate if the measured gain equalled the theoretical gain of unity, and give brief reasons for any differences.

____

____

____

(c) For all three circuits, assuming a supply voltage of ±12V, state the maximum allowable value of the input voltage (Vin) for:

- a DC input signal: = ±_______V
- an AC input signal: = _______Vp-p

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 7

OPERATIONAL AMPLIFIER CIRCUITS - 2

AIM: To examine the operation of the inverting summer and the differential amplifier op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	27, 100, 470, 2 x 10k, 18k, 33k, 68k, 2 x 82k, all ¼W
Semiconductors:	TL071 op amp IC
Equipment:	dual trace CRO, DVM, variable power supply, breadboarding system.

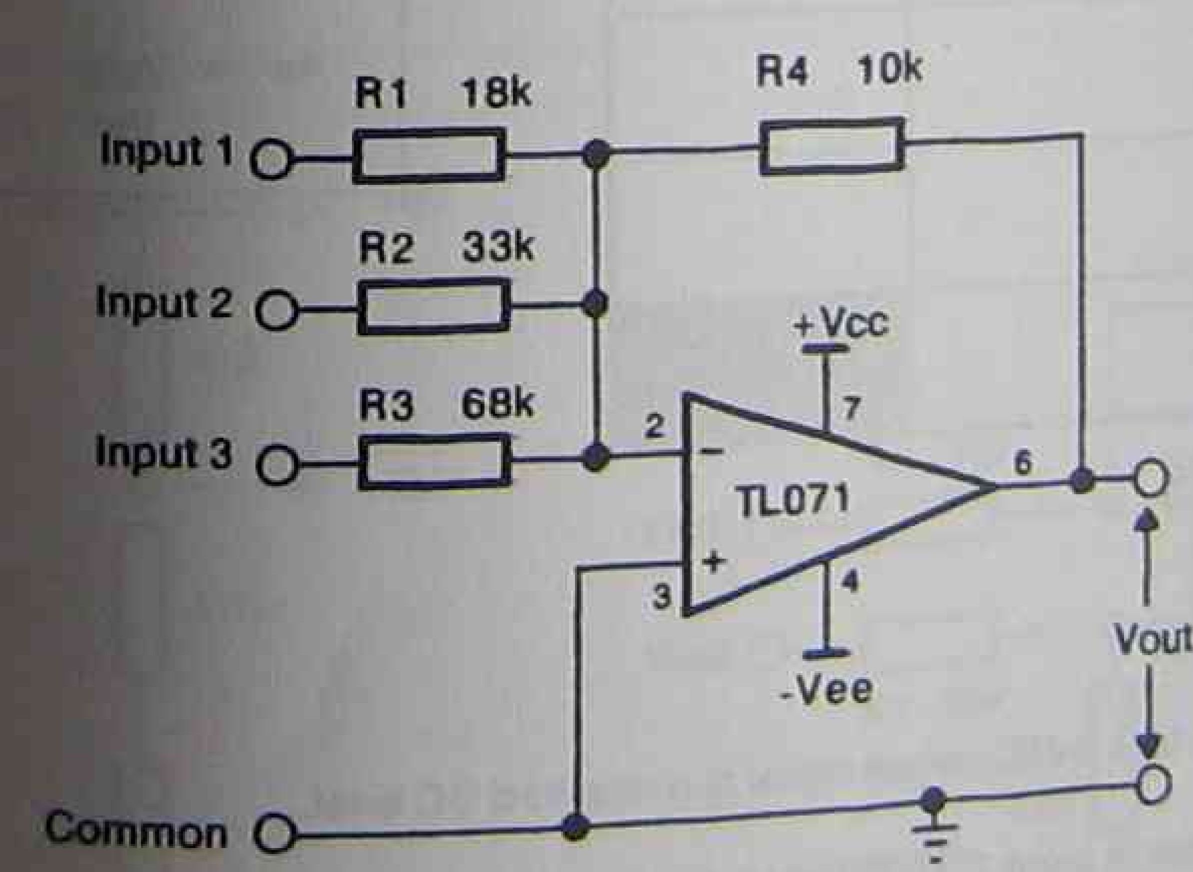
INTRODUCTION

Summing amplifiers are used to combine signals onto a common output. The input signals can be either AC or DC, and each channel can have a gain different to the others. Typical applications include adding a DC component to an AC signal and combining (mixing) several AC signals. In this assignment, the operation of the summer is examined along with summing an AC and a DC signal.

The operation of the discrete component differential amplifier has already been examined, and the op amp version is analysed in this assignment. In this application, the differential amplifier is used to produce an output proportional to the *difference* of two input signals.

PROCEDURE

Step 1: The Summing Amplifier



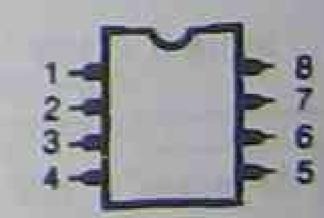


Fig.1: The summing amplifier

QUESTIONS

(a) For the non-inverting amplifier of Fig.2, calculate the theoretical voltage gain (Av) if resistor R_F is 100k.

____

____

(b) For the voltage follower circuit of Fig.3, indicate if the measured gain equalled the theoretical gain of unity, and give brief reasons for any differences.

____

____

____

(c) For all three circuits, assuming a supply voltage of ±12V, state the maximum allowable value of the input voltage (Vin) for:

- a DC input signal: = ±______ V
- an AC input signal: = ______ Vp-p

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 7

OPERATIONAL AMPLIFIER CIRCUITS - 2

AIM: To examine the operation of the inverting summer and the differential amplifier op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	27, 100, 470, 2 x 10k, 18k, 33k, 68k, 2 x 82k, all ¼W
Semiconductors:	TL071 op amp IC
Equipment:	dual trace CRO, DVM, variable power supply, breadboarding system.

INTRODUCTION

Summing amplifiers are used to combine signals onto a common output. The input signals can be either AC or DC, and each channel can have a gain different to the others. Typical applications include adding a DC component to an AC signal and combining (mixing) several AC signals. In this assignment, the operation of the summer is examined along with summing an AC and a DC signal.

The operation of the discrete component differential amplifier has already been examined, and the op amp version is analysed in this assignment. In this application, the differential amplifier is used to produce an output proportional to the *difference* of two input signals.

PROCEDURE

Step 1: The Summing Amplifier

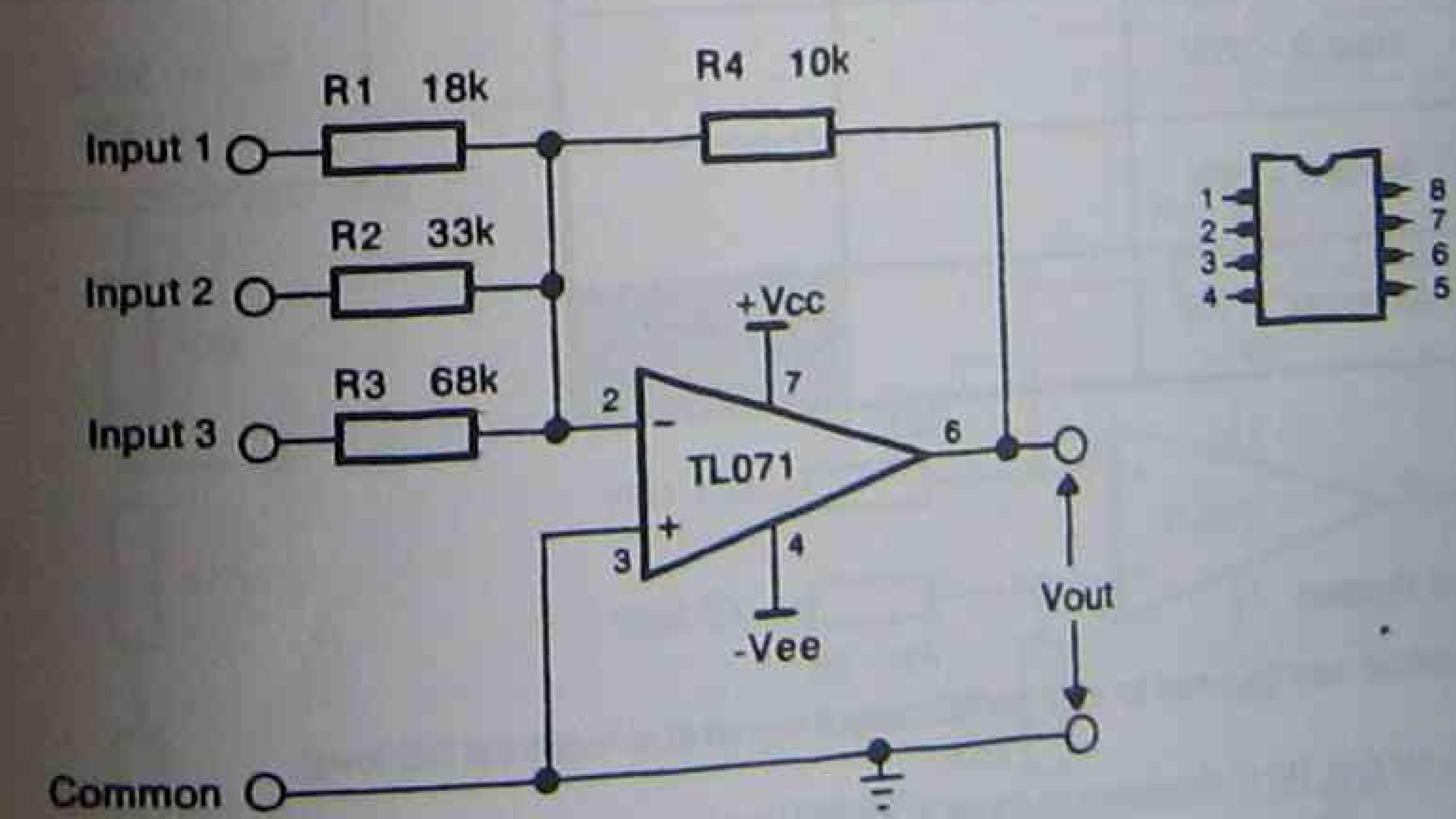


Fig.1: The summing amplifier

- For Fig.1, calculate the gain of each input.

Av1 = ______ Av2 = ______ Av3 = ______

- Calculate the DC output voltage of Fig.1 if +10V DC is applied to all three inputs.

Vout (all inputs = +10V) = ______

- Construct the circuit of Fig.1 on a prototyping board. The circuit is powered with a dual polarity power supply, either ±12V or ±15V.
- Connect inputs 2 and 3 to the common line and apply +10V DC to input 1. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of input channel 1. Record the results in Table 1.
- Connect inputs 1 and 3 to the common line and apply +10V DC to input 2. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of input channel 2. Record the results in Table 1.
- Connect inputs 1 and 2 to the common line and apply +10V DC to input 3. Measure the value and polarity of the output voltage (Vout). From these values, determine the gain (Av) of input channel 3. Record the results in Table 1.
- Connect all inputs to +10V. and record the resulting output voltage

Table 1

Input	Output	Gain (Av)
Input 1 = 10V		
Input 2 = 10V		
Input 3 = 10V		
All inputs = 10V		

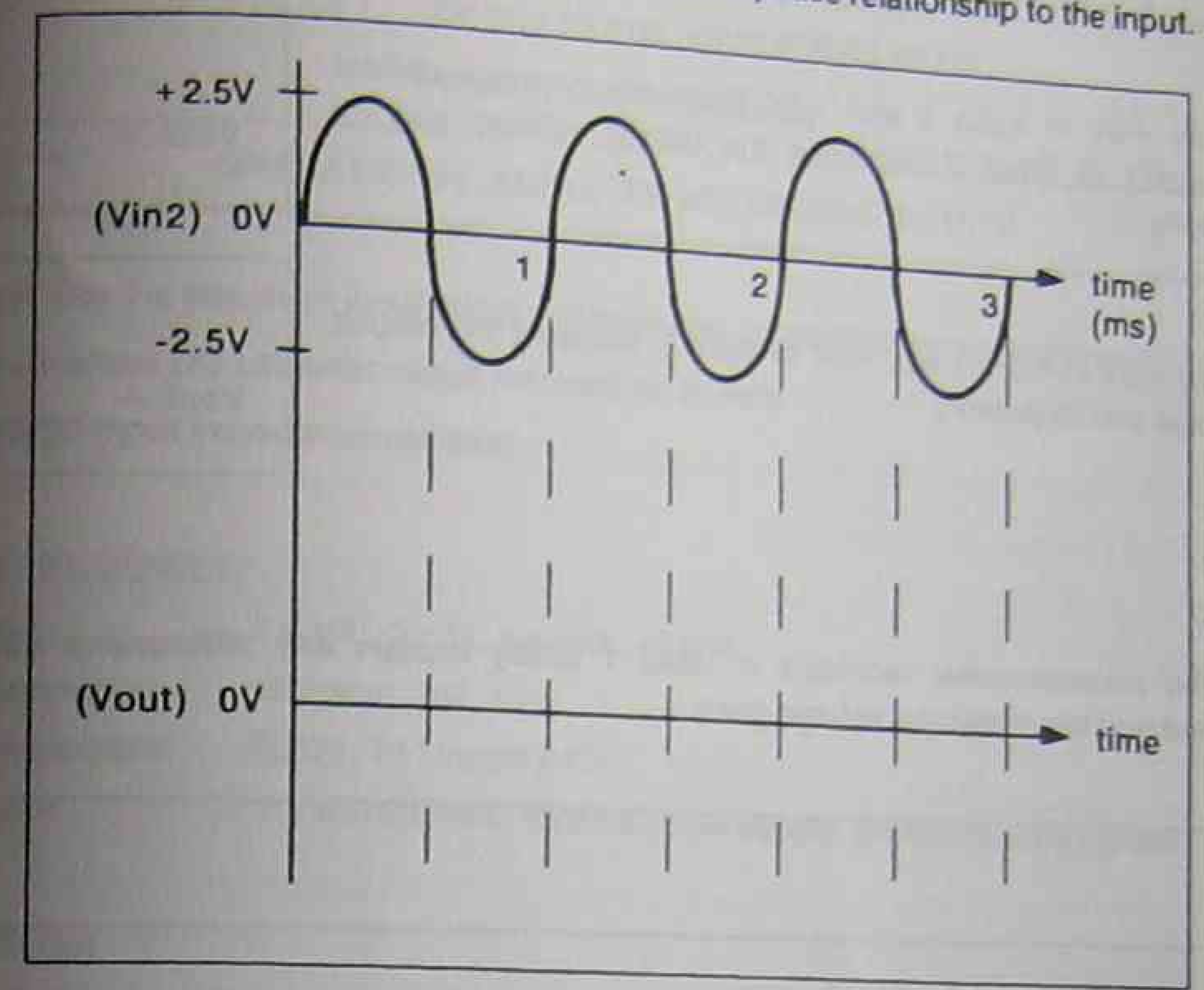
$$Av = \frac{Vout}{Vin}$$

Step 2: Biased Output

The summing amplifier can be used to bias an AC output signal at a required DC level.

- Apply a 5V p-p,1kHz sinewave to input 2, and +10V DC to input 1. Short input 3 to ground.
- Connect channel 1 of a CRO to input 2 and channel 2 of the CRO to the output. **Note**: Ensure that there is no DC component in the output of the signal generator. If using a BWD Minilab, use the DC offset control to centre the sinewave around zero volts.

- Sketch the input and output waveforms on the axes below. Record both the AC and DC components of the output waveform and show its phase relationship to the input.



Step 3: Differential Amplifier:

The operational amplifier can also be used to subtract two input signals and amplify the difference between them.

- Calculate the gain of the circuit of Fig.2 and determine the output voltage if V_{IN1} = +6V and V_{IN2} = +5V.

Av = ______

Vout = ______

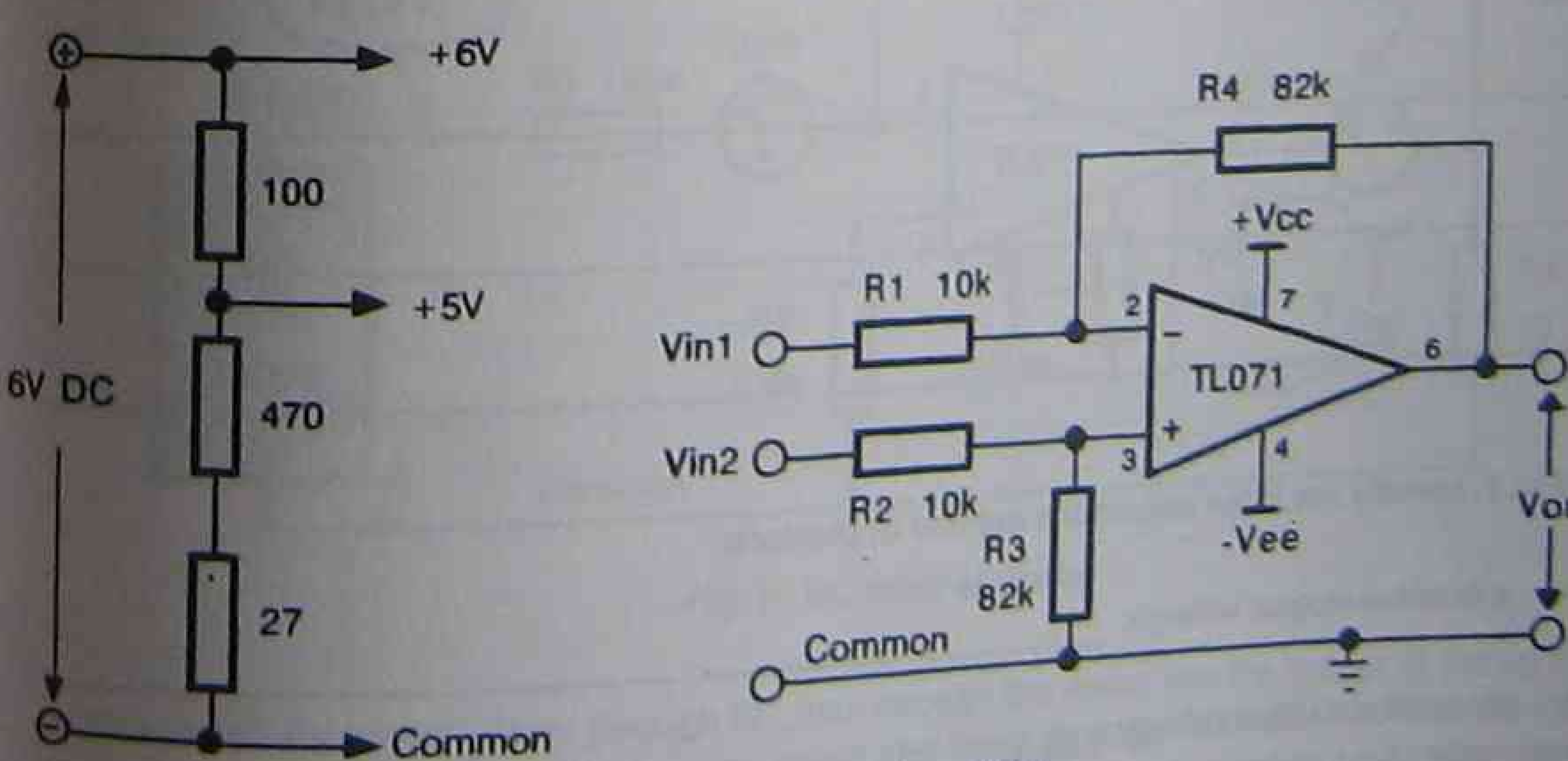


Fig.2: Differential amplifier

- Construct the circuit of Fig.2 including the voltage divider shown. This network is used to derive +5V and is connected to a DC power supply adjusted to 6V.

- Apply +6V to input 1 and +5V (from the potential divider network) to input 2. Measure the output voltage and its polarity.

Vout = ______

- Apply +5V to input 1 and +6V to input 2. Measure the output voltage and its polarity.

Vout = ______

QUESTIONS

(a) Using the measurements recorded in Table 1 briefly explain any differences between the calculated and the measured voltage gains.

______

______

______

(b) From the results of Table 1, add the output voltages obtained for each individual input connected to +10V. Compare this to the output voltage when all inputs are connected to +10V.

______

______

(c) With reference to the waveforms obtained from Fig.1, briefly explain why the output waveform has a DC component. Also comment on the polarity of the AC component compared to the input signal.

______

______

______

(d) For Fig.2, identify the input conditions needed to produce:

- a positive output voltage: ______
- a negative output voltage:

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 8

OPERATIONAL AMPLIFIER METERING CIRCUITS

AIM: To examine the effects of input offset and input bias currents in the applications of:

(a) a sensitive DC ammeter circuit that uses an op amp

(b) a high input impedance voltmeter

EQUIPMENT REQUIRED:

Resistors:	2 x 100, 2 x 1k, 2 x 100k, all ¼W
Potentiometers:	150 ohm, 10k, 100k
Semiconductors:	TL071, 741 op amp ICs
Equipment:	2 x multimeters, variable power supply, breadboarding system.

INTRODUCTION

Practical limitations of op amps require additional external circuitry around the op amp to provide compensation for bias currents and offsets. The smaller the input voltage or current, then the greater will be the need for compensation. In this practical session offsets will be observed in an op amp driven DC ammeter and a DC voltmeter. Comparisons will be made between the general purpose 741, which has BJT inputs, and the TL071 (or LF 411), which has JFET inputs.

The DC Micro- Ammeter

The circuit of Fig.1 uses an op amp to drive a 1mA meter. RF and Rs are used to 'scale' the circuit. The purpose is to create an accurate, sensitive ammeter with ranges of 10μA and 100μA while using the 1mA meter movement.

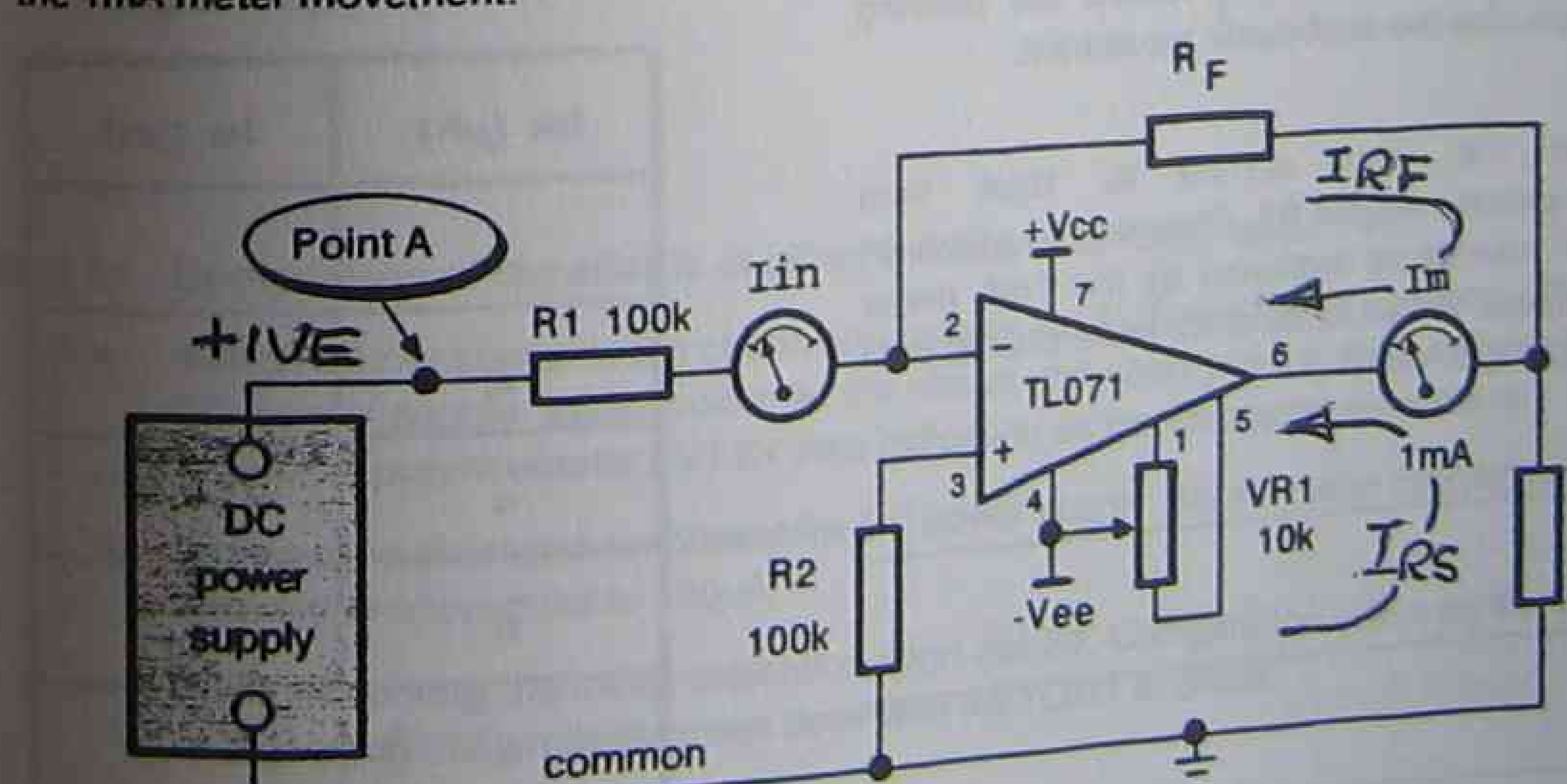


Fig.1: DC micro-ammeter

In this circuit IIN (or IRF) flows through RF, then through the meter into the output of the op amp. Rs ... which also flows through the meter and into the output of ... current of 1mA, when the circuit

The relationship of the full scale input current I_{IN} to the full scale meter current I_M is:

- $I_M = I_{IN}(1 + \frac{R_F}{R_S})$

This equation can be rearranged for calculating R_F for a particular range:

- $R_F = R_S(\frac{I_M}{I_{IN}} - 1)$

R2 is used to minimise the effects of input offset current and the offset null potentiometer VR1 is used to cancel the effect of input offset voltage.

Step 1: Practical Procedure 10μA Range

- Calculate the value of R_F needed in Fig.1 to cause a 10μA input current to produce full scale deflection on the 1mA meter.

R_F = ________

Value for a 10μA ammeter

- Adjust a 100k potentiometer to the calculated value of R_F. Construct the circuit of Fig.1, using the adjusted potentiometer for R_F. Keep wiring neat and as short as practicable. The 1mA meter movement can be a multimeter set to its 1mA DC current range. The ammeter used to measure I_{IN} is a multimeter set to a range that can read 10μA.
- Short POINT A to common and adjust the offset null pot VR1 to zero the 1mA meter (when I_{IN} is zero, I_M should be zero). NOTE: The power supply is not connected at this stage.
- Remove the short and connect the DC power supply.
- Adjust the power supply to give each value of I_{IN} listed in Table 1 and record the reading indicated by the 1mA meter movement.
- Adjust the input current to 10μA then short-circuit resistor R2. Observe the effect on meter current as indicated by the 1mA meter movement. Is the value of R2 more critical than the setting of VR1?

____

____

____

____

Table 1 - 10μA range

I_{IN} (μA)	I_M (μA)
0	
2	
4	
6	
8	
10	

Step 2: Practical Procedure 100μA Range:

- Calculate the value of R_F required to give a full scale deflection on the 1mA meter movement for an input current of 100μA.

R_F = ________

Value for a 100μA ammeter

- Adjust a 10k pot to the calculated value of R_F then, with the power to the circuit switched off, connect it in place of R_F.
- Disconnect the DC power supply and short point A to ground. Apply power to the circuit, and adjust the offset pot VR1 to give zero deflection on the 1mA meter movement.
- Remove the short, reconnect the DC power supply and adjust it to give the values for I_{IN} listed in Table 2. Complete Table 2 by recording the values indicated by the 1mA meter movement for each value of I_{IN}.

Table 2 - 100μA range

I_{IN} (μA)	I_M (μA)
0	
20	
40	
60	
80	
100	

Step 3: Comparison of the effects of offset between the TL071 and a 741

- Switch off the power to the circuit and replace the TL071 with a 741 op amp. Disconnect the DC power supply, apply power to the circuit and short point A to common. Re-adjust the offset null potentiometer RV1 for zero indication on the 1ma meter movement.
- Remove the short and reconnect the DC power supply. Set the meter current indicated by the 1mA meter movement to 100μA.
- While observing the meter indication short out R2. Compare the effect on the meter current to that when this procedure was done with the TL071 in circuit.

____

____

____

The DC Milli-Voltmeter

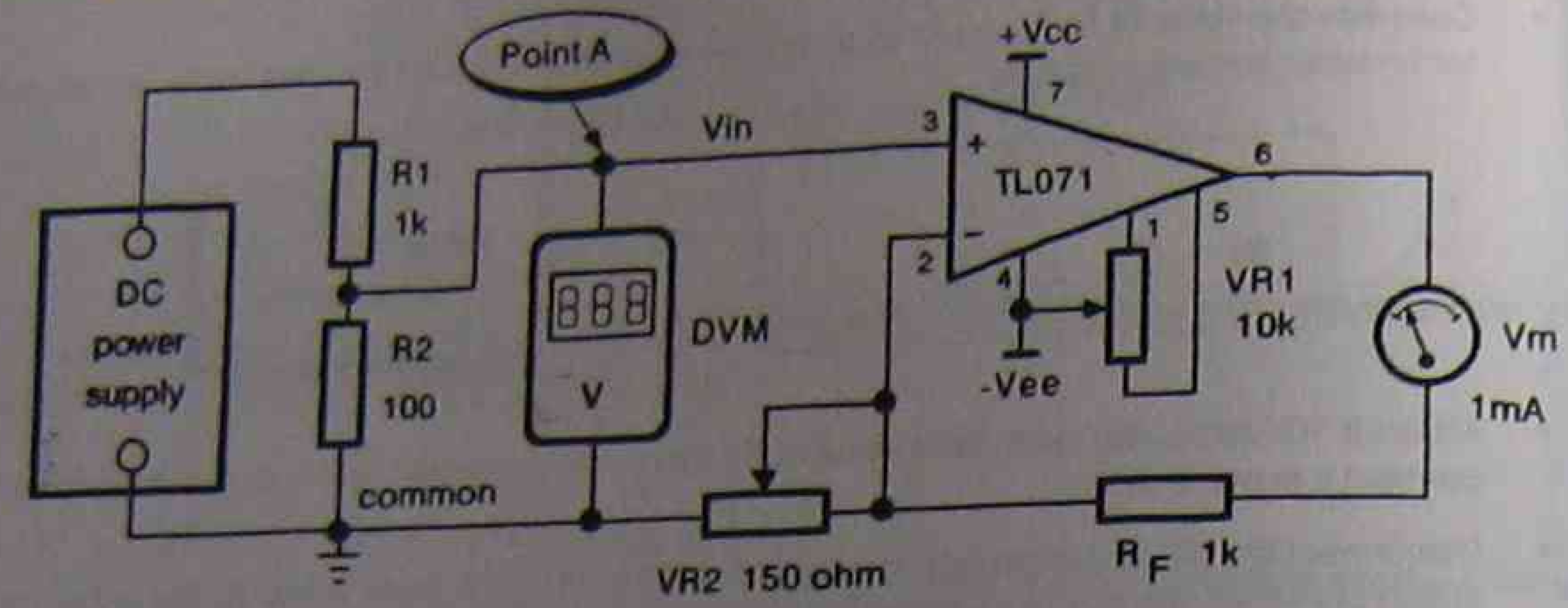


Fig.2: DC milli-voltmeter

Circuit Operation

A non-inverting op amp configuration is used to control the meter current by placing the meter in the feedback path. The input voltage V_{IN} will appear across VR2, causing a current to flow in VR2. The current in VR2 is supplied by the output of the op amp, and this current also flows through the meter movement. The value of VR2 therefore sets the value of the meter current. The value of R_F is not critical and provides protection against overload of the meter. Thus:

- □ $I_M = \frac{V_{IN}}{V_{R2}}$

Step 1: Practical Procedure

- Calculate the value of VR2 to allow a full scale deflection current of 1mA to flow through the meter when V_{IN} equals 100mV.

V_{R2} = ________

- Adjust VR2 to the calculated value then connect the circuit of Fig.2. Note that the non-inverting input (pin 3) is connected to the input voltage, derived from the potential divider formed by R1 and R2. Use a DVM to monitor V_{IN}.
- With the DC power supply set to 0V, connect point A to common and adjust the offset null pot VR1 to zero the meter.
- Set V_{IN} to 100mV and then carefully adjust VR2 so the 1mA meter is at full scale.
- By setting V_{IN} to the values specified in Table 3, complete Table 3.

Table 3 - 100mV range

V_{IN} (mV)	V_M (mV)
0	
20	
40	
60	
80	
100	

Step 2: Effects of Bias Currents and Offset

- Place a 100 ohm bias compensation resistor in series with the non-inverting input (pin 3) and the junction of R1 and R2. Set V_{IN} to zero, short point A to common and re-adjust the offset null pot VR1.
- Remove the short and set V_{IN} to 100mV. While observing the 1mA meter short out the bias compensation resistor and comment on the effect this has on the meter reading.

____

____

- Turn off the power to the circuit and replace the TL071 with a 741 op amp. Set to V_{IN} to zero, short point A to ground and re-adjust the offset null circuit.
- Remove the short and set V_{IN} to 100mV. While observing the 1mA meter short out the bias compensation resistor and comment on the effect this has on the meter reading. Compare it to the effect when this was done with the TL071

____

____

Questions

(a) Identify which of the two op amps has the larger input bias currents, and briefly explain why.

____

____

____

(b) Briefly explain why an offset null adjustment is important when the op amp is required to amplify small voltages.

____

____

____

(c) For the circuit of Fig.2, calculate the value of VR2 required to give full scale deflection for a 10mV input voltage.

____

****

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1 OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 9

OPERATIONAL AMPLIFIER CHARACTERISTICS

AIM: To measure the frequency response and the slew rate of a typical op amp. Also to observe the effect of slew rate limiting on a sine wave output from a typical op amp.

EQUIPMENT REQUIRED:

Resistors:	10k, 47k, 1M, all ¼W
Semiconductors:	741 and TL071 op amp IC
Equipment:	CRO, DVM, signal generator, power supply, breadboarding system.

INTRODUCTION

The internal construction of an op amp determines its frequency response and its slew rate. Most op amps are internally compensated to prevent instability and the compensation sets the frequency response of the device. The open loop frequency response of an op amp is usually very low, and is increased as the gain of the op amp is reduced by negative feedback. In this assignment, the frequency response of a 741 op amp is measured and the results are plotted on a graph. Slew rate is also a limiting factor on the range of frequencies and the amplitude of the output signal that an op amp can handle, and this is measured for both the 741 and the TL071.

PROCEDURE

Step 1: Frequency Response Of The 741

- For Fig.1, calculate the theoretical gain of the circuit.

Av = ______

- Calculate the output voltage (Vout) for the circuit if Vin equals +0.5VDC.
- Construct the circuit of Fig.1 on a prototyping board. The circuit is powered with a dual polarity power supply, either ±12V or ±15V.

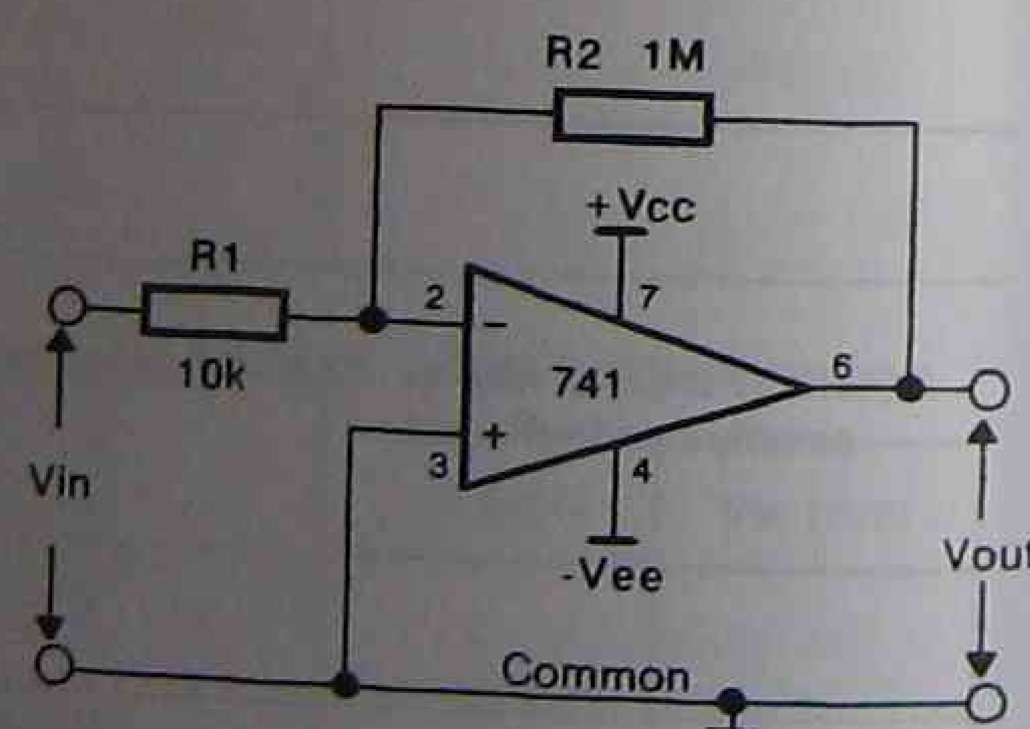


Fig.1

- Apply a DC voltage of -0.1V DC and measure the DC output voltage of the circuit. From this determine the actual voltage gain of the circuit.

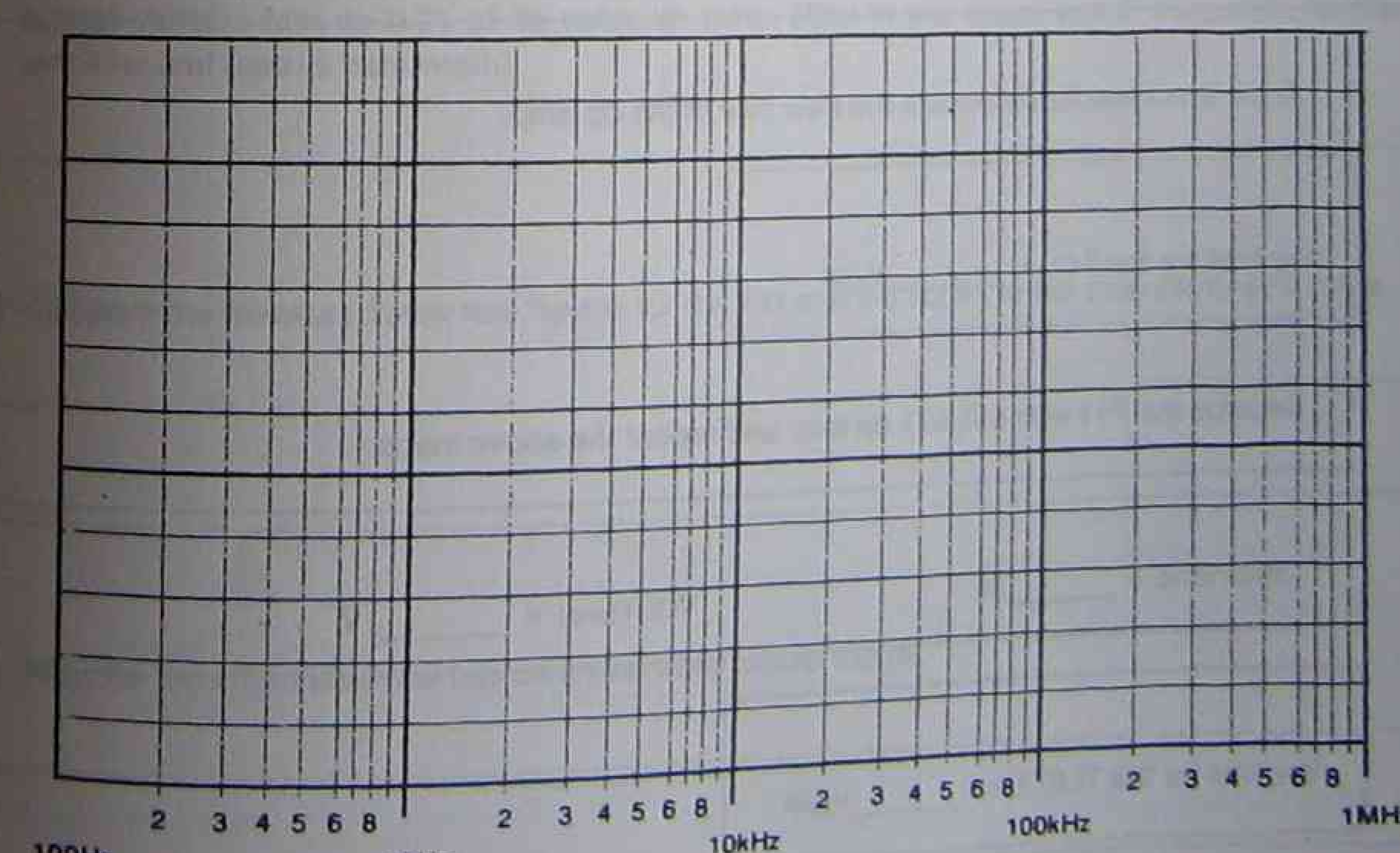
Vout = ______ V DC Av (measured) = ______

- Remove the DC voltage at Vin and connect a CRO with channel 1 to the input and channel 2 to the output of the amplifier. Connect a signal generator to the input and adjust it to give a 0.2Vp-p, 1kHz sine wave to the input (Vin). *NOTE: the input voltage must not contain any DC: If Minilabs are used for the signal source, adjust the DC offset so that the output voltage swings evenly around zero volts.*
- Record in Table 1 the output voltage for this frequency and calculate the gain of the circuit.
- For each of the frequencies listed in Table 1, measure and record the amplitude of the output voltage. Also determine and record the gain of the circuit for each frequency.

Table 1

Frequency	Vin	Vout	Gain (Av)
0Hz	-0.1VDC		
1kHz	0.2Vp-p		
10kHz	0.2Vp-p		
50kHz	0.2Vp-p		
100kHz	0.2Vp-p		
500kHz	0.2Vp-p		
frequency where Vout equals Vin	0.2Vp-p	0.2Vp-p	

- Plot voltage output versus frequency on the log-linear graph below.



Step 2: Slew Rate

Note: The slew rate specification for the 741 is 0.5V/μs and 13V/μs for the TL071.

- Change the circuit to that shown in Fig.2.
- Calculate the output voltage of the circuit for an input of 4Vp-p.

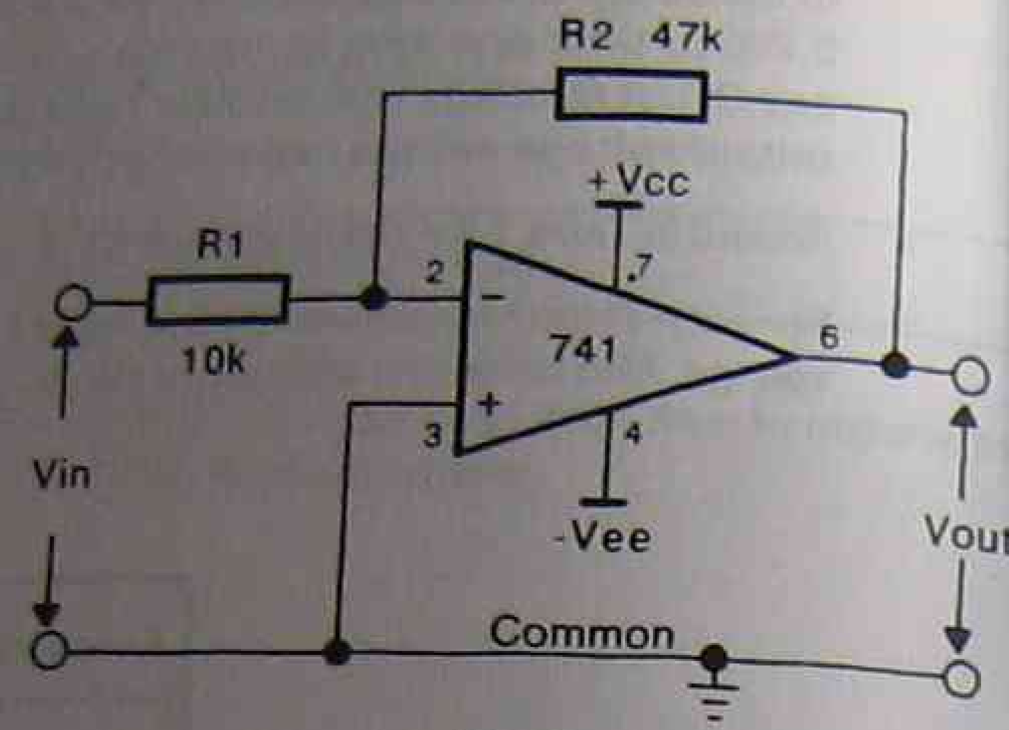
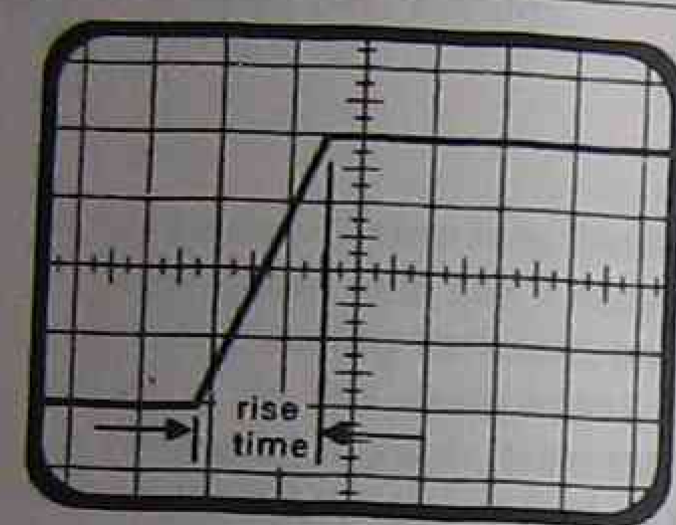


Fig.2

Vout = ______

To measure the rise time of the waveform adjust the CRO to obtain a display that is nearly the height of the screen adjusted so the leading edge is displayed. Then select the x5 magnification and readjust the CRO so the display is like that shown on the right. The rise time equals the distance in divisions multiplied by the (TIME/DIV setting divided by 5). For example, a setting of 50μs/division and a width of 1.2 divisions gives a rise time of 12μs.



- Measure the peak to peak voltage of the output waveform and record this and the rise time.

Rise time = ______ μs	VOUT(p-p) = ______ V

- From these results determine the slew rate of the op amp.

rise time for the 741 = ______ V/μs

- Replace the 741 with a TL071 op amp and repeat the above steps.

Rise time = ______ μs	VOUT(p-p) = ______ V

rise time for the TL071 = ______ V/μs

Step 3: Effects of Slew Rate

- Connect the circuit of Fig.3. Note that the circuit uses a 741 op amp.
- Calculate the maximum frequency for the circuit for each of the following input voltages, where:

$$f_{MAX} = \frac{\text{slew rate}}{6.28 \times V_{OMAX}}$$

slew rate (741) = 0.5V/μs

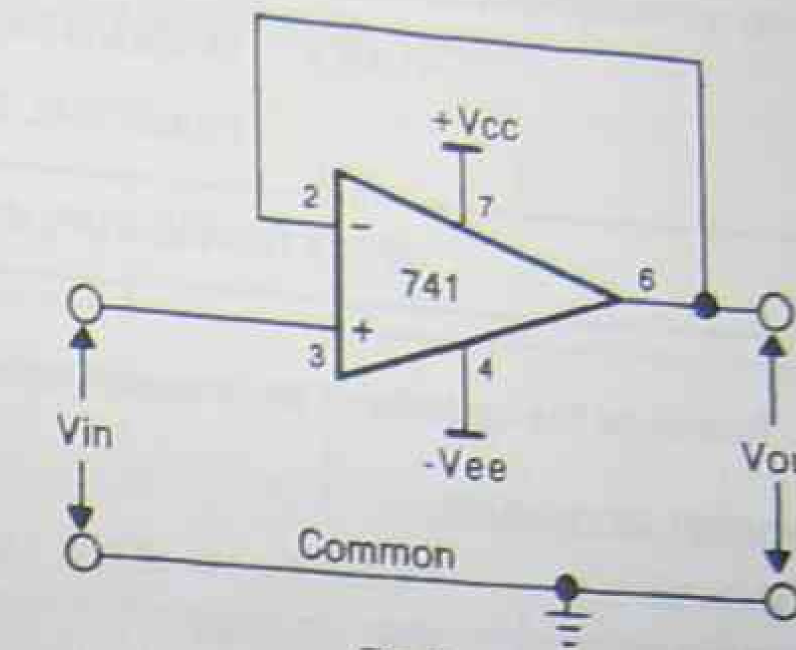


Fig.3

1. 2Vp-p f_{MAX} = ______
2. 10Vp-p f_{MAX} = ______
3. 5Vp-p f_{MAX} = ______

- Apply each of these voltage levels at a frequency around 90% of that calculated. Confirm that the output waveform has the same shape as the input by observing the input and output waveforms with both channels of a dual trace scope. Then increase the frequency until the output signal starts to distort. Record this frequency in Table 2 for each signal.

Table 2

Vin (= Vout)	f_{MAX} (measured)
2Vp-p	
5Vp-p	
10Vp-p	

QUESTIONS

(a) From the frequency response graph for the circuit of Fig.1, determine the frequency where the output voltage falls to 0.71 of its value at 1kHz. (This is the upper cutoff frequency for the amplifier and also its bandwidth).

______

(b) Compare the measured slew rate figures for the 741 and the TL071 to the manufacturer's value.

______

______

(c) Describe the effect slew rate has on a sine wave output signal.

______

______

ADVANCED CERTIFICATE IN APPLIED INDUSTRIAL ELECTRONICS

YEAR 1   OPERATIONAL AMPLIFIERS 6016C

PRACTICAL SESSION 10

OPERATIONAL AMPLIFIER DIODE CIRCUITS

AIM: To analyse the operation of the precision half wave rectifier and peak detector op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	1 x 4k7, 5 x 10k all ¼W ; 10k potentiometer
Capacitors:	10nF polyester
Semiconductors:	TL071 op amp, 2 x 1N4148 signal diodes
Equipment:	CRO, signal generator, DVM, breadboarding system.

INTRODUCTION

A precision rectifier can rectify small AC signals, as the op amp overcomes the 0.6V forward bias of the diodes. In this practical session, the half wave precision rectifier is examined, along with the peak detector. Both circuits use diodes, and the peak detector is used to give a measurement of the peak value of an AC waveform.

Step 1: Precision Half Wave Rectifier

- Construct the circuit of Fig.1. Then connect Vin to ground and adjust RV1 so that Vout - op amp is as close to 0V as possible. Use a DVM to measure Vout - op amp.

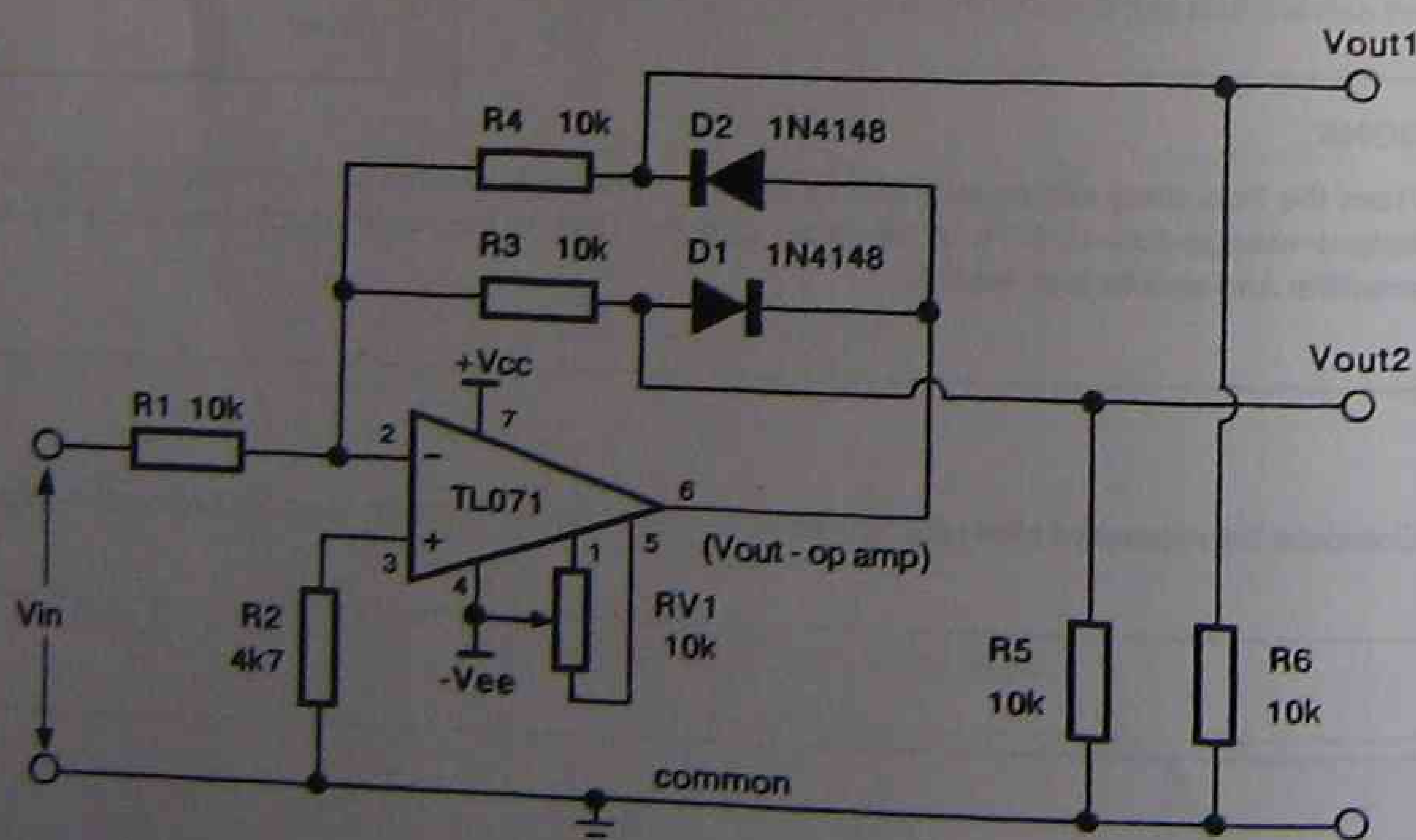
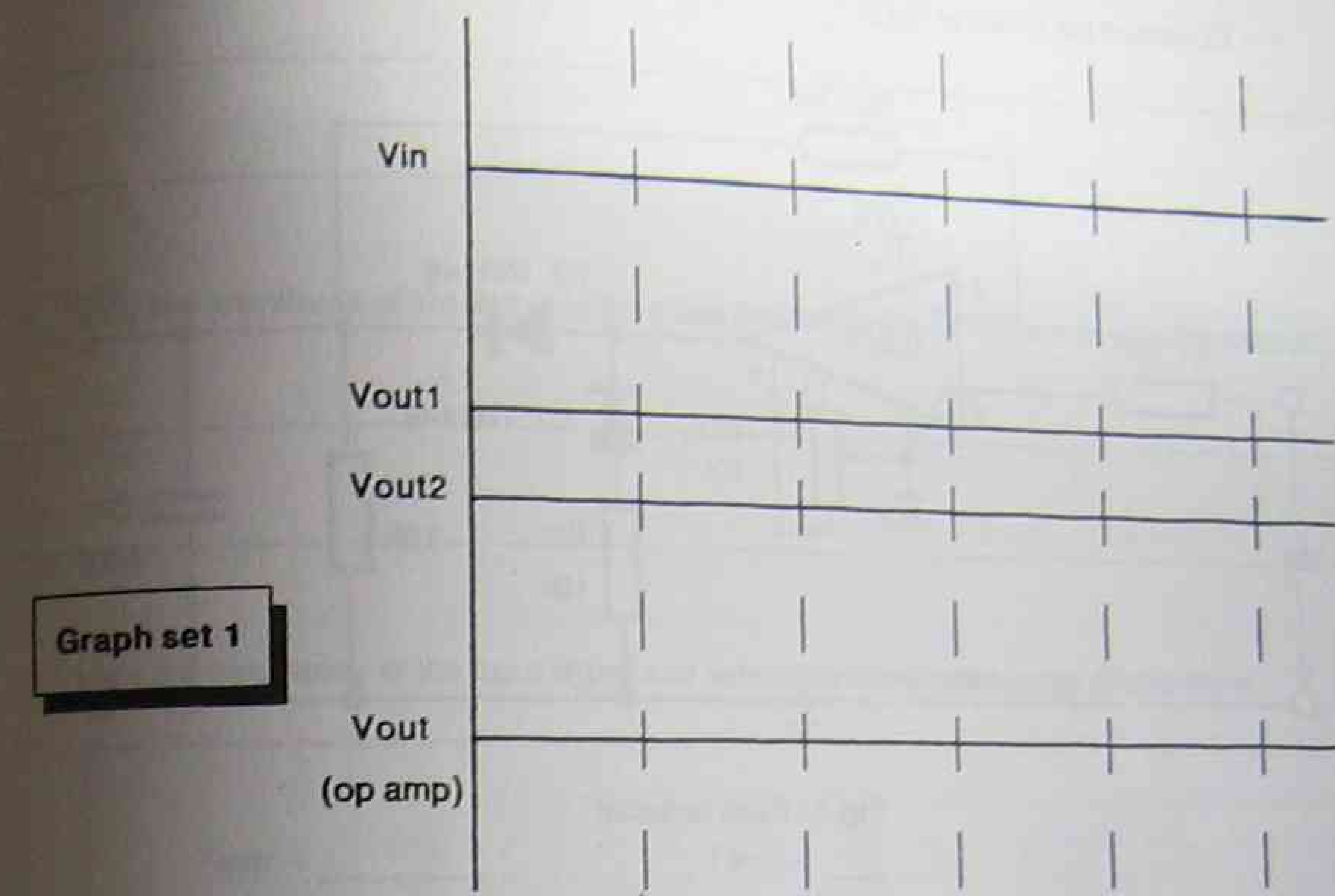


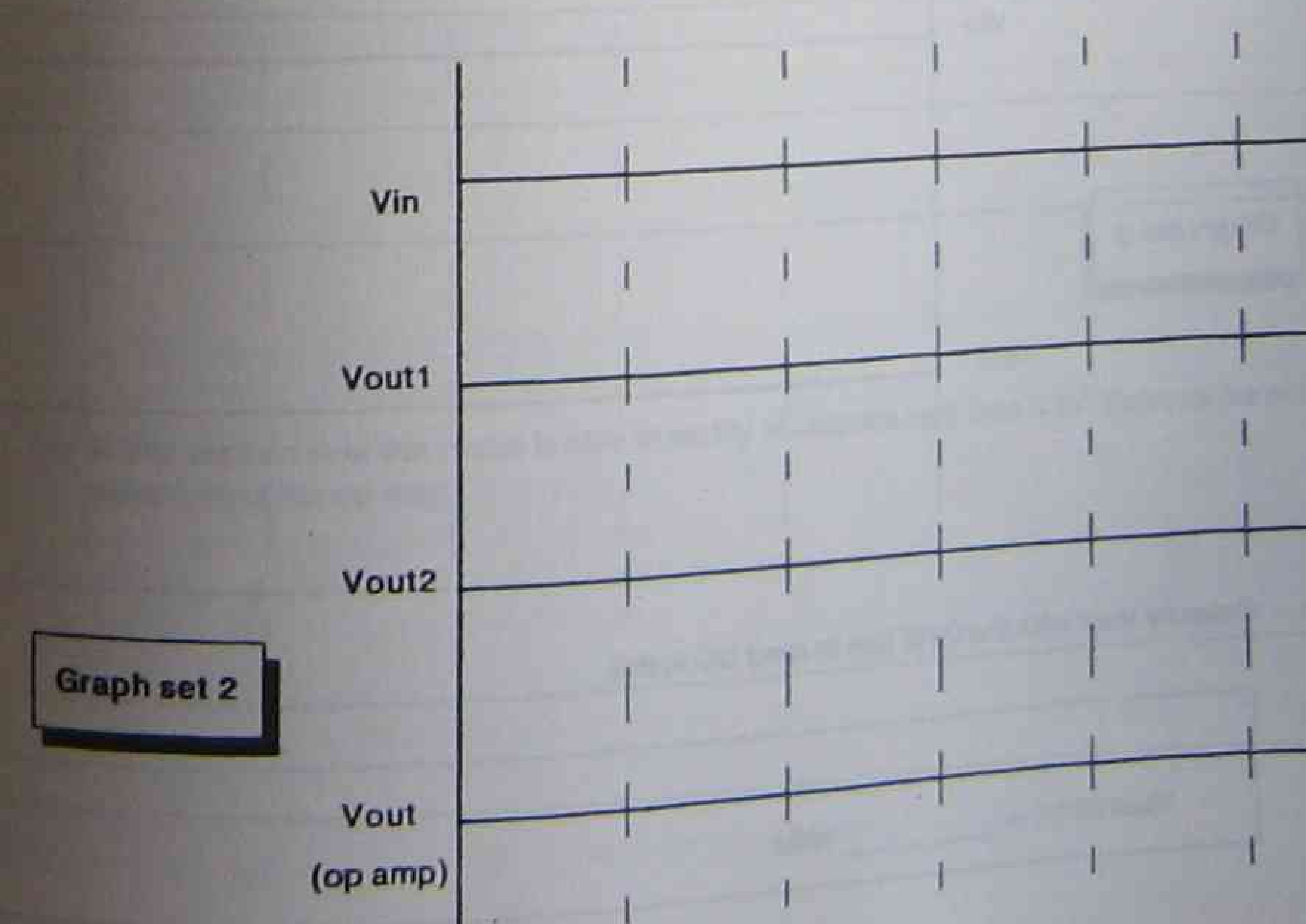
Fig.1: Precision half wave rectifier

- Disconnect Vin from ground and apply a 1kHz, 10Vp-p sinewave to Vin. Make sure there is no DC component in the input waveform.

- Connect channel 1 of a CRO to Vin and channel 2 to Vout1. Sketch the waveforms on the axes below showing their phase relationship. Also label the peak voltages.



- Connect channel 2 of the CRO to Vout2 (leave channel 1 at the input) and sketch this waveform on the axes, again showing its phase relationship and peak to peak voltage values.
- Connect channel 2 to the output of the op amp. Sketch the waveform at the output of the op amp, recording the peak to peak voltage and its phase relationship to Vin. Indicate on the waveform the part the represents the diode forward voltage drop.
- Disconnect the anode of D2 and repeat the above three steps. Record the waveforms on graph set 2 below. Include voltage values and phase relationships.



Step 2: Peak Detector

- Construct the circuit of Fig.2.

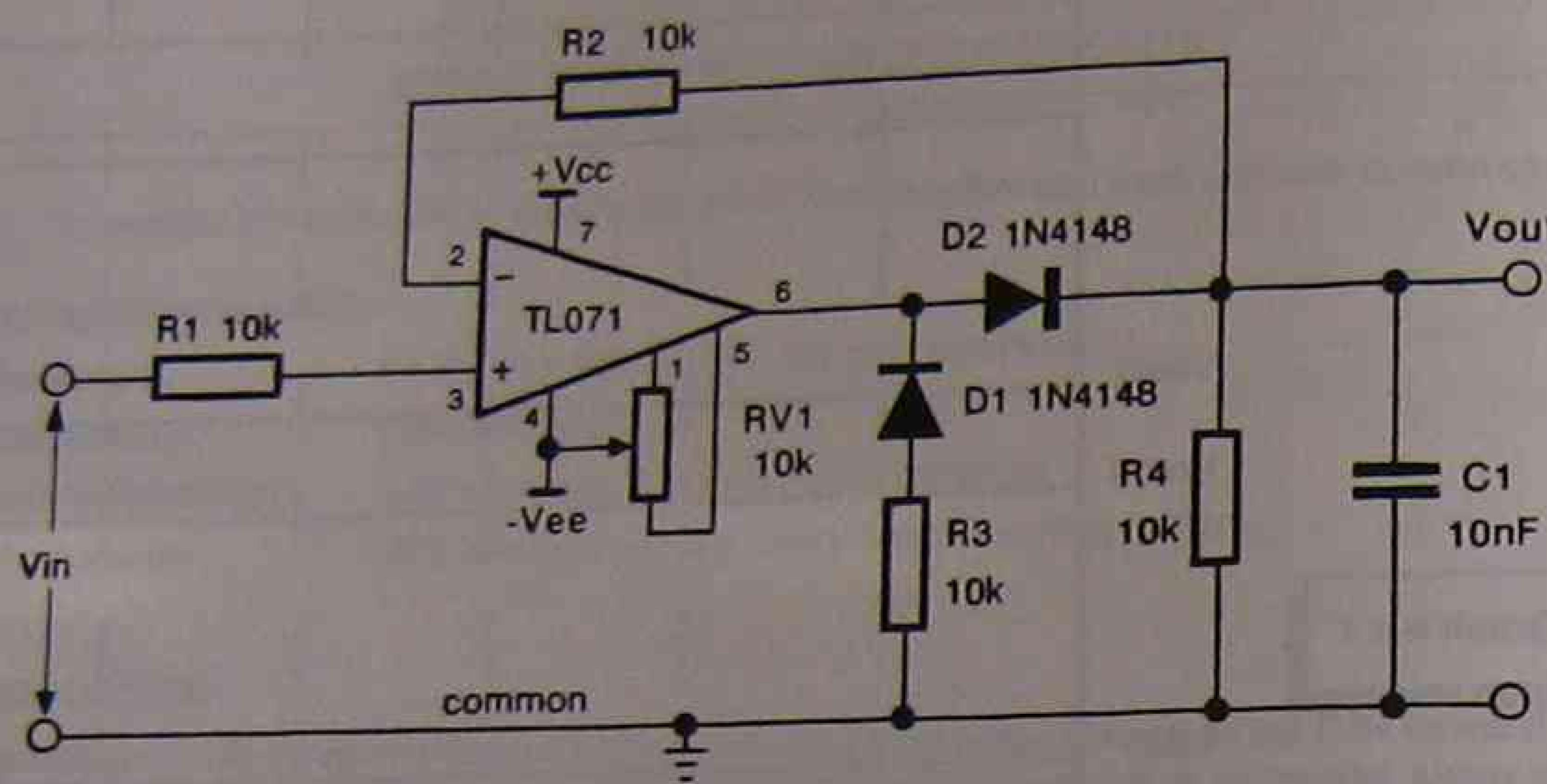
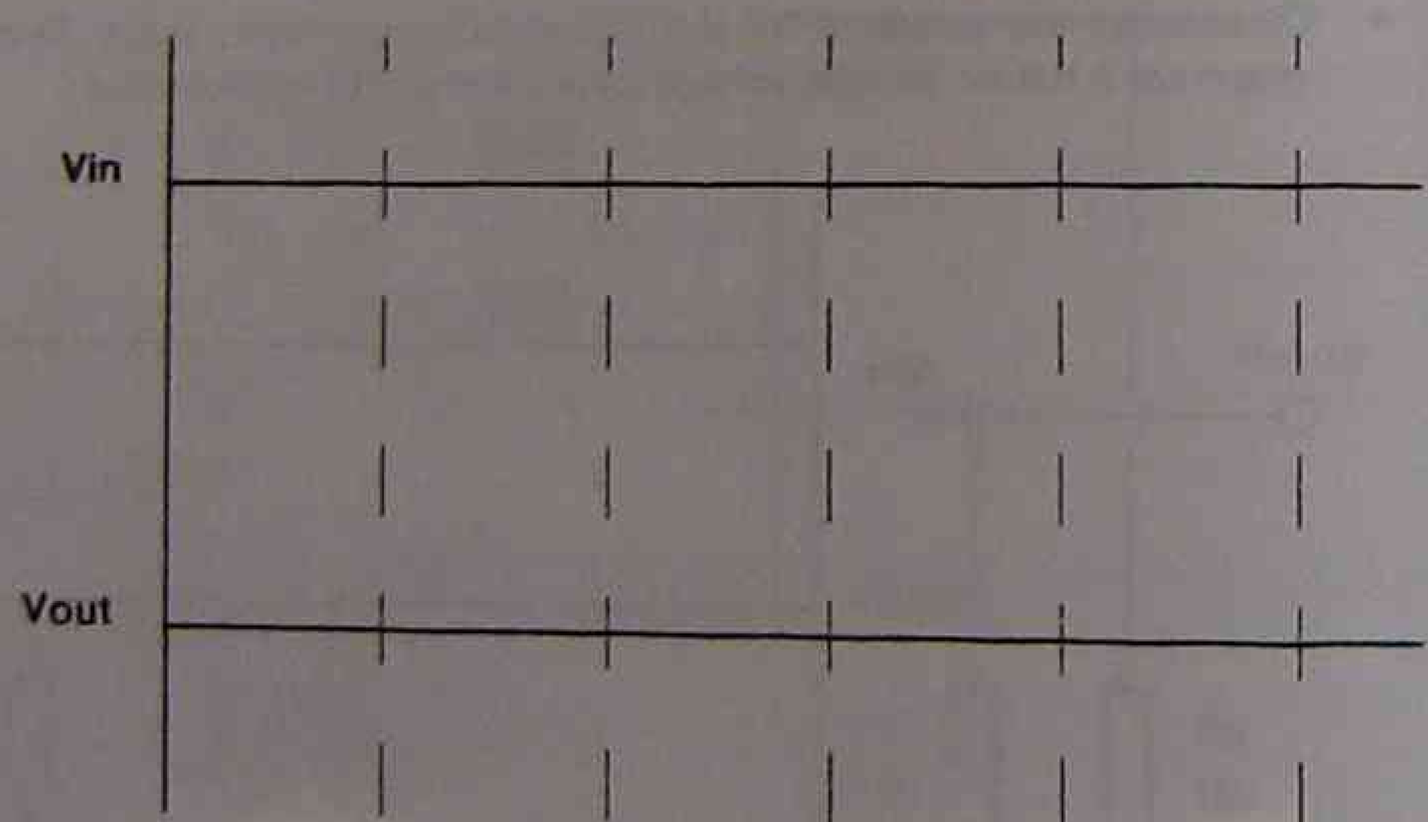


Fig.2: Peak detector

- Apply a 50kHz, 10Vp-p square wave to Vin. Make sure there is no DC component.
- Connect channel of a CRO to Vin and channel 2 to Vout. Also connect a DVM across Vout.
- Sketch Vin and Vout on the axes below and record the peak voltage of Vout as measured by the CRO.

Graph set 3



- Measure Vout with the DVM (set to read DC volts)

Vout (DVM) = ________ volts

- Compare the reading of the DVM to the peak voltage obtained from the CRO.

____

____

- Vary the amplitude of Vin and note the effect this has on the DC voltage level at the output.

____

____

- Vary the frequency of the input signal and determine the effective range of operation.

FMIN = ________ FMAX = ________

QUESTIONS

(a) From the results shown in graph set 1, comment on the polarity and shape of the waveforms at Vout1 and Vout2.

____

____

____

(b) Briefly explain how the circuit is able to rectify AC signals less than 0.6V. Refer to the output waveform of the op amp.

____

____

____

(c) With reference to the waveforms of graph set 2, briefly comment about the effect disconnecting D2 had on the waveforms for Vout1, Vout2 and Vout - op amp.

____

____

____

(d) Determine the closed loop gain of the peak detector circuit of Fig.2.

____

____

(e) For Fig.2, identify the component(s) that cause the capacitor to charge and then discharge.

____

____

____

____

****

SINE AND SQUARE WAVE OSCILLATORS

AIM: To analyse the operation of the symmetrical and non-symmetrical astable, the Wein bridge and the phase shift oscillator op amp circuits.

EQUIPMENT REQUIRED:

Resistors:	2k7, 4k7, 5k6, 3 x 6k8, 2 x 10k, 18k, 33k all ¼W
Potentiometers:	10k, 100k and 1M potentiometers
Capacitors:	3 x 4.7nF, 2 x 10nF polyester
Semiconductors:	2 x TL071 op amps, 2 x 1N4148 signal diodes
Equipment:	dual trace CRO, breadboarding system.

INTRODUCTION

Oscillators have many applications in industrial circuits, such as providing an accurate reference frequency for control, digital or transmission circuitry. Sinewave oscillators are used in the transmission of information by radio as well as providing signal sources for transducers and for other applications. Square wave oscillators are often used in pulse circuits and to provide timing signals. In this assignment both types of oscillators are examined.

Step 1: The Op Amp Astable

- For the circuit of Fig.1, calculate and record:
  - the feedback factor B
  - the frequency of oscillation

NOTE: Refer to the theory notes for the relevant equations.

feedback factor B = __________ frequency = __________

- Construct the circuit of Fig.1. The circuit can be powered with either a 12V or 15V dual polarity power supply.
- Using an oscilloscope, measure Vout, Vx and the waveform across C_T. Sketch the waveforms on graph set 1, labelling the period of each half cycle and the peak voltages.
- From these results, determine the value of $\pm V_{O(SAT)}$ and the frequency of operation.

NOTE: Do not disconnect the circuit.

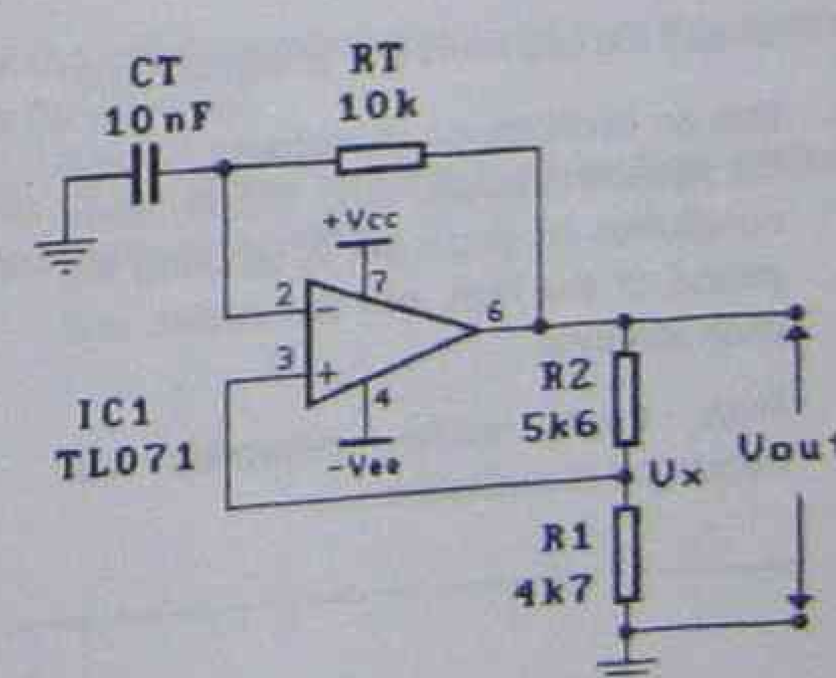
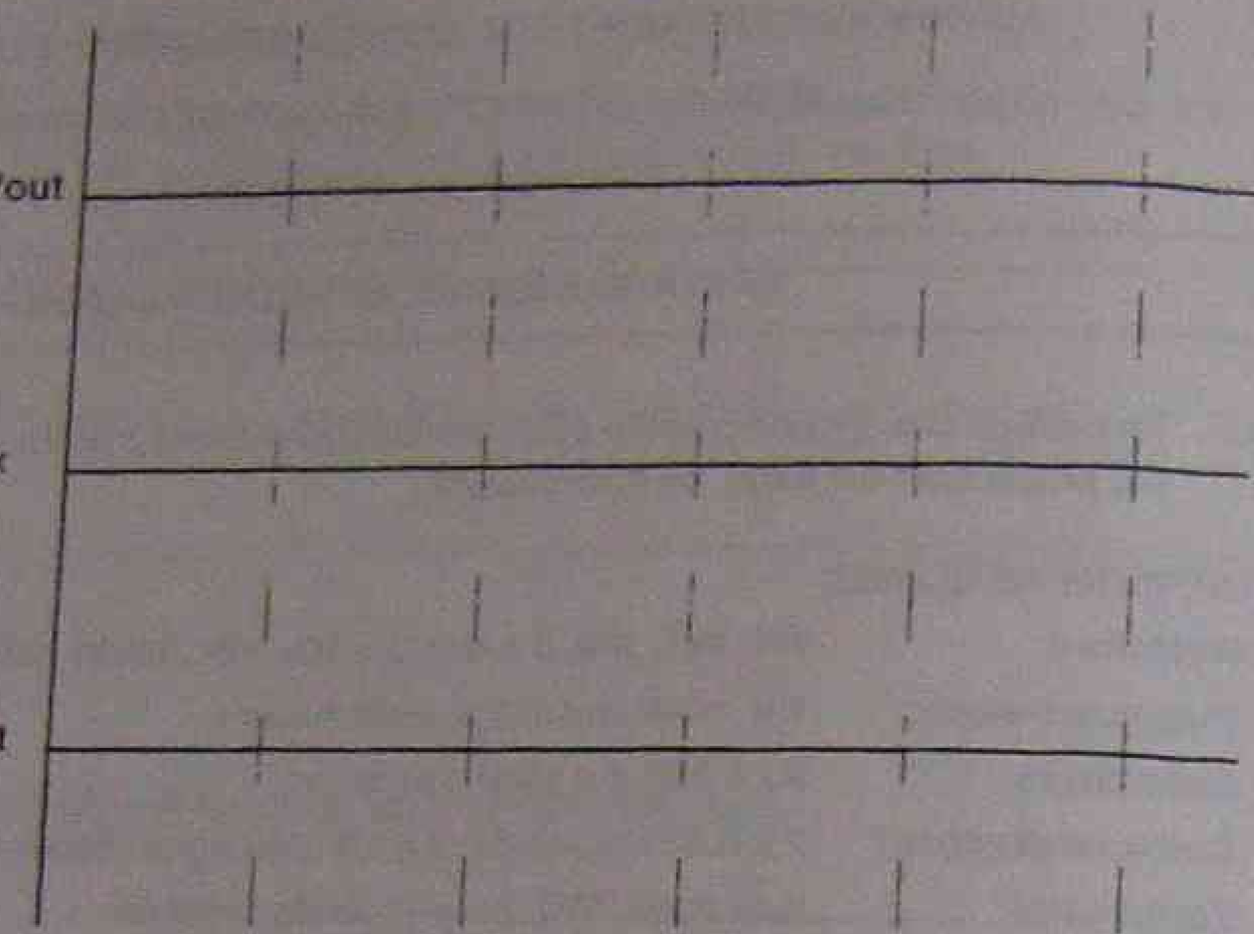


Fig.1: The astable



Graph set 1

Astable

$V_{O(SAT)}$ = ________ frequency = ________

Step 2: The Non-Symmetrical Astable

- For the circuit of Fig.2, calculate and record:
  - the period the output is high
  - the period the output is low
  - the frequency of oscillation

NOTE: Refer to the theory notes for the relevant equations.

T_{HIGH} = ________ T_{LOW} = ________ frequency = ________

- Modify the circuit of Fig.1 to that of Fig.2.
- Use an oscilloscope to measure Vo and the waveform across V_{C1}. Sketch these waveforms on graph set 2, labelling the period of the high and low states and peak voltages.
- From these results, determine the frequency of oscillation.

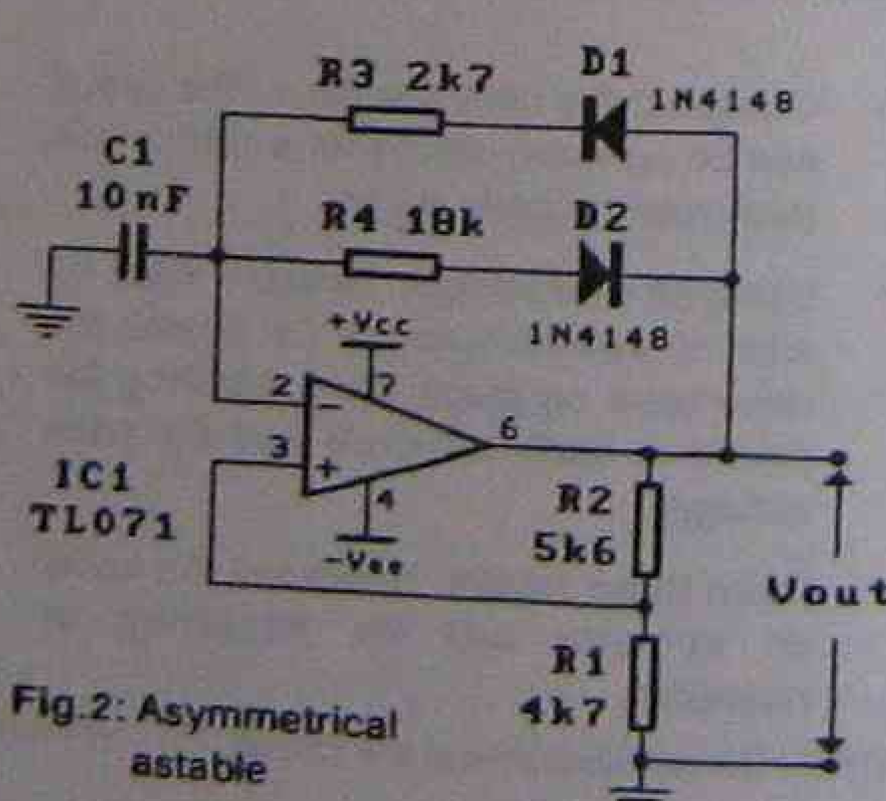
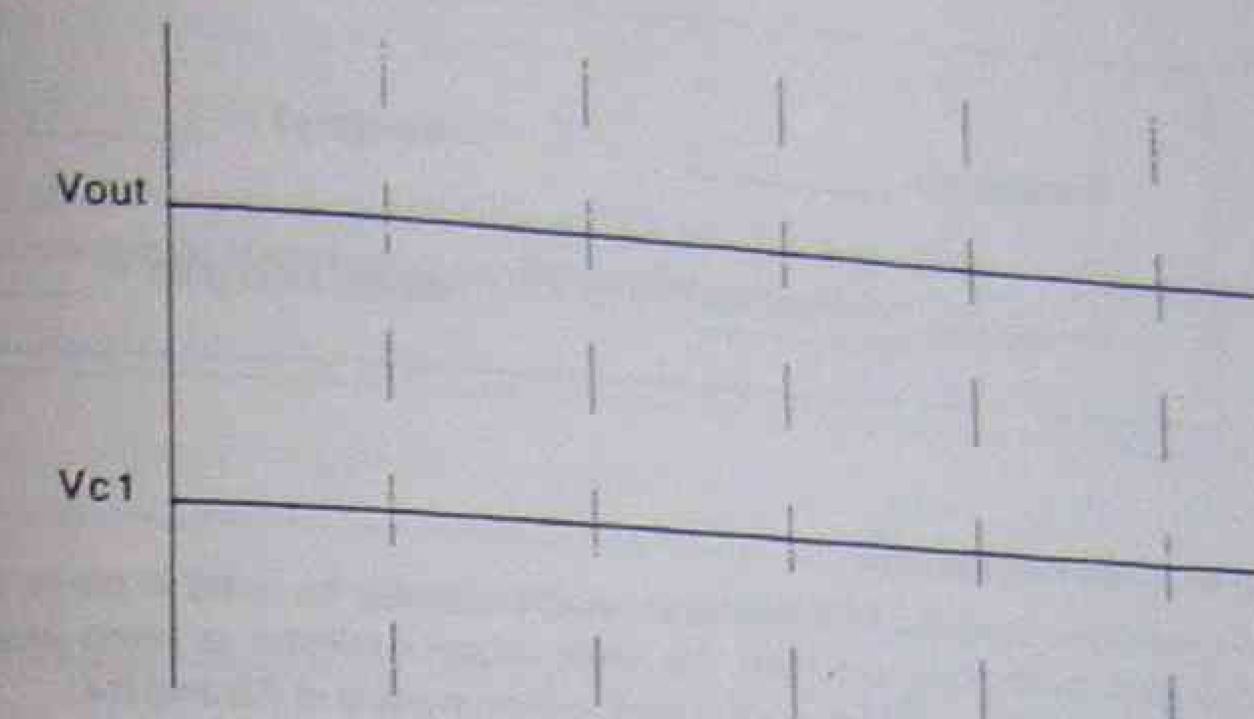


Fig.2: Asymmetrical astable



Graph set 2

non-symmetrical astable

frequency = ________

Step 3: The Phase Shift Oscillator

NOTE: This circuit is suitable for fixed frequency applications only.

- For the circuit of Fig.3, calculate the frequency of operation.

Freq = ______

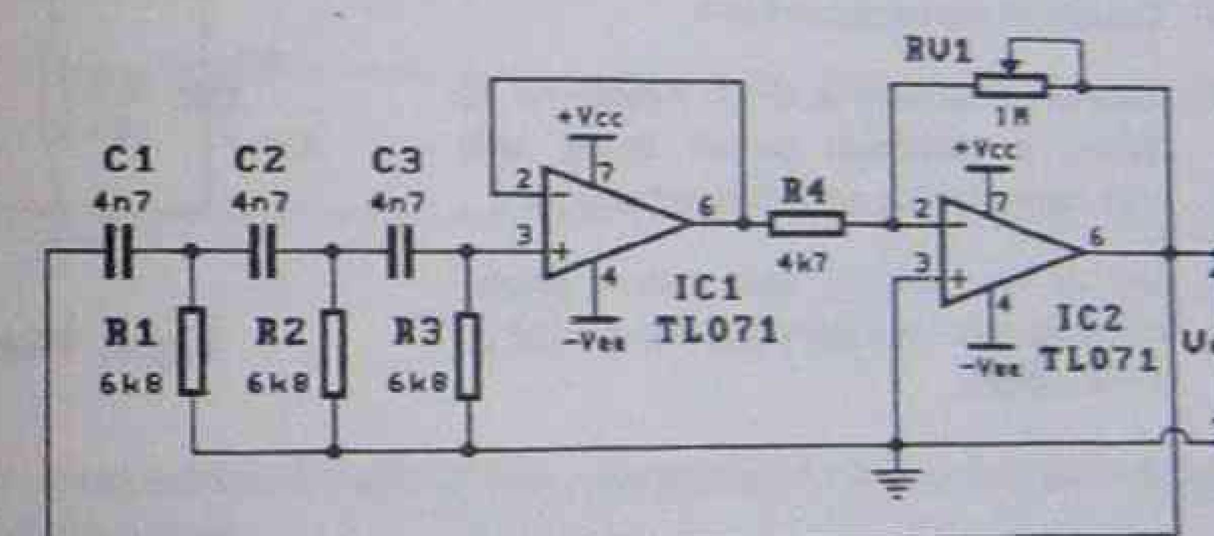
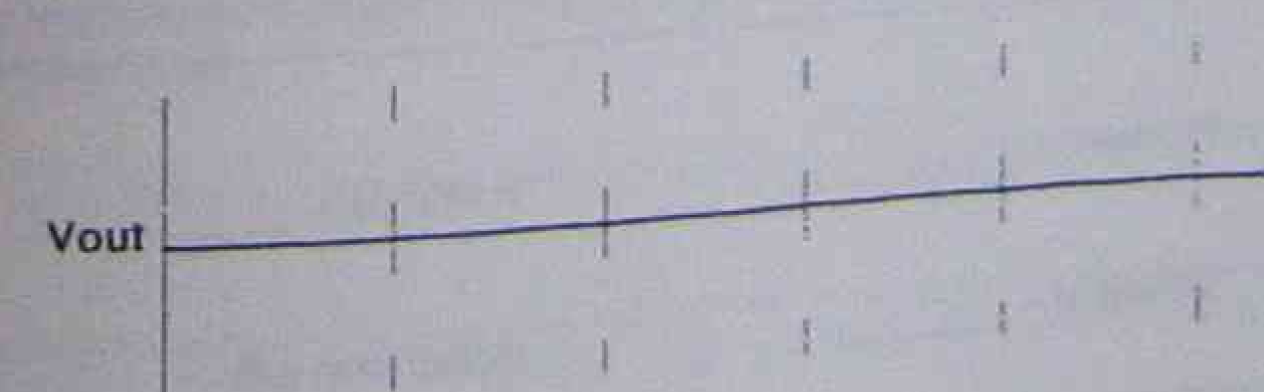


Fig.3: Phase shift oscillator

- Construct the circuit of Fig.3.
- Adjust RV1 to provide a maximum undistorted output voltage. Use an oscilloscope to monitor Vout and sketch the waveform on the axis below. Label the peak voltages and period of the waveform.
- Measure the peak to peak voltage at the inverting (pin 2) terminal of IC2 and from the results determine the closed loop gain of the amplifier (V_{out}/V_{PIN2}).
- From the waveform, determine the frequency of oscillation and the peak to peak output voltage. Record all results in the space provided on the next page.

Graph set 3



Frequency = ________ Vout(p-p) = ________

VPIN2(p-p) = ________ closed loop gain = ________

Step 4: The Wein Bridge

In precision circuits an NTC thermistor would normally be used in place of R2 to control the closed loop gain, minimising distortion. The Wein bridge oscillator is often used as a variable frequency oscillator by using a dual ganged potentiometer in place of R3 and R4.

- Calculate the frequency of oscillation for the circuit of Fig.4.

Frequency = ________

- Construct the circuit of Fig.4.
- Monitor Vout with a CRO. Adjust R2 to obtain a maximum output swing with minimum clipping and reliable oscillation.
- Sketch Vout on the axis shown below, labelling peak voltages and the period of the waveform.

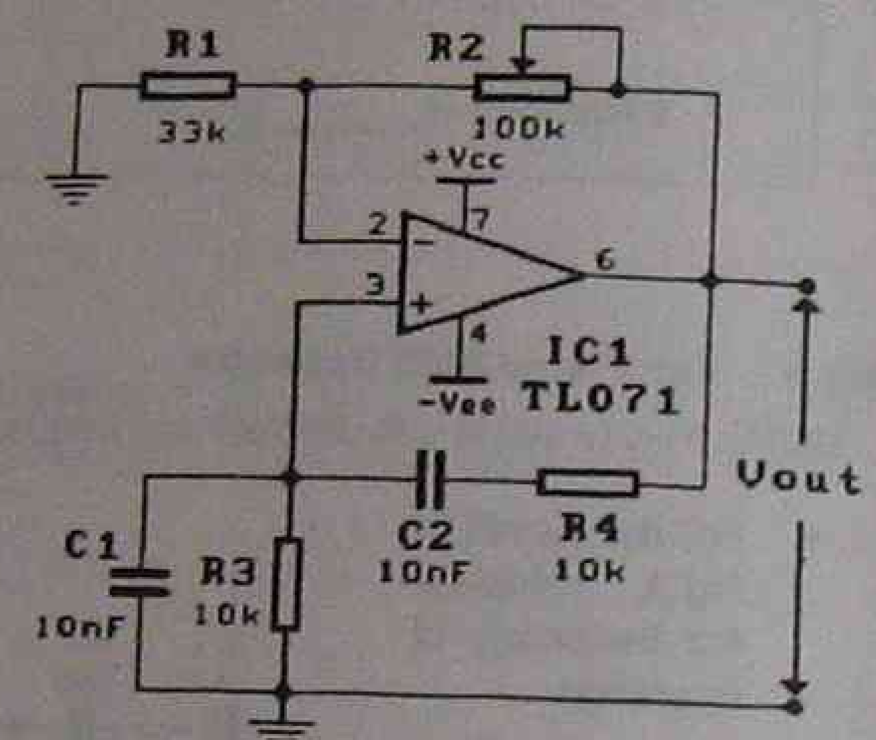
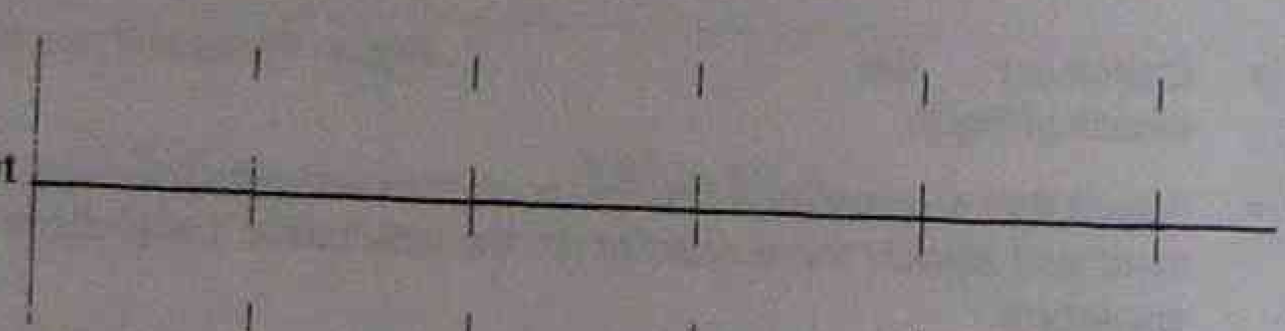


Fig.4: Wein bridge oscillator

Graph set 4

Vout



- Measure the peak to peak voltage at the inverting terminal (pin 2) of the op amp and from the results determine the closed loop gain of the amplifier (Vout/VPIN2). From the waveform, determine the frequency of the oscillator.

Frequency = ________ Vout(p-p) = ________

VPIN2(p-p) = ________ closed loop gain = ________

QUESTIONS

(a) Identify the components that provide positive feedback in the circuit of Fig.1. Briefly explain how these components can be identified as providing positive feedback.

(b) Briefly describe how the ratio of R3 to R4 varies the mark-space ratio in the circuit of Fig.2.

(c) For the circuit of Fig.4, if capacitors C1 and C2 were changed from 10nF to 1nF, determine the operating frequency of the circuit.

(d) Compare the measured values of amplifier closed loop gain for the oscillator circuits of Figs.3 and 4 to the theoretical values.

OP AMP RC CIRCUITS

AIM: To observe the operation of the integrator and the differentiator op amp circuits and to analyse the operation of a basic function generator circuit.

EQUIPMENT REQUIRED:

Resistors:	1k, 3 x 4k7, 6k8, 2 x 10k all ¼W
Capacitors:	1nF, 47nF, 100nF polyester
Semiconductors:	2 x TL071 op amps, 2 x 3V9 and 2 x 5V6 zener diodes
Equipment:	dual trace CRO, function generator, breadboarding system.

INTRODUCTION

RC integrators and differentiators are used to change the shape of a waveform, and when combined with an op amp waveshapes closer to the ideal can be obtained. A useful application is to use these circuits in conjunction with an oscillator to form a function generator able to produce square and triangular waveforms. All of these circuits are examined in this assignment.

Step 1: The Integrator

Note: The equations given in the theory notes assume ideal conditions in which the capacitor is charged with a constant current, giving a triangular waveform out for a square wave input. Practical limitations may cause non-linearity in the output waveform.

- Construct the circuit of Fig.1.
- Connect channel 1 of a dual trace CRO to monitor Vin and channel 2 to monitor Vout. Then apply a 10Vp-p, 1kHz square wave to Vin.
- Observe the output signal and record the waveform on graph set 1 (next page). Record the period and the peak to peak voltage on the waveform.
- Set the signal generator to 10Vp-p, 10kHz and record the peak to peak value of the output voltage.

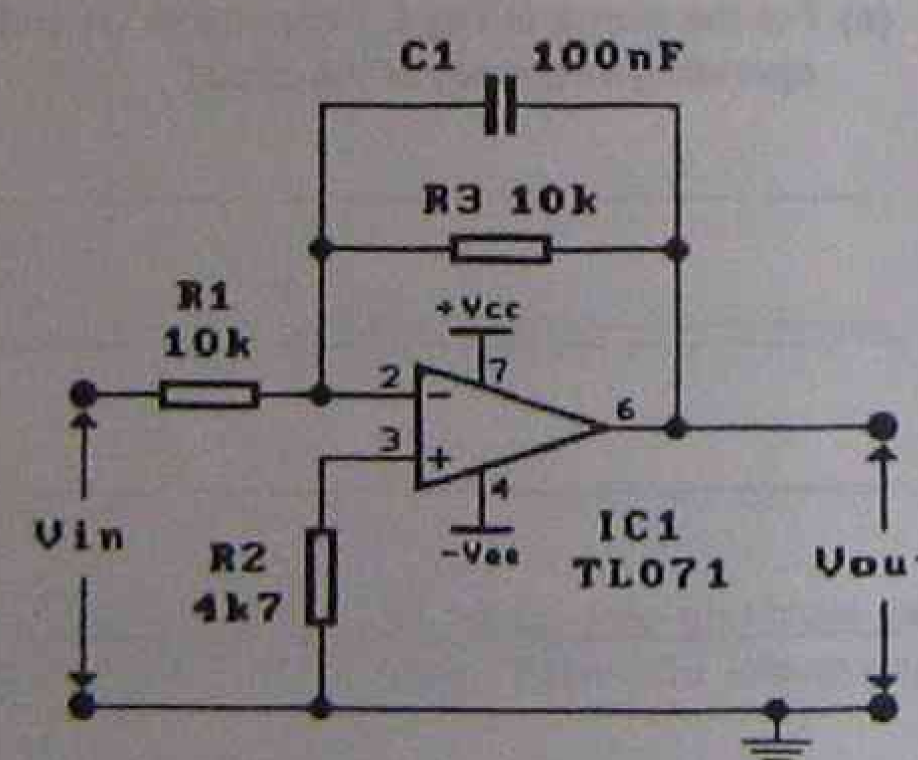
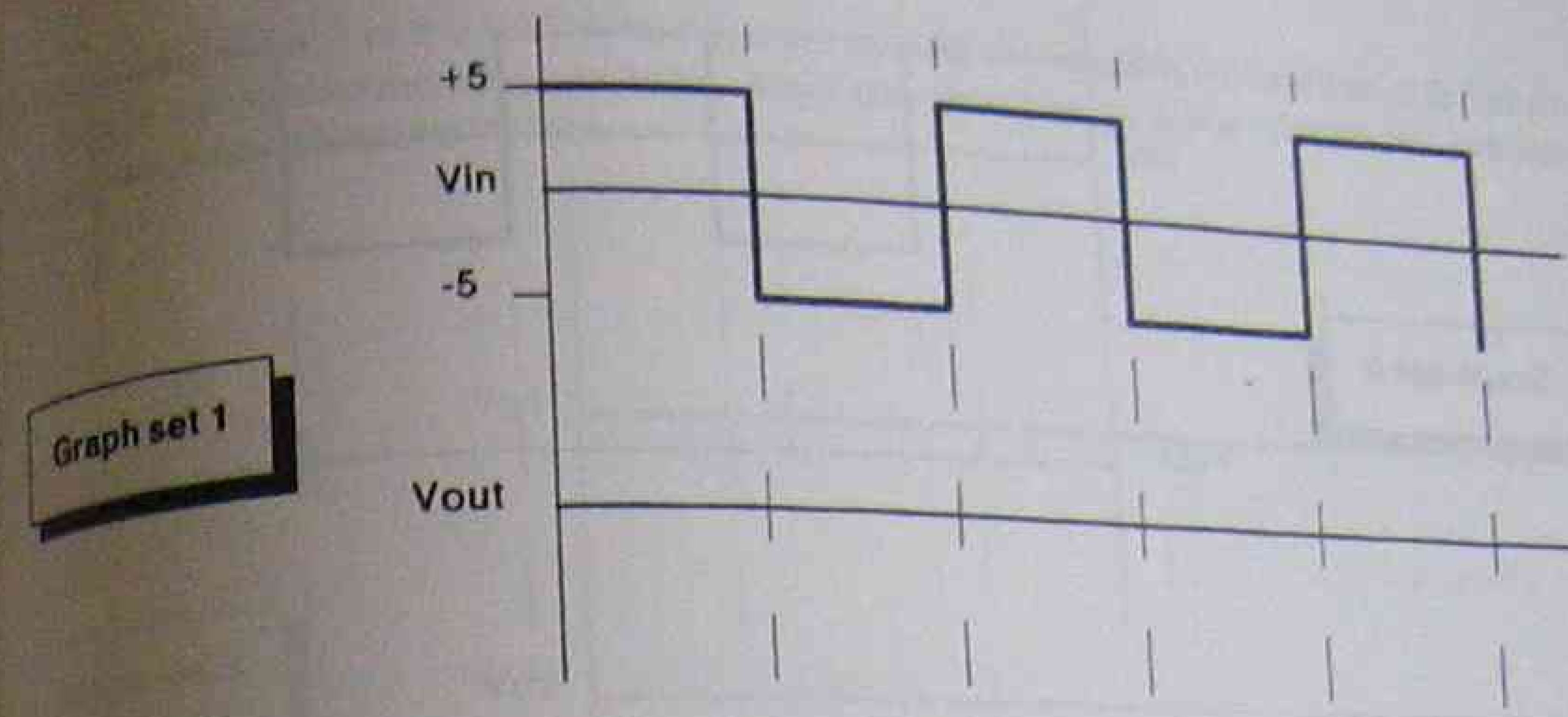
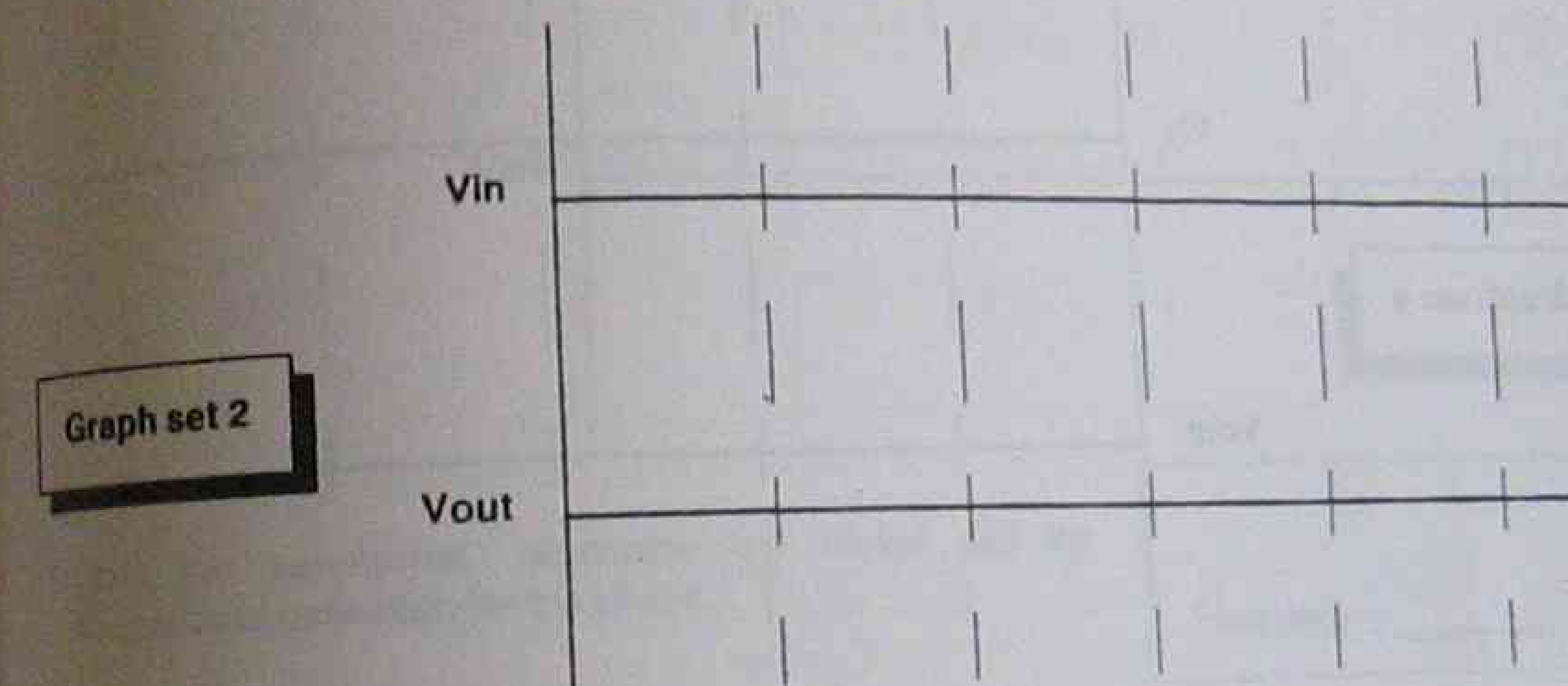


Fig.1: The integrator

Voutp-p at 10kHz = ______

- Apply a 10Vp-p, 1kHz triangular wave to Vin. Record the output waveform on graph set 2. If a triangular wave output is not available, use a 10Vp-p sine wave and note particularly the phase difference between the input and output signals.





Step 2: The Differentiator

- Construct the circuit of Fig.2.
- Apply a 1KHz, 10Vp-p square wave to the input of Fig.2. Use channel 1 of the CRO to monitor Vin and channel 2 to monitor Vout.
- Sketch the output waveform on graph set 3.
- Apply a 10Vp-p, 1kHz triangular wave to the input of Fig.2. If a triangular wave output is not available, use a sine wave and note the phase difference between the input and output signals.
- Record the output waveform on graph set 4.

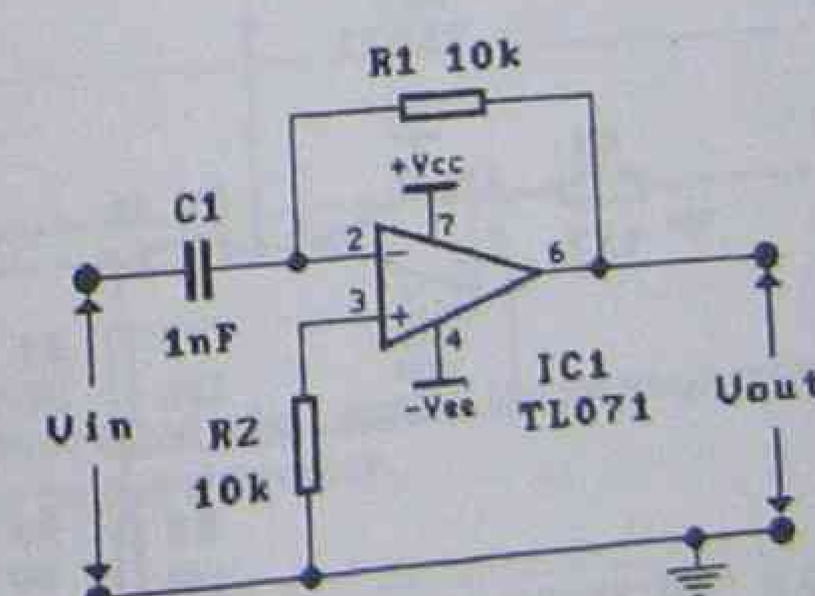
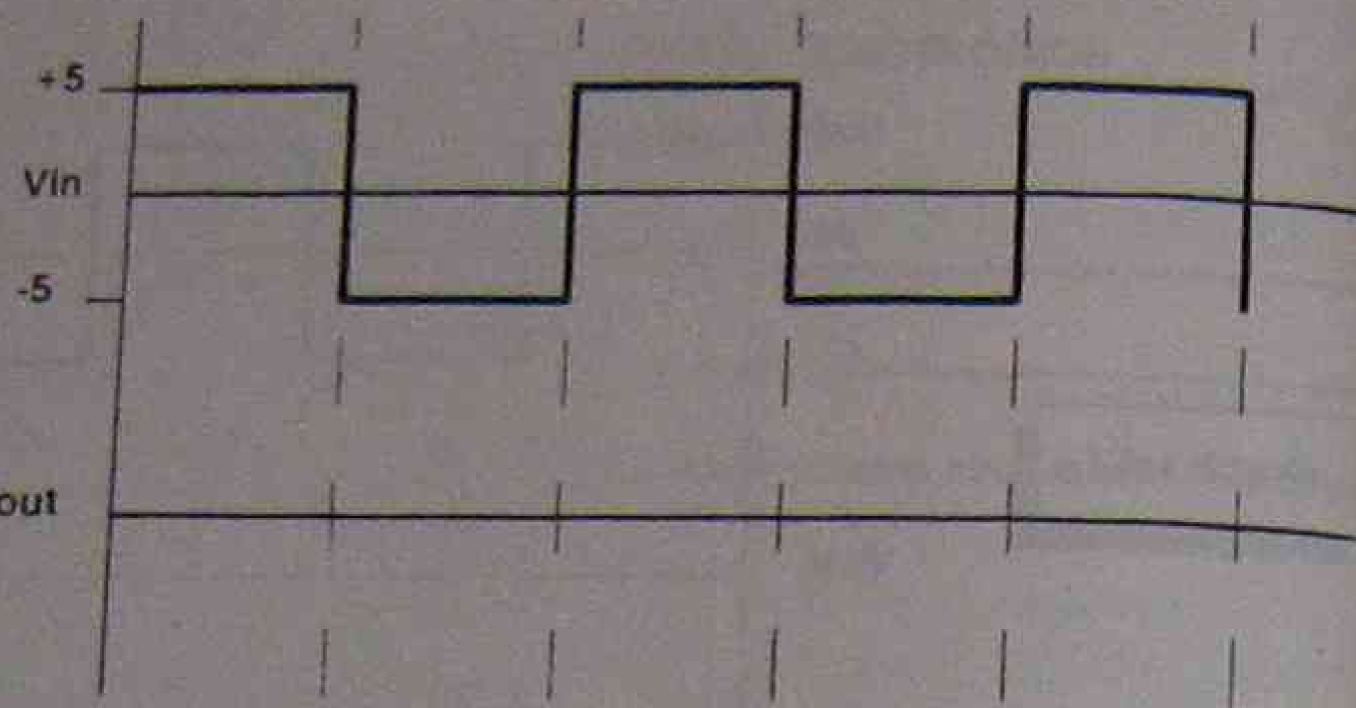
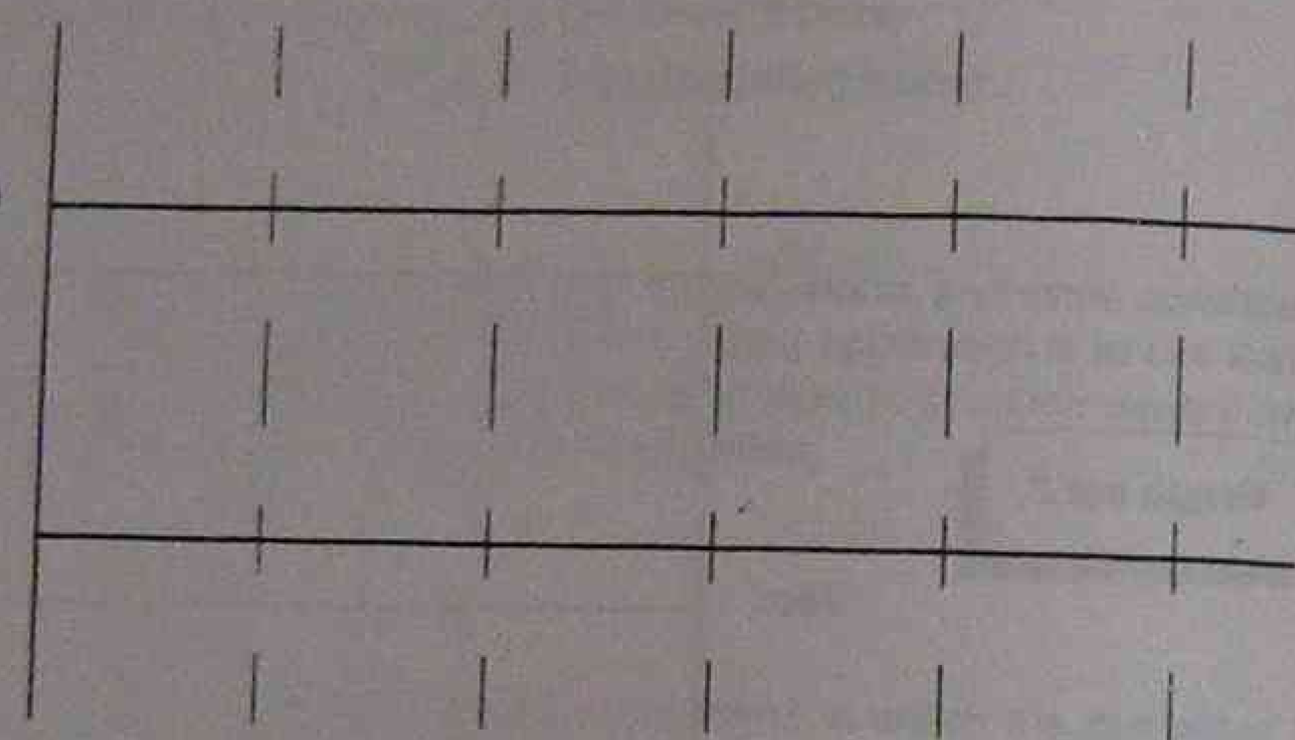


Fig.2: The differentiator

Graph set 3



Graph set 4



Step 3: The Function Generator

- Construct the circuit of Fig.3.

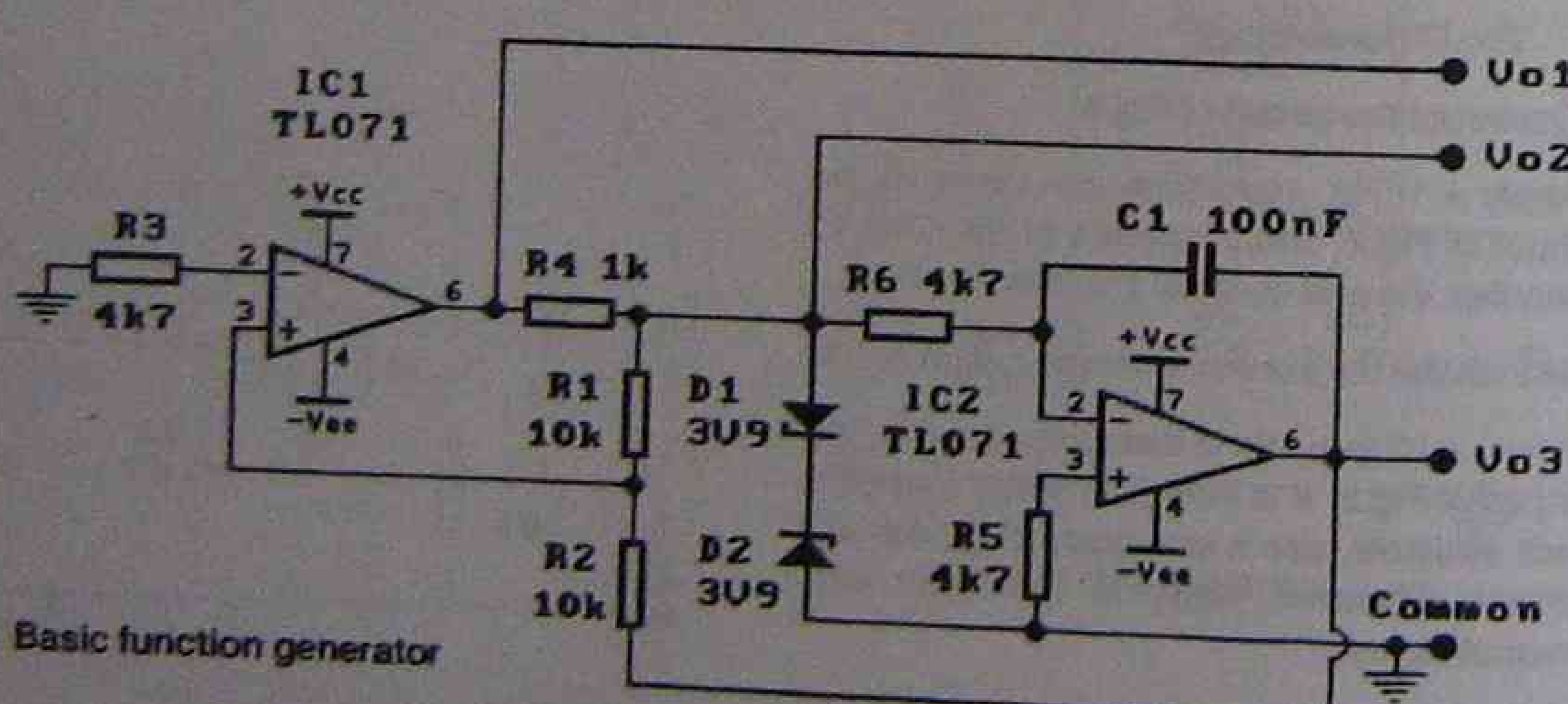


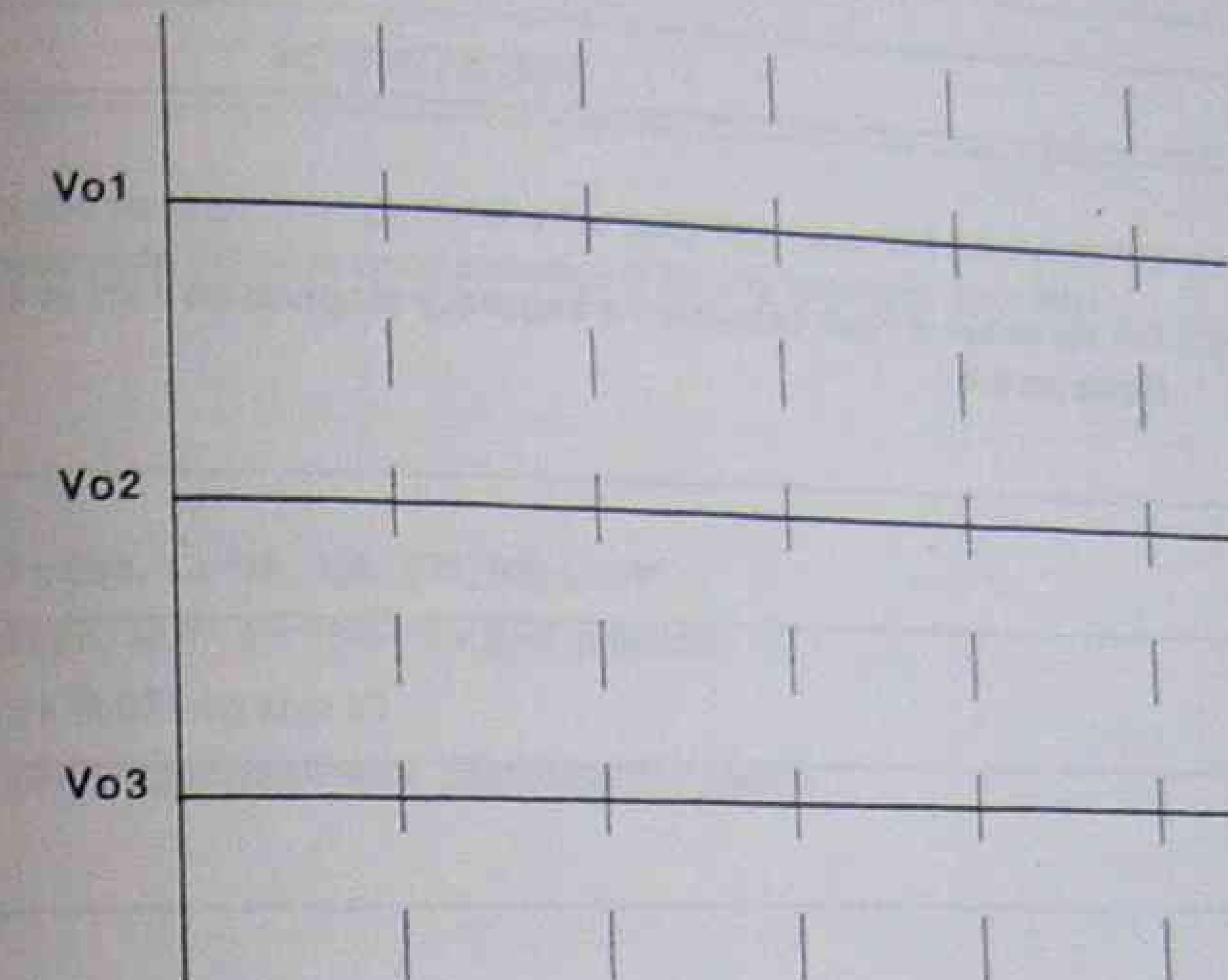
Fig.3: Basic function generator

- Calculate the period and the frequency of the output waveform for the circuit, using the relevant equation from the theory notes.

Frequency = ______

- Connect channel 1 of the CRO to Vo1 and use channel 2 to measure Vo2 and then Vo3. Sketch each of the waveforms on graph set 5. Make sure you show the phase relationships between the waveforms compared to Vo1.

Graph set 5



- From the waveforms, determine the period and the frequency of operation for the circuit.

Frequency = ______

QUESTIONS

(a) For the circuit of Fig.1, identify the components that determine the slope of the output waveform.

______

______

(b) For the circuit of Fig.1, if the amplitude of the input signal is held constant and the frequency is increased, state the effect on the voltage level of the output waveform.

______

______

(c) If the output of the circuit of Fig.1 was connected to the input of Fig.2, identify the shape of the waveform at the output of Fig.2 if a square wave was applied to the input of Fig.1.

(d) For the circuit of Fig.3, calculate the frequency of operation if C1 is 47nF and R6 is 6k8. (Try this if time permits).

(e) For the circuit of Fig.3, describe the effect on the operation of the circuit if D2 and D3 are changed to 5V6 zener diodes. (Try this if time permits)

(f) Compare the measured frequency to the calculated frequency for the circuit of Fig.3.

****

PRACTICAL SESSION 14

ACTIVE FILTERS

AIM: To measure and plot the frequency response of a low pass, high pass and band pass filter and from the curves to determine the pass band and slope of the attenuation for each filter.

EQUIPMENT REQUIRED:

Resistors:	2 x 6k8, 2 x 10k, 22k, 27k, 56k all ¼W
Capacitors:	2n2F, 4n7F, 2 x 10nF, 2 x 22nF polyester
Semiconductors:	2 x TL071 op amp IC
Equipment:	CRO, signal generator, breadboarding system.

INTRODUCTION

Filters are circuits that pass a predetermined band of frequencies while blocking frequencies outside their pass band. The three basic types are the low pass, high pass and band pass filters. All three are examined in this assignment. Refer to the theory notes for the relevant equations.

Step 1: The Low Pass Filter

- For the circuit of Fig.1, calculate the cut-off frequency f_H.

f_H = ________

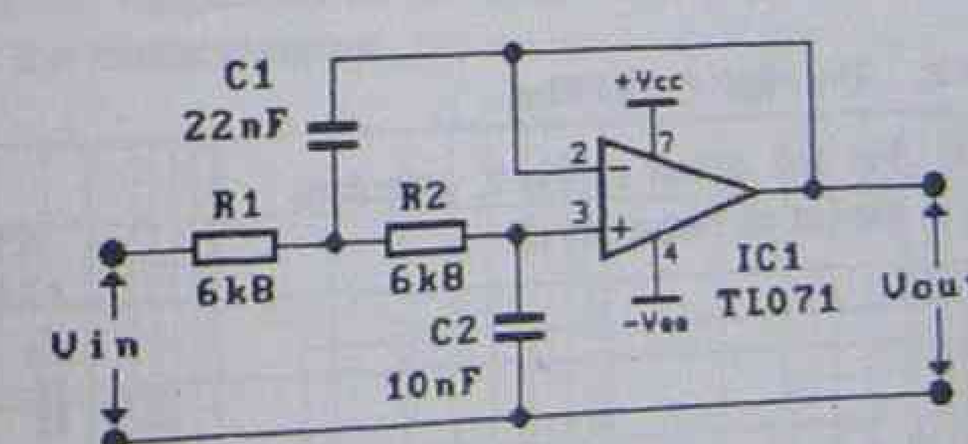


Fig.1: Low pass filter

- Construct the circuit of Fig.1.
- Connect channel 1 of a CRO to the input and channel 2 to the output. Apply a 20Vp-p, 1kHz sinewave to the input of the circuit.
- Measure the peak to peak voltage of the output for the frequencies shown in Table 1. Record these results in Table 1 in the column for the low pass filter.

NOTE: Ensure that Vin remains constant at 20Vp-p for each measurement.

Table 1

Frequency	Vout (Vp-p)		
	Low pass filter	High pass filter	Band pass filter
100Hz			
200Hz			
300Hz			
500Hz			
1kHz			
2kHz			
3kHz			
5kHz			
10kHz			
20kHz			

Step 2: The High Pass Filter

- For the circuit of Fig.2, calculate the cut-off frequency fL.

fL = ______

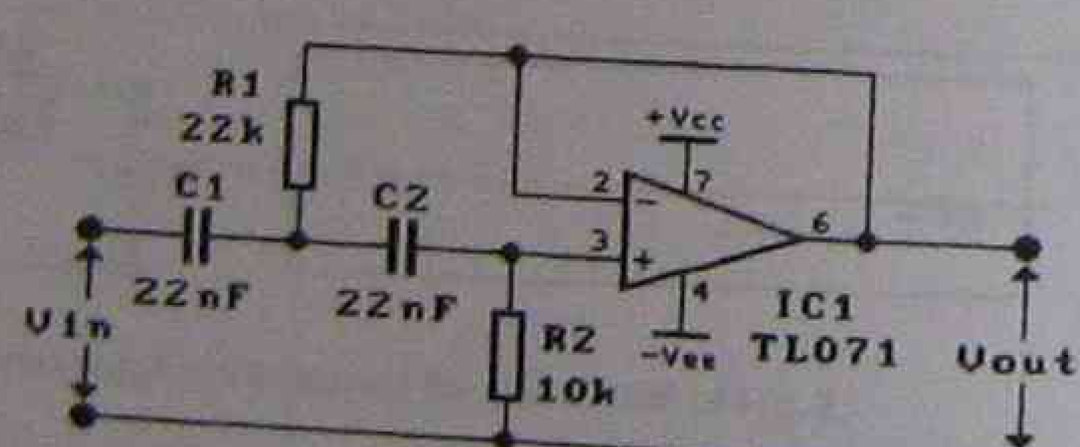


Fig.2: High pass filter

- Construct the circuit of Fig.2.
- Connect channel 1 of a CRO to the input and channel 2 to the output. Apply a 20Vp-p, 1kHz sinewave to the input of the circuit.
- Measure the peak to peak voltage of the output for the frequencies shown in Table 1. Record these results in Table 1 in the column for the high pass filter.

NOTE: Ensure that Vin remains constant at 20Vp-p for each measurement.

Step 3: The Band Pass Filter

- For the circuit of Fig.3, calculate the upper and lower cut-off frequencies.

fL = ______	fH = ______

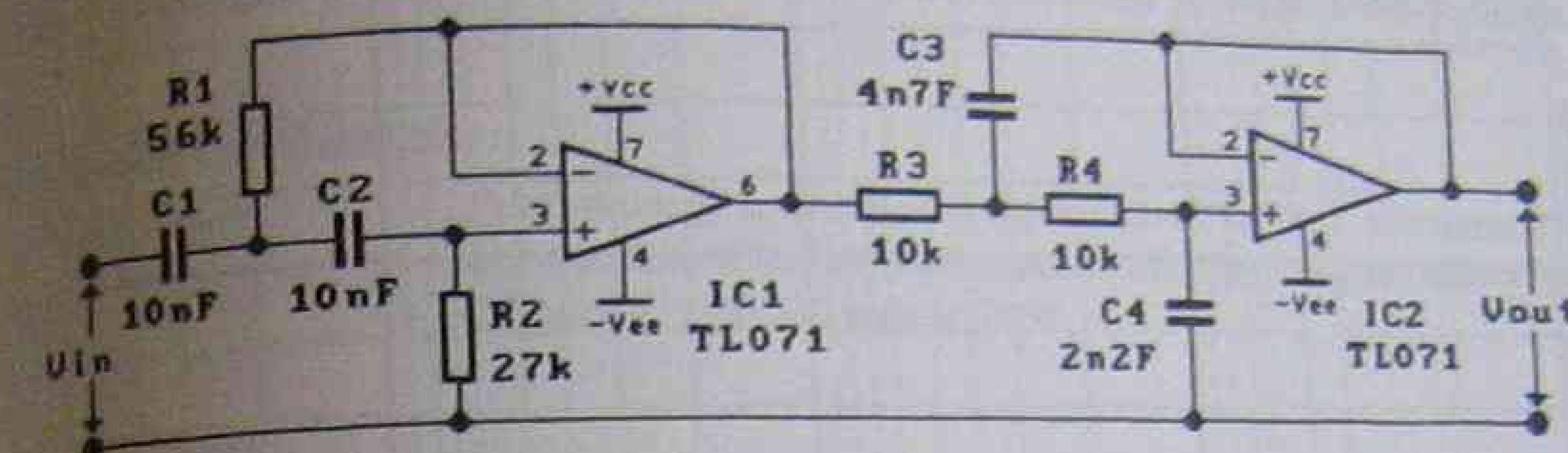


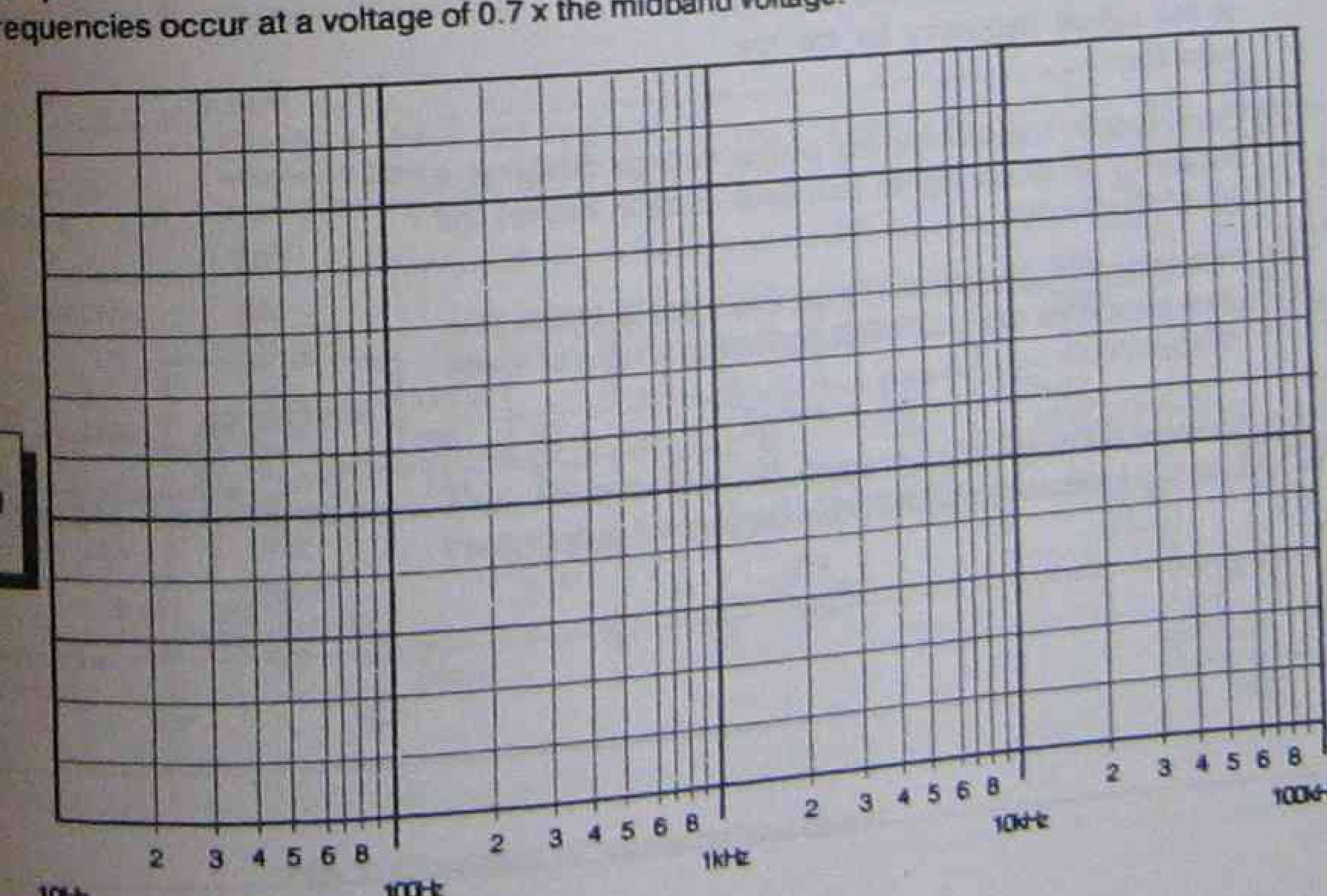
Fig.3: Band pass filter

- Construct the circuit of Fig.3.
- Connect channel 1 of a CRO to the input and channel 2 to the output. Apply a 20Vp-p, 1kHz sinewave to the input of the circuit.
- Measure the peak to peak voltage of the output for the frequencies shown in Table 1. Record these results in Table 1 in the column for the band pass filter. NOTE: Ensure that Vin remains constant at 20Vp-p for each measurement.

Step 4: Frequency Response Curves

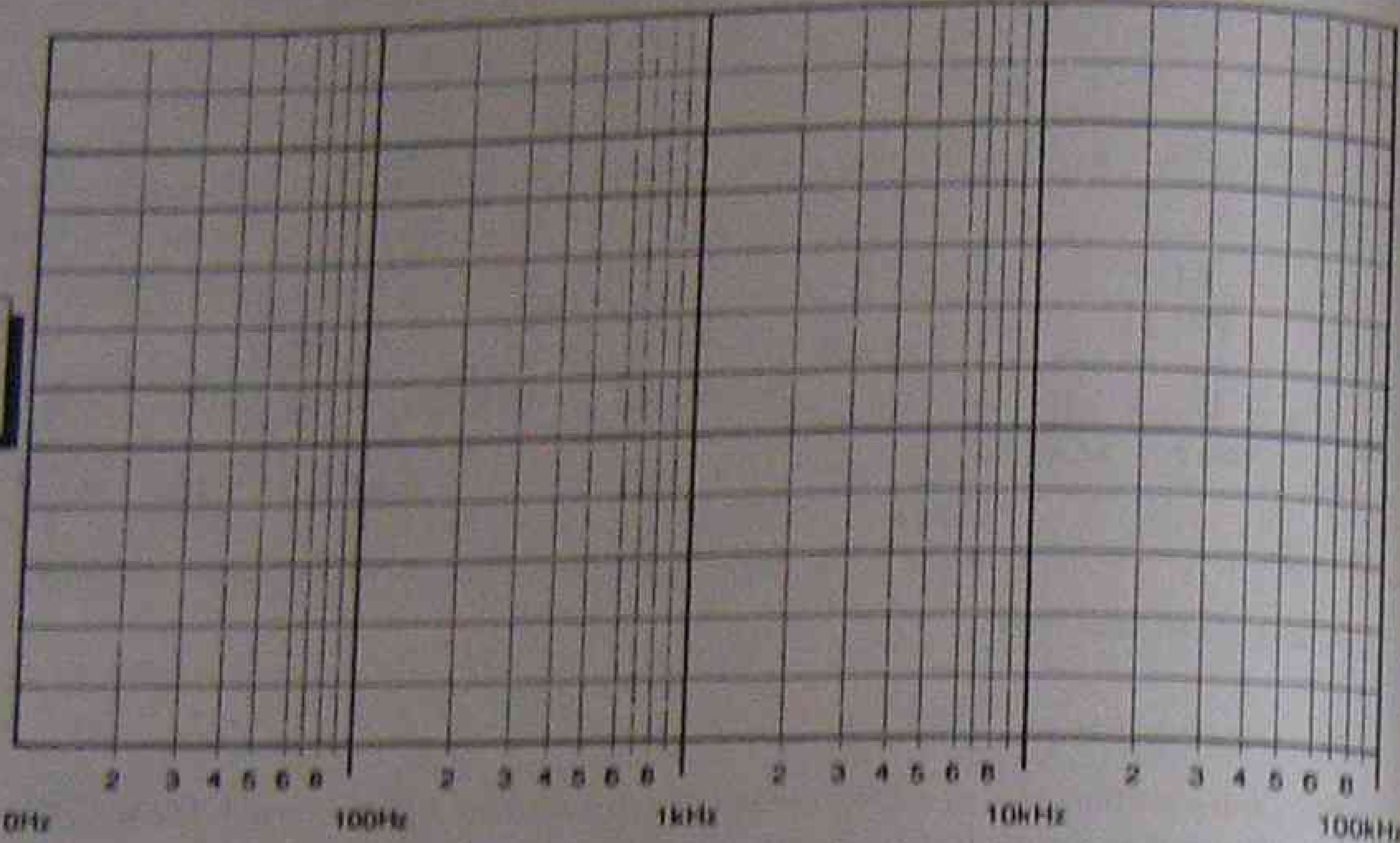
- From the results recorded in Table 1, sketch the response of the low pass filter and the response of the high pass filter on Graph 1. Use different colours to sketch each curve. Show the pass band and the measured cut-off frequencies on the graph. The measured cut-off frequencies occur at a voltage of 0.7 x the midband voltage.

Graph 1



- From the results recorded in Table 1, sketch the response of the band pass filter on Graph 2. Show the pass band and the measured cut-off frequencies on the graph. The measured cut-off frequencies occur at a voltage of 0.7 x the midband voltage

Graph 2



A method of classifying filters is by the slope of rejection of unwanted frequencies. The theory notes state that a slope of -40dB/decade categorises a filter as a 2nd order filter. This equals a loss of 100 times over a decade of frequencies, as shown in Fig.4. The following should confirm this.

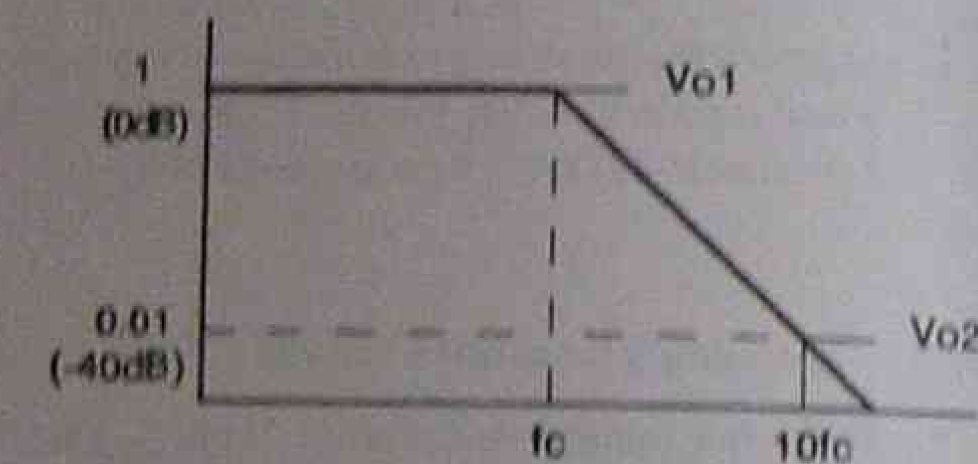


Fig.4: Low pass filter response

(a) From the measurements determine the peak to peak output voltage (Vo1) at the cut-off frequency for the low pass filter........................ Vo1 = ______ V

(b) From Graph 1 determine the output voltage (Vo2) at a frequency 10 times higher than (one decade above) the cut-off frequency........................ Vo2 = ______ V

(c) Determine the attenuation [20 log (Vo2/Vo1)]. **Note**: This value should be close to -40dB, indicating a ratio of 1:100 for Vout:Vin........................ attenuation = ______ dB

(d) Compare the slope of each graph. Do they have the same slope as the circuit of Fig.1? If so, does this indicate that all the filters are second order types?

PRACTICAL SESSION 15

MULTISTAGE CIRCUITS

AIM: To examine the operation of a DC motor control circuit that uses four different stages.

EQUIPMENT REQUIRED:

Equipment: Printed circuit board containing the circuit of Fig.1; thermocouple thermally bonded to a 100 ohm, 10W resistor; DC power supply; 300mA ammeter; 6V DC motor; ±15V DC supply; DVM.

INTRODUCTION

The circuit being studied in this assignment is a typical op amp circuit used in industrial equipment. It has a number of stages, is DC coupled and requires DC nulling. The first step in determining how the circuit works is to identify the function of each stage. Once this is done, their function in the system is determined. Calculations can also be made to determine expected gain, frequency response, signal shaping, power control or frequency of oscillation if the circuit stage is an oscillator.

The circuit of Fig.1 is an open loop, DC motor control circuit. A fan mounted between a furnace and a cooling chamber is required to control the temperature of a cooling chamber using heat from the furnace to heat the cooling chamber.

If natural air flow is sufficient to maintain the cooling chamber's temperature, the fan is off. If the cooling chamber's temperature is too cool, the fan will run clockwise and boost air flow from the furnace. If the cooling chamber's temperature is too hot, then the fan will run anticlockwise, to oppose the flow of air from the furnace, and vent external air into the chamber.

A thermocouple provides temperature sensing within the cooling chamber. The output of the thermocouple is amplified, filtered and processed after which it controls the direction of the motor current through two output transistors.

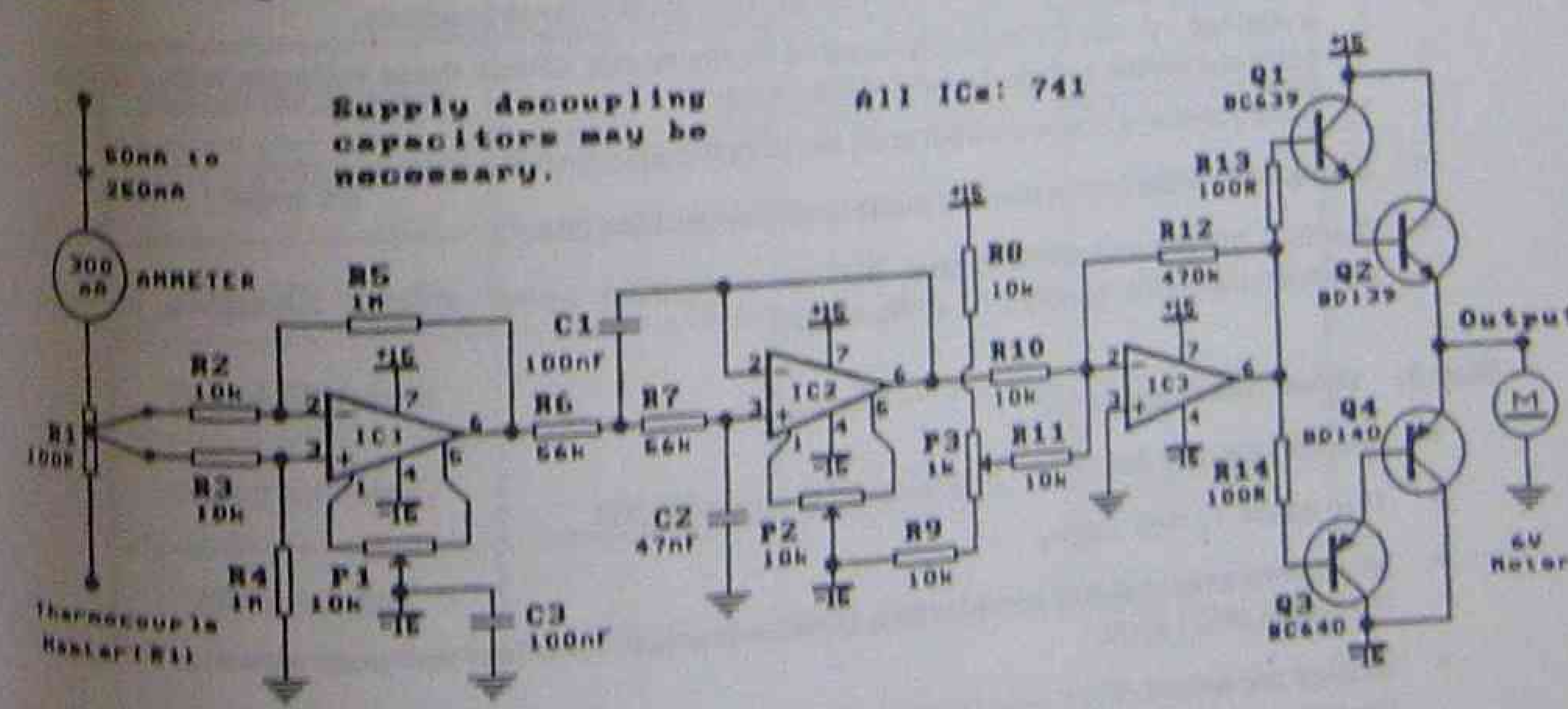


Fig.1: Open loop DC motor control circuit

Step 1: Calculations

- For the circuit of Fig.1, identify the configuration of each stage

Stage 1 (IC1) = ______________________

Stage 2 (IC2) = ______________________

Stage 3 (IC3) = ______________________

Stage 4 (transistor output stage) = ______________________

- Calculate the gain of stages 1, 3 and 4. (Assumptions may be used). Enter these results in Table 1.
- Determine the cut-off frequency for stage 2.

fH = ______

- Determine the ideal voltage gain at frequencies in the pass-band. Enter this result in Table 1.

Table 1

Stage	Theoretical gain (Av)	Measured gain (Av)
IC1		
IC2		
IC3		

Step 2: Practical Procedure

Note: This circuit should be already constructed on a printed circuit board. However, if necessary, individual stages could be constructed on a breadboard and their operation analysed if a PCB design is not available. The circuit can also be used a theoretical exercise.

- Don't apply power to the heating resistor (R1) until indicated.
- Connect the circuit as advised by the teacher, noting the following points:
  - a dual rail +/- 15V DC supply is required for the circuit. Check these voltages with a digital voltmeter before applying power to the circuit.
  - the DC motor is connected between the circuit output and the common (0V).
- Connect the leads from a thermocouple to the input of the circuit.
- Connect the thermocouple heater (R1) to a separate power supply, through a 300mA ammeter. Leave this supply OFF at this stage.

Step 3: Offset Null Adjustment

- Short the input of the circuit. That is, so 0V is applied to IC1.
- Turn on the +/-15V supply.
- Monitor the output of IC1 (pin 6) with a DVM, and adjust the offset null potentiometer P1 until the output of IC1 is 0V.
- Monitor the output of IC2 (pin 6) with a DVM, and adjust the offset null potentiometer P2 until the output of IC2 is 0V.
- Remove the short from the input.

Step 4: Setting The Quiescent Conditions

The quiescent conditions occur when the circuit is in its balanced state. That is at the temperature that requires no movement of air from the fan, or when the output of the circuit is 0V.

- Turn on the supply to the thermocouple heater and adjust the supply current to 200mA. Allow 5 minutes for the temperature to stabilise.
- Monitor the output of IC3 with a DVM and adjust P3 to give 0V at the output (pin 6) of IC3. This should cause the motor to stop rotating and represents the desired set point.

Step 5: Circuit Measurements

- Check that the thermocouple heater current is still 200mA.
- Use a DVM to accurately measure the:
  - thermocouple output voltage
  - voltage at the wiper of P3
  - output voltage of each op amp.
- Record these values in Table 2. Note also if the motor is rotating and its direction.
- Increase the thermocouple heater current to 250mA. Allow 5 minutes for the temperature to stabilise then repeat the above measurements and record the results in Table 3.
- Decrease the thermocouple heater current to 150mA. Allow 10 minutes for the temperature to stabilise, then repeat the above measurements. Again record the results in Table 3.
- Using the measured DC voltages from Table 2, determine the measured voltage gain for each op amp. Enter the results in Table 1.

Table 2

Voltage measured at (volts)	Current in R1		
	150mA	200mA (set point)	250mA
Thermocouple output			
Wiper P3			
Output IC1 (pin 6)			
Output IC2 (pin 6)			
Output IC3 (pin 6)			
Motor rotation			

Conclusion

(a) State the purpose of using a differential amplifier as the input stage.

____

____

(b) State the purpose of using a low pass filter as the second stage.

____

____

(c) Using the results of Table 2, explain how the inverting summer is used to set the quiescent output conditions.

____

____

(d) State why IC3 has no offset null adjustment.

____

____

Fault Analysis

1. Describe the effect on the operation of the circuit if:

(a) the thermocouple leads were to become open circuit.

____

(b) R10 became open circuit.

____

(c) R11 became open circuit.

2. If the motor does not operate when the output of IC3 is negative, state the most likely fault.

____

____

POWER OP AMPS

AIM: To use a power op amp type SGS L165 in the applications of: DC motor control, dual rail supply generator and a power oscillator.

EQUIPMENT REQUIRED:

Resistors: 2 x 1 ohm, 1k5, 2 x 2k2, 3k3, 2 x 5k6, 2 x 47k all ¼W, 12ohm 10W, 1M pot

Capacitors: 1 x 10nF, 2 x 220nF polyester, 2 x 47 μF, 100μF electrolytic

Semiconductors: 2 x SGS L165 power op amps, 4 x 1N4004 1A diodes, 2 x 1N4148 diodes, 2 x 24V zener diodes

Equipment: CRO, DC power supply, DVM, breadboarding system, 24V DC motor.

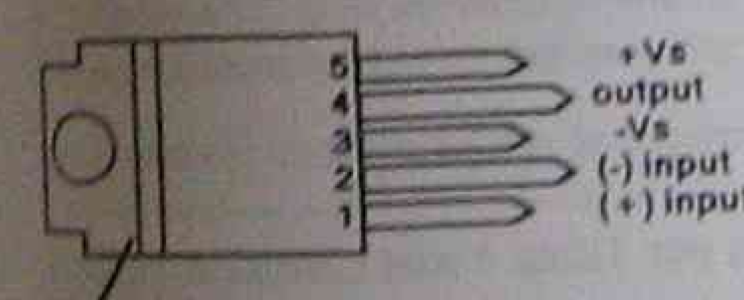
INTRODUCTION

Power op amps are used as servo amplifiers, in power supplies and other areas where the superior performance of an op amp needs to be combined with a boost in power output. In this assignment, the high current, high power op amp, type SGS L165 is used. Features of the L165 are:

- output current up to 3A.
- total power output up to 20W.
- large common mode and differential mode ranges.
- safe operating area (SOA) protection.
- thermal protection.

Abbreviated electrical characteristics:

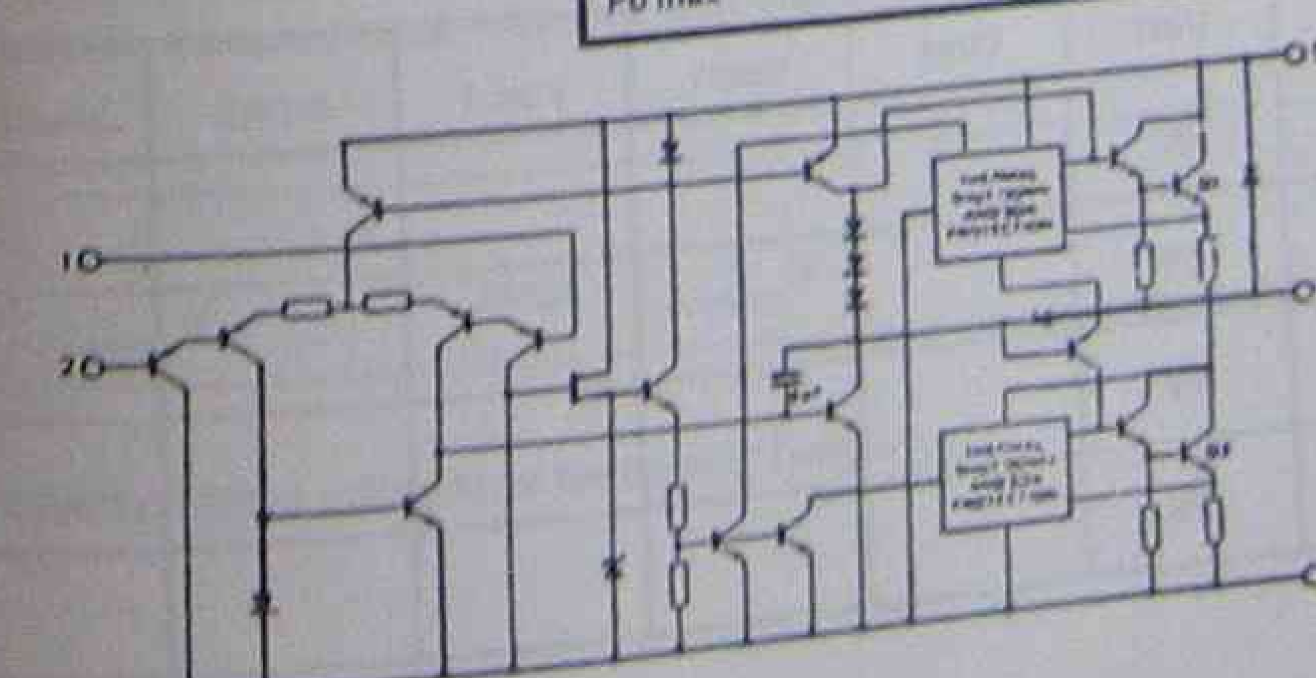
Parameter	Min	Typ	Max	Unit
Vs (supply voltage)	6		18	V
Vo (output voltage swing)		27		Vp-p
Vin (input voltage)			Vs	Vp-p
Vd (differential input voltage)			±15	V
Rin (input resistance)	100	500		kohm
Av (voltage gain, open loop)			80	dB
CMR (common mode rejection)			70	dB
Efficiency (Po = 18W, Ip = 3A)			60	%
Slew rate		8		V/μs
Io (peak output current)			3.5	A
Po max			20	W



tab connected to pin 3

Pin connections

The internal schematic diagram is shown below.



Step 1: Bi-Directional DC Motor Control

This circuit provides an interface from TTL/CMOS/µP compatible logic inputs to control the direction of rotation of a DC motor.

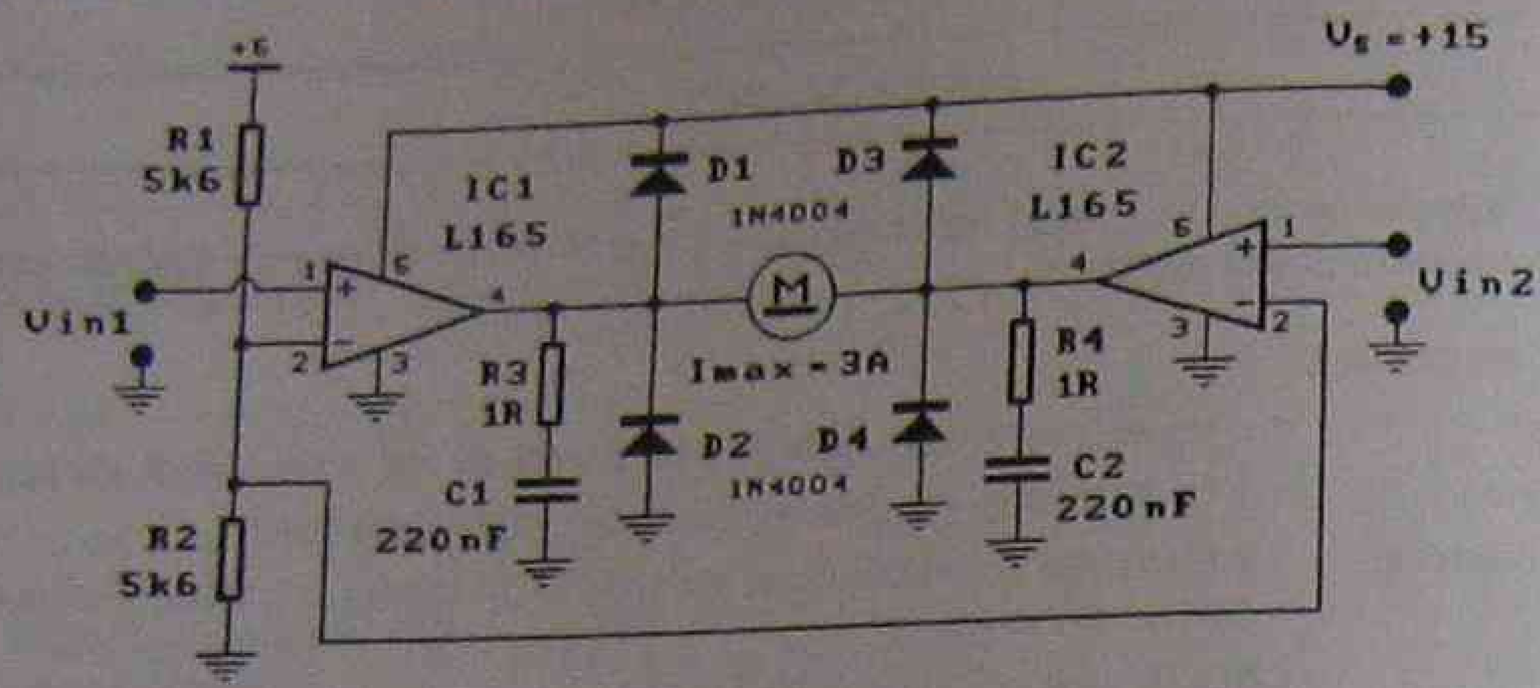


Fig.1: Bi-directional DC motor control circuit

- For Fig.1:
  - calculate the reference voltage at the inverting input (pin 2)of each op amp.
  - Identify the circuit configuration for IC1.
  - Identify the circuit configuration for IC2.

VREF = ______ V

Circuit configuration IC1 = ____________________

Circuit configuration IC2 = ____________________

- Connect the circuit of Fig.1.
- Apply 5V or 0V (logic 1 and logic 0) to the circuit inputs as per Table 1 and complete Table 1. Also measure and record the supply voltage Vs.

Vs = ______ V

Table 1

Vin1	Vin2	VREF	VoIC1	VoIC2	ILOAD	Direction of motor
0V	0V					
0V	5V					
5V	0V					
5V	5V					

Step 2: Dual Power Supply

This circuit provides a dual polarity power supply at the output from a single rail power supply.

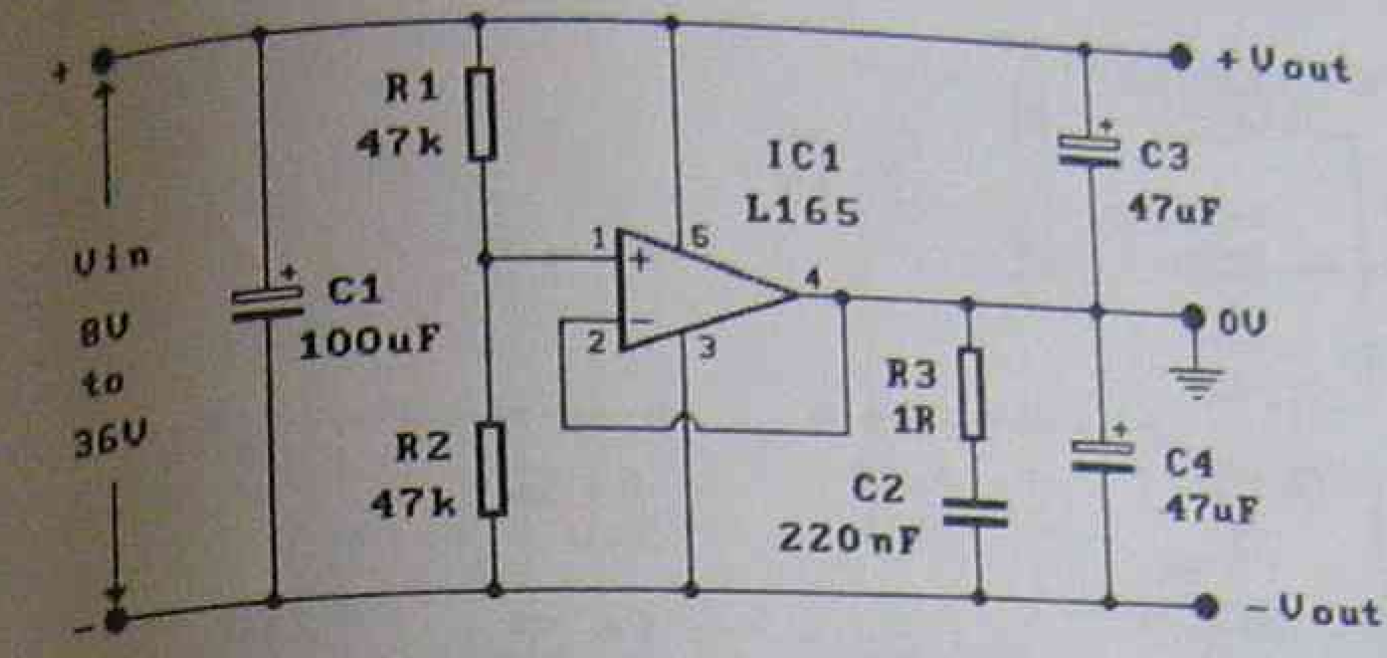


Fig.2: Dual rail supply from a single supply

- For the circuit of Fig.2:
  - Determine the reference voltage at the non-inverting terminal.
  - Identify the circuit configuration used for IC1.

VREF = ______ V

Circuit configuration IC1 = ____________________

- Connect the circuit of Fig.2. .
- Apply a DC supply voltage with a value between the limits of 8V and 36V (suggest 20V) using a power supply able to deliver 1A or more.
- Measure and record the two output voltages (+Vout and -Vout) with respect to the 0V output.

+Vout = ______ V -Vout = ______ V

- Connect a 12 ohm 10W load resistor between the 0V and +Vout terminals and quickly measure the voltage between these terminals. Also measure the voltage between the -Vout terminal (with respect to 0V).

+Vout = ______ V -Vout = ______ V

- Observe the effect on these voltages as the op amp heats up.

Step 3: A Power Oscillator

- Connect the circuit of Fig.3. Don't connect a load to the circuit at this stage.

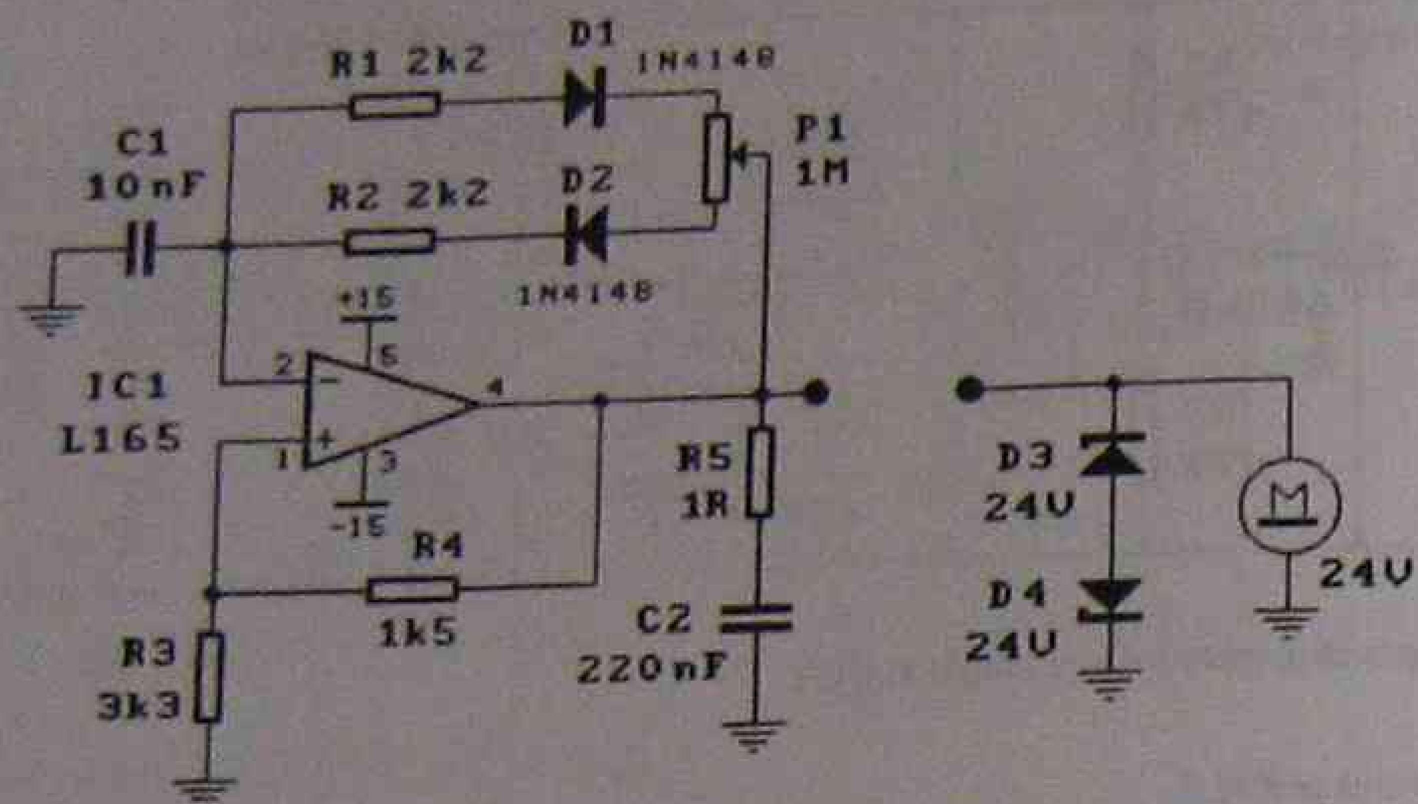
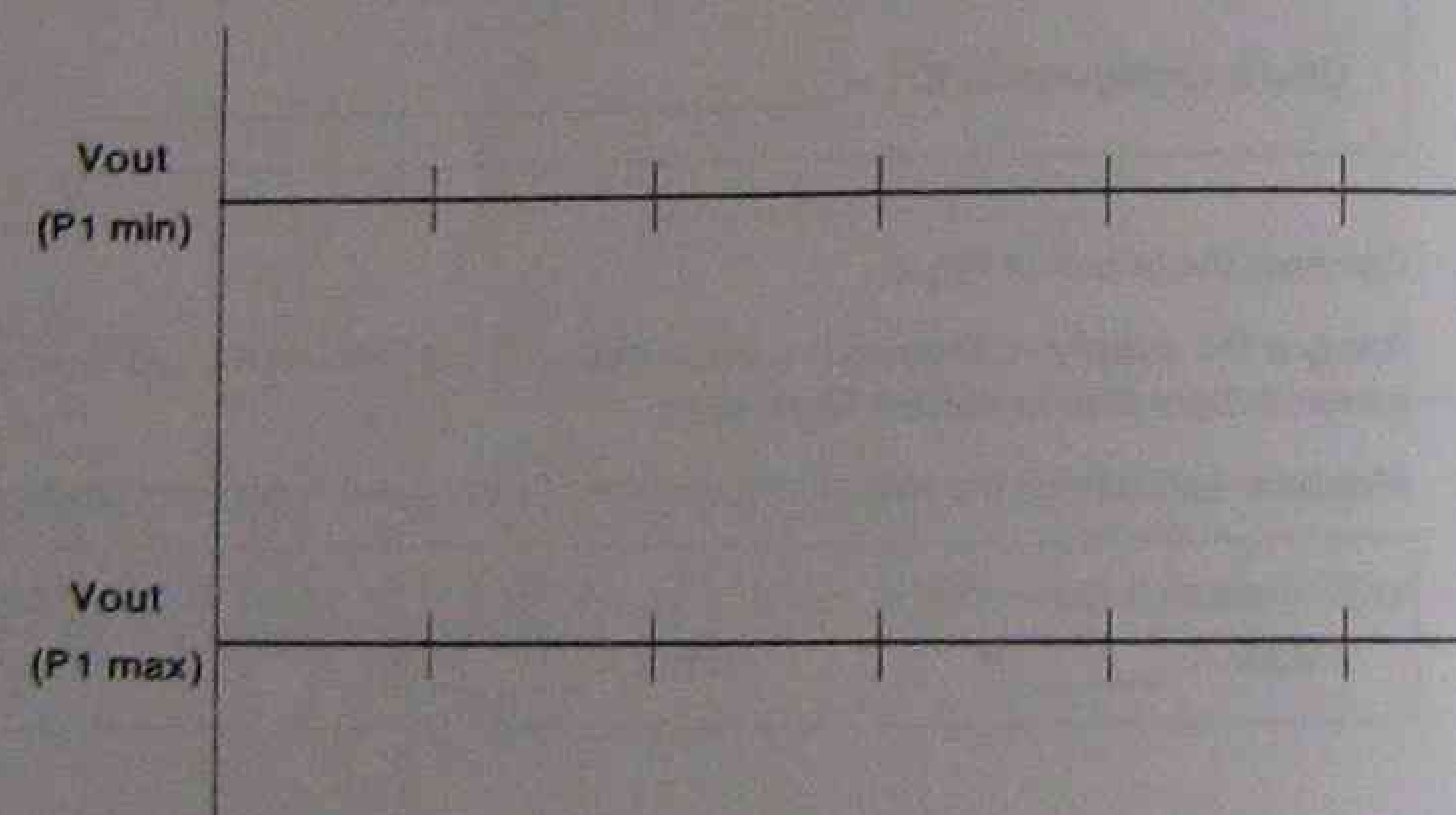


Fig.3: Power square wave oscillator, with adjustable duty cycle

- Connect channel 1 of an oscilloscope to measure Vout. Sketch the waveforms for both settings of P1 on graph set 1. Record the time intervals and all relevant voltage levels.

Graph set 1



- Connect a 24V DC motor as shown in Fig.3.
- Vary P1 from minimum duty cycle to maximum duty cycle and observe the effect on the DC motor. Measure and record the average (DC) voltage across the motor for both settings.

+Vmotor = ______ V   -Vmotor = ______ V

CONCLUSION

(a) From the results of Table 1, calculate the power dissipated in both op amps and the load, when Vin1 = 0V and Vin2 = 5V.

P_{IC1} = ______ W P_{IC2} = ______ W P_{LOAD} = ______ W

(b) For Fig.1, explain the purpose of the diodes D1 to D4.

______

______

(c) For Fig.1 and Fig.3, explain the purpose of the 1 ohm resistor and 220nF capacitor connected between the output of the op amp and common.

______

______

(d) For Fig.3, briefly explain the purpose of the zener diodes connected across the motor.

______

______

______

(e) For the circuit of Fig.2, briefly explain why the negative output voltage varies when a high load current is drawn from the positive supply, irrespective of heat dissipation by the op amp

______

______

______

(f) For Fig.3, describe how the speed of the DC motor is controlled. What causes the variation in the voltage applied to the motor?

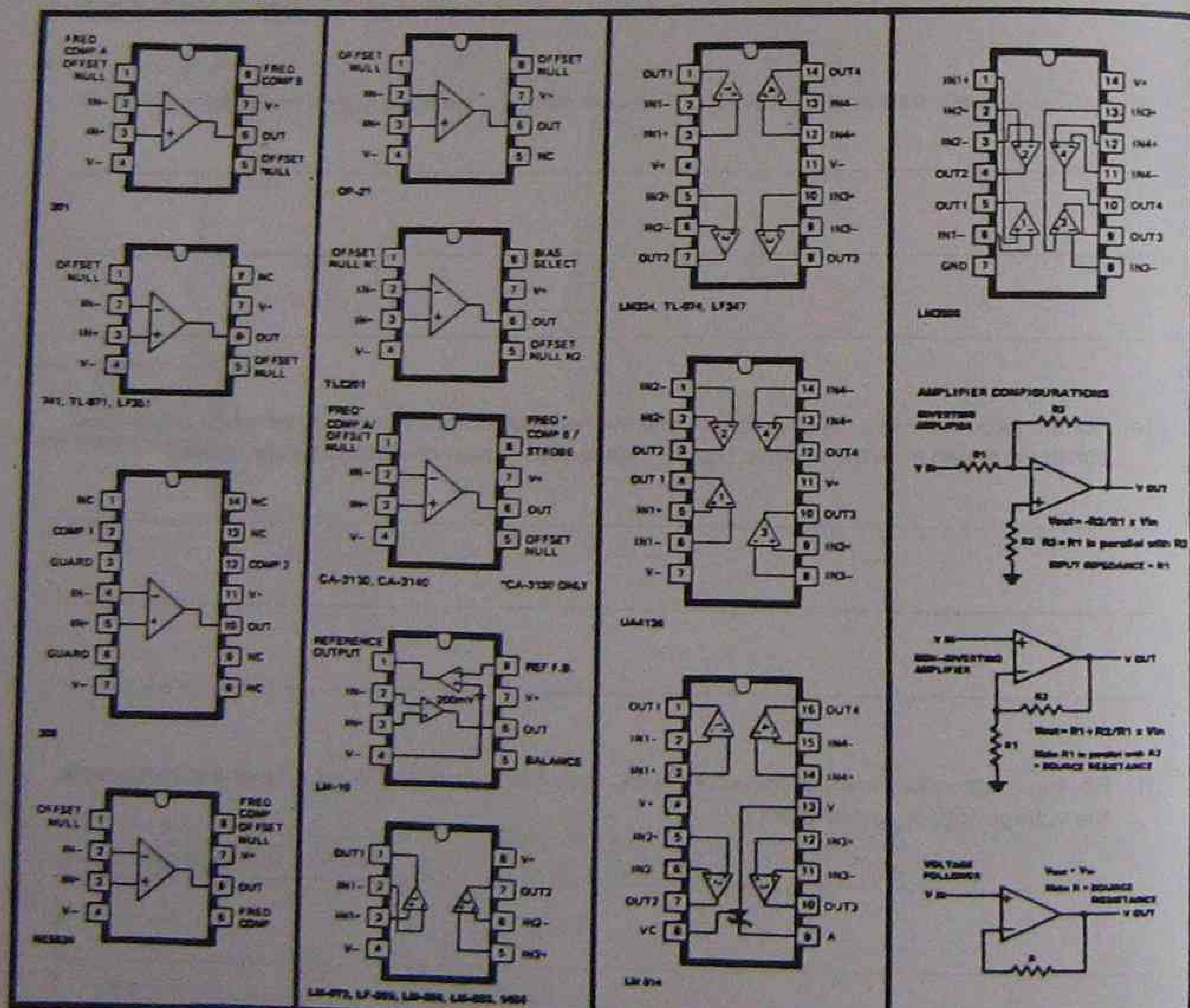
______

______

______

Type	Device	Input Offset Volt Max (mV)	Input Offset Current Max (nA)	Input Bias Current Max (nA)	Volt Gain Min (Volts/V)	Unity Gain B.W. Type (MHz)	Slew Rate Av = 1 Typ (V/uS)	Output Volt Swing RL = 10k (Vs = +/-15) (V)	Supply Volt Min (V)	Supply Volt Max (V)	CMRR Min (dB)	Differential Input Volt (V)	Supply Current Ta = 25C (Note 1) Max (mA)	Internal Freq. Comp.
	SINGLE OP AMPS													
LM301	G.P. Amplifier	10	70	300	15k	1	0.5	+/- 12	+/-3	+/-18	70	+/-30	3	No
LM741	G.P. Amplifier	7.5	300	800	15k	1	0.5	+/- 12	+/-3	+/-18	70	+/-30	2.8	Yes
LM308	Low Input Current	10	1.5	10	15k	1	0.3	+/- 13	+/-2	+/-18	80	+/-1.0(Note 2)	0.8	No
NE5534	Low Noise	5	400	2000	15k	10	6	+/- 12	+/-3	+/-22	70	+/-0.5(Note 2)	8	No
OP27	Lo Noise Precision	0.25	135	150	450k	8	1.7	+/- 11	+/-4	+/-22	96	+/-0.7(Note 2)	6.7	Yes
TL071	JFET Lo Noise	13	2	7	15k	3	13	+/- 12	+/-3.5	+/-18	70	+/-30	2.5	Yes
LF351	JFET Lo Noise	13	4	8	15k	4	13	+/- 12	+/-4	+/-18	70	+/-30	3.4	Yes
LTC251	Prog Lo Power CMOS	10	0.3	0.8	7.5k	0.7	0.8	7.8(Vs=10V)	1.4	18	60	+/-18	0.22	Yes
CA3130	MOSFET Input	15	0.03	0.05	50k	4	10	-/-8(Vs=+/-8)	+/-2.5	+/-8	70	+/-8	15	No
CA3140	MOSFET Input	15	0.03	0.05	20k	4.5	9	+/- 12	+/-2	+/-18	70	+/-8	6	Yes
LM10C	Inc. Voltage Ref	3	3	40	15k	0.1	*	+/- 14.8	1.1	45	87	+/-40	0.27	Yes
LM3080	Transconductance	5	500	5000	5 ammho	2@Ac1	50	+/- 12	+/-2	+/-18	80	+/-5	typ 1.1	No
	DUAL OP AMPS													
LM1458	G.P. Amplifier	7.5	300	800	15k	1	0.5	+/- 12	+/-3	+/-18	70	+/-30	5.6	Yes
LM358	Low Power	9	150	500	15k	1	0.5	+/- 12	+/-1.5	+/-16	65	32	2	Yes
LM833	Hi-Fi	5	200	1000	30k	15	7	+/- 12	+/-5	+/-18	80	+/-30	8	Yes
TL072	JFET Lo Noise	10	2	7	15k	3	13	+/- 12	+/-3.5	+/-18	70	+/-30	5	Yes
LF353	JFET Lo Noise	10	4	8	15k	4	13	+/- 12	+/-4	+/-18	70	+/-30	4.5	Yes
LM13600	Transconductance	4	600	5000	5 ammho	2@Ac1	50	+/- 12	+/-2	+/-18	80	+/-5	typ 2.6	No
	QUAD OP AMPS													
LM324	G.P. Amplifier	9	150	500	15k	1	0.5	+/- 13	+/-1.5	+/-16	65	32	3	Yes
UA4136	G.P. Low Noise	7.5	300	800	15k	3	1	+/- 12	+/-2	+/-18	70	+/-30	11	Yes
TL074	JFET Lo Noise	10	2	7	15k	3	13	+/- 12	+/-3.5	+/-18	70	+/-30	10	Yes
LF347	JFET Lo Noise	10	4	8	15k	4	13	+/- 12	+/-4.5	+/-18	70	+/-30	11	Yes
LM614	Inc. Volt Ref	7	50	250	25k	>1	0.8	12.4	+3	+36	80	36	typ 0.8	Yes
LM2900	Norton Amp	*	*	200	1.2k	2.5	20	10	+4	+32	*	*	10	Yes

* Not Specified Note 1: Supply current for all channels in the package Note 2: Inputs have shunt diode protection, current must be limited.



Scientists have just discovered that the active ingredient in IC's is smoke, when it leaks out... they stop.

OPERATIONAL AMPLIFIER DATA

The following data is for the more common types of operational amplifiers. The specifications shown are the worst case values, and have been derived from a number of data manuals. The description attempts to generally describe the amplifier, although manufacturer's notes will vary in their opinion of the IC. For example, most op amps are listed by their makers as being 'high performance'. Many are also shown as 'low noise', and an actual noise figure is the only way to determine if that amplifier is a true low noise device. Different terminology is often used by manufacturers, for example the proprietary term 'BIFET' refers to an op amp that uses both FETs and bipolar transistors. Some amplifiers are listed as being suitable for use with a single rail supply, and have a different circuit configuration to those listed as dual supply only. However, any op amp can be used with a single supply with voltage divider biasing at the input.

Type	Description	Av (min) V/mV	Slew rate V/µs	BW (Av=1) MHz	Ibias (max) nA	Vs V ± min	Vs V ± max	Is (max) MA
LM301	General purpose — controllable	25	0.5	1	250	5	18	3
741	General purpose audio	20	0.5	1	500	3	18	2.8
TI071	Low noise JFET input	50	13	3	0.2	5	18	2.5
LF351	JFET input	25	13	4	0.2	5	18	3.4
CA3130	MOSFET input-ext comp	50	10	4	0.5		8	15
CA3140	MOSFET input-int comp	20	9	4.5	0.5		18	5.5
NE5534	Low noise — high performance	25	13	10	1500	3	22	8
TL072	Dual TL071	25	13	4	0.2	5	18	5
LF353	Dual LF351	25	13	4	0.2	5	18	6.5
MC1458	Dual 741 — no offset	20	0.8	1.1	500	3	18	6.6
MC4558	Dual general purpose	20	3	1.5	500	3	18	6.6
LM833	Low noise — audio (dual)	30	5	9	1000	5	18	8
NE5532	Dual low noise — high perf.	25	9	10	800	3	20	16
TL074	Quad TL071	25	13	3	0.2	5	18	10
LM324	Quad general purpose	25	0.5	1	250	1.5	16	1.2
LF347	Quad LF351	25	13	4	0.2	5	18	11
RC4136	Quad — similar to 741	20	1	3	500	4	18	13.7
TL075	JFET input Quad 071	25	13	3	0.2	3.5	18	10
LM3900	Quad Norton	1.2	0.5	2.5	200	+4.5	+32	10

Notes:

(1) The TL06, 07 and 08 series are all pin compatible. The 08 series is the standard, the 07 the low noise version, and the 06 series the low power types, in which the supply current is 10% of the 07 and 08 types.

(2) The supply current shown for each type is that required by the IC – not individual amplifiers in the IC.

(3) The input resistance of the LM5532 and LM5534 is very low; minimum specification is 30k ohms.

(4) The differential input voltage should never exceed the supply voltage.

(5) In most cases the output voltage swing is within 2 volts of the supply rail.

(6) Both the comparator types shown are open-collector and require a pull up resistor.