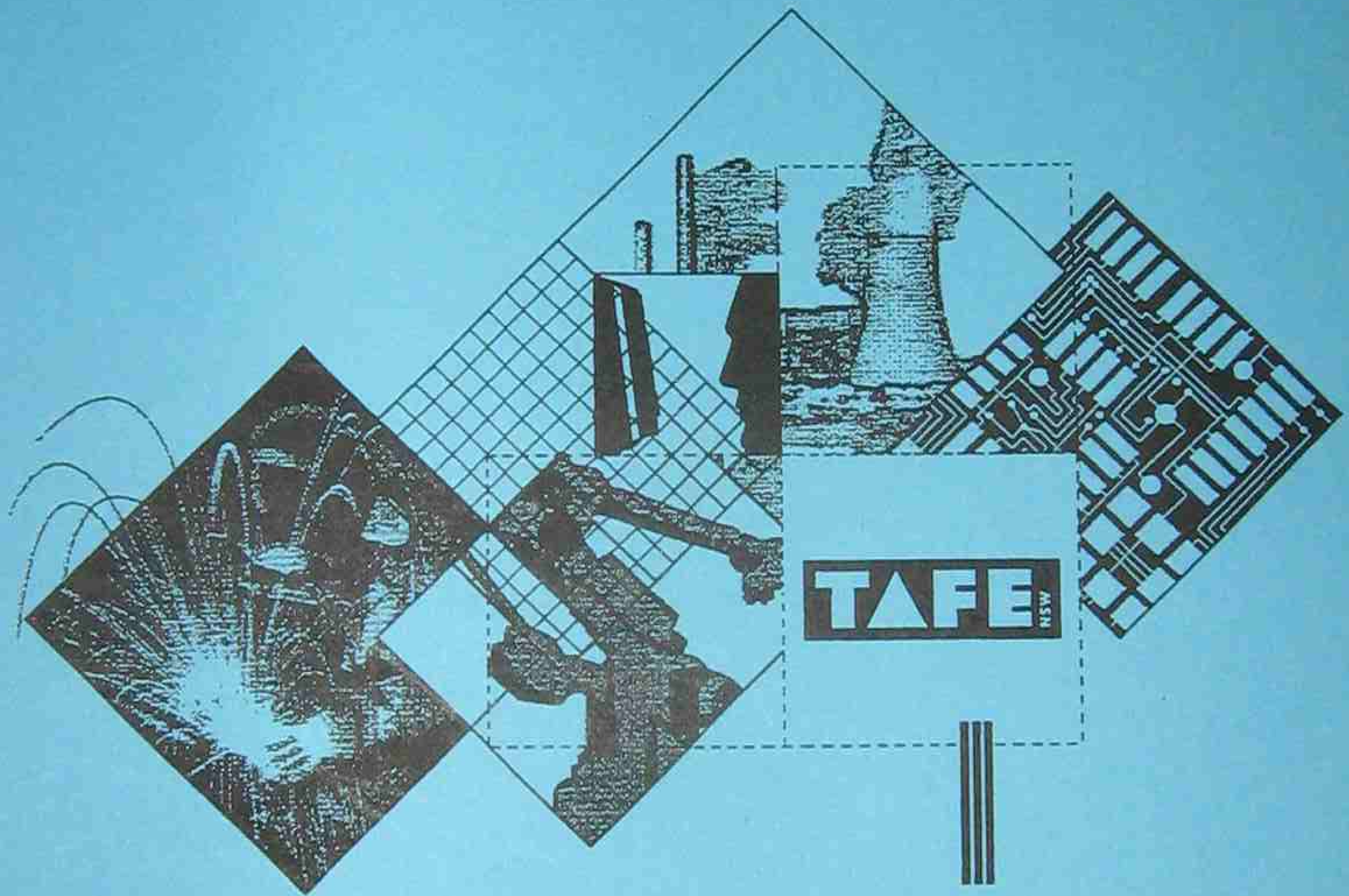
NSW Module Resource Manual for the TAFE Engineering Technician and Engineering Associate National Curriculum



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

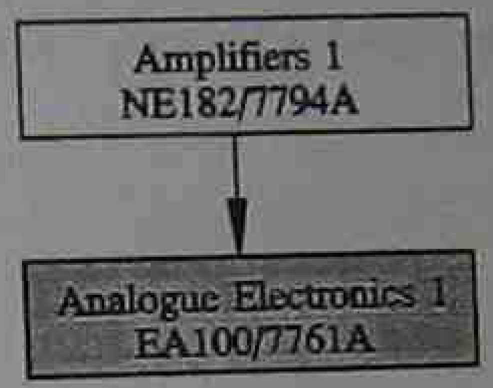
Notes

INTRODUCTION

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.

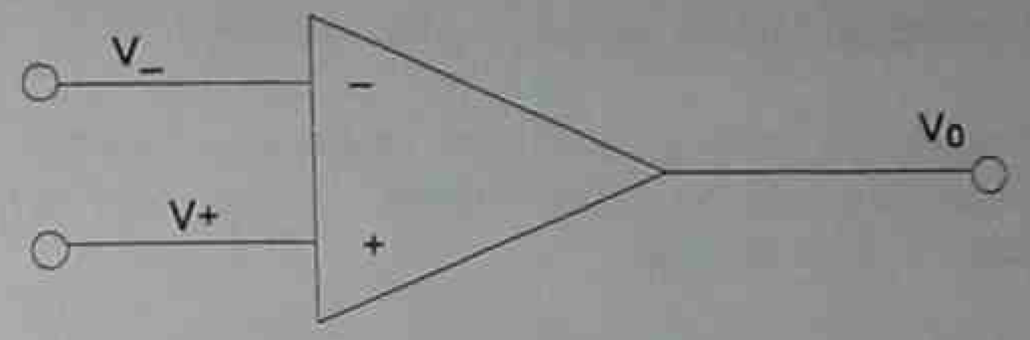


Fig. 1 Operational amplifier

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.
  $\therefore V_+ = V_-$ 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

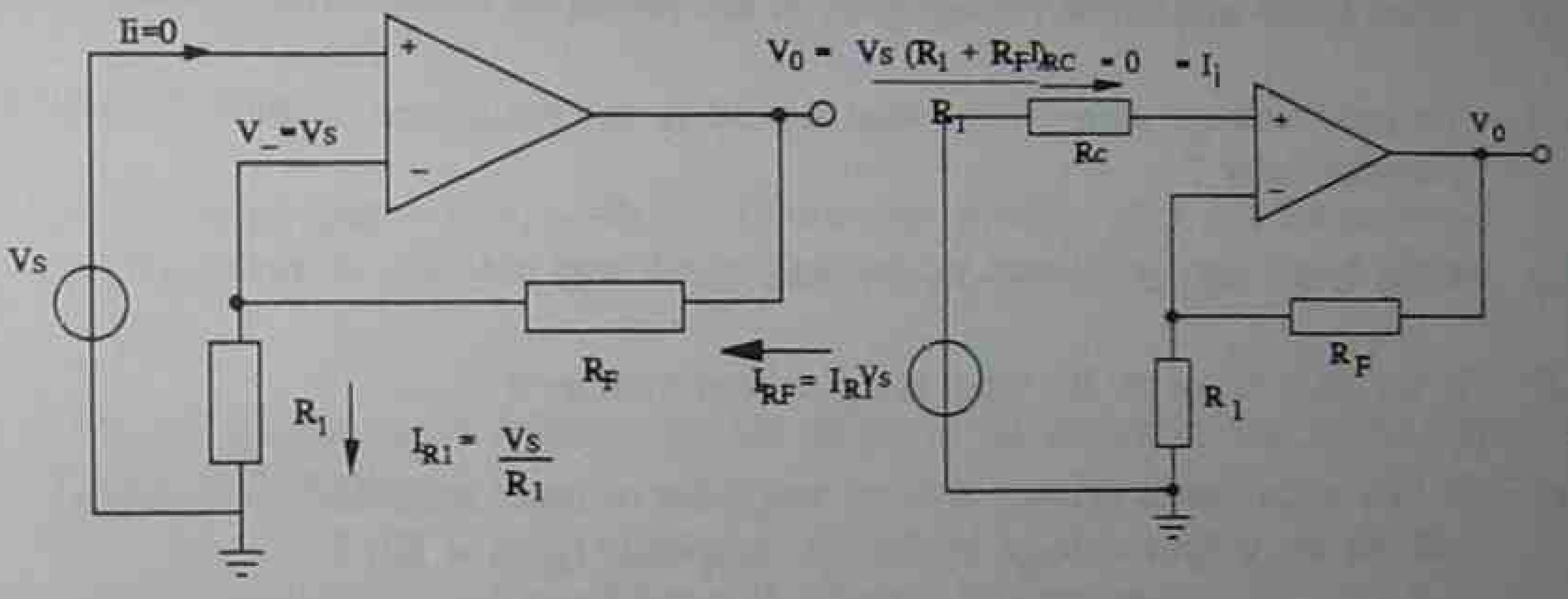


Fig. 2

Fig. 3

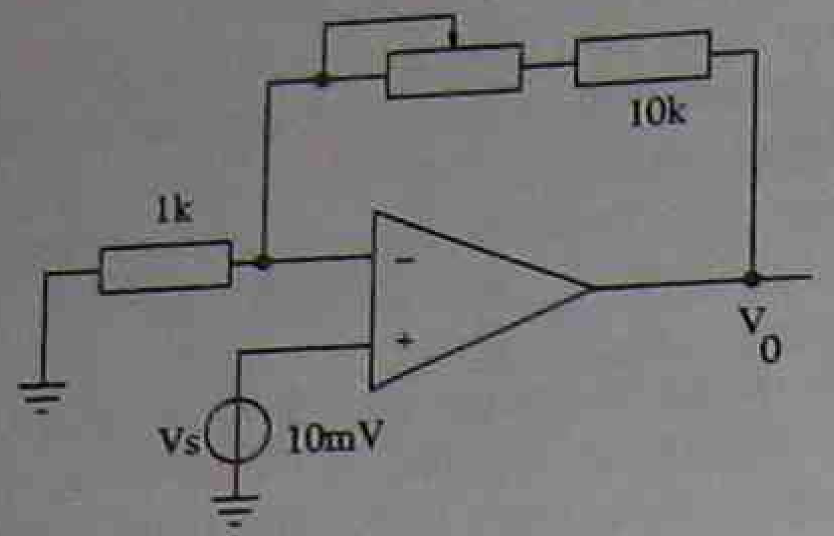
Non-inverting amplifier configurations

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

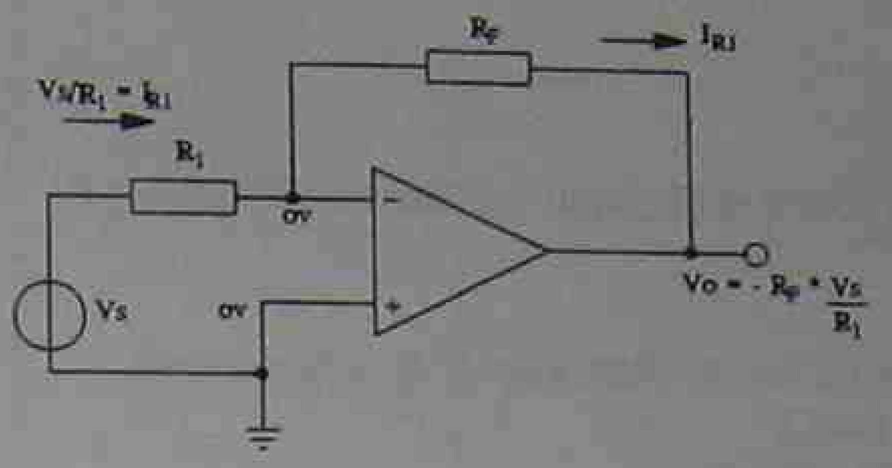


Fig. 4

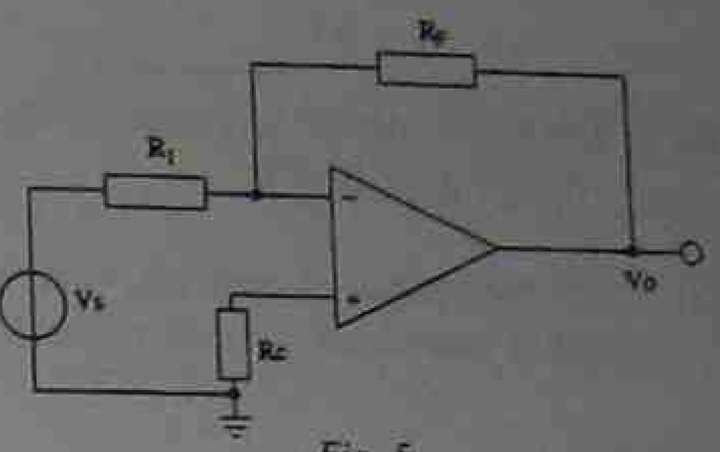


Fig. 5

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.

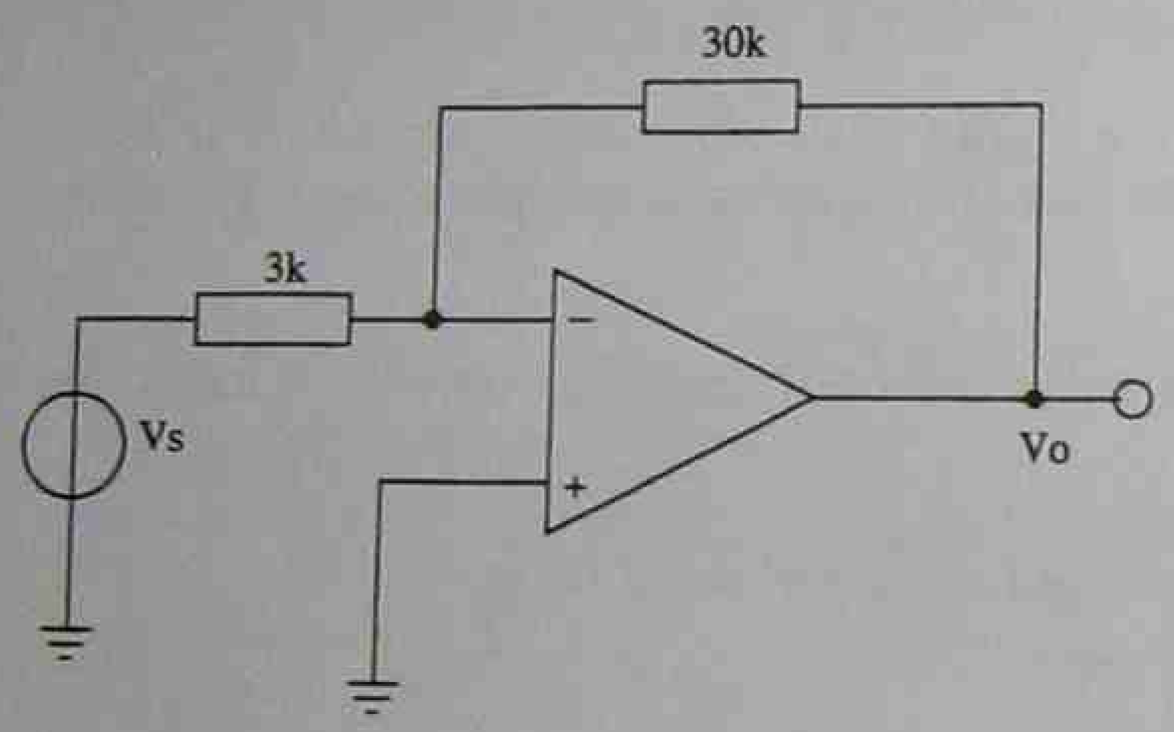
(b) For the circuit in part (a), what will be the voltage gain if it is used with a signal source of internal resistance 600Ω.

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

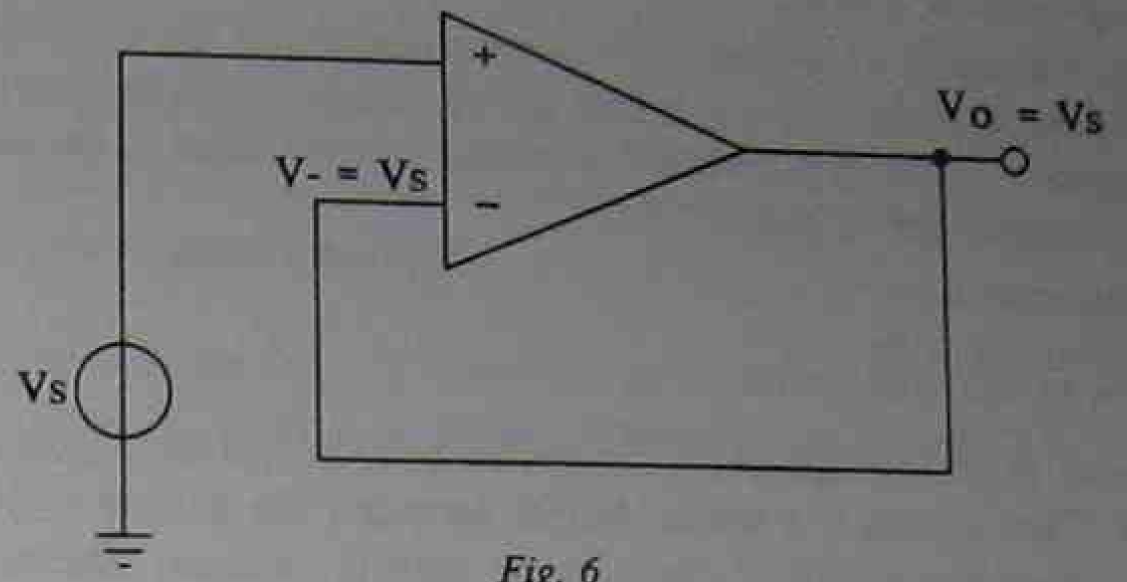


Fig. 6

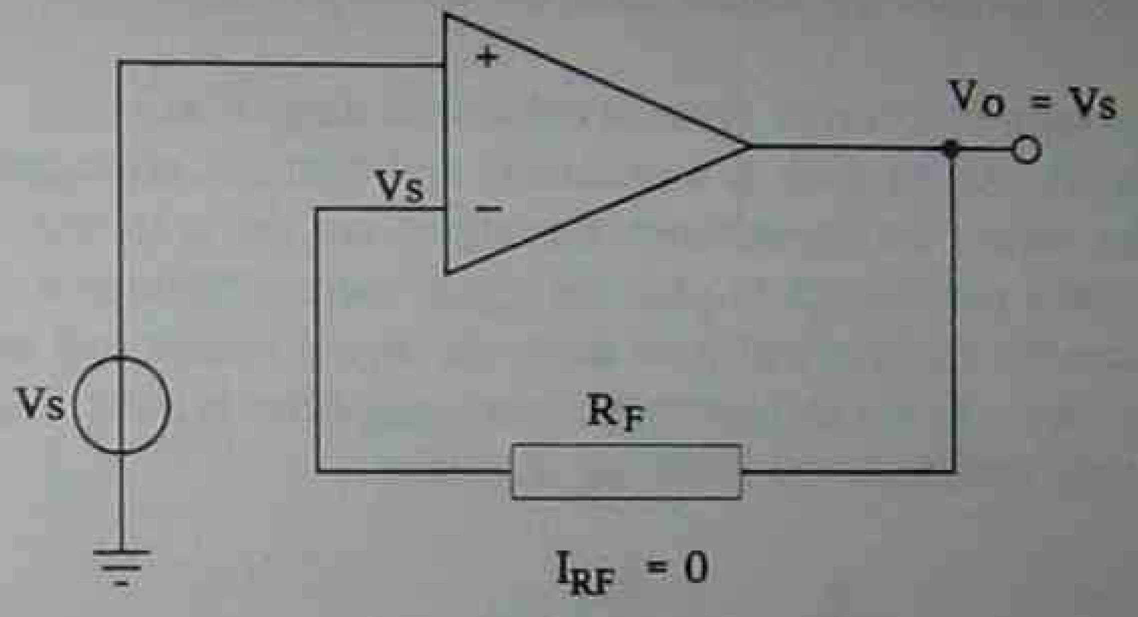


Fig. 7

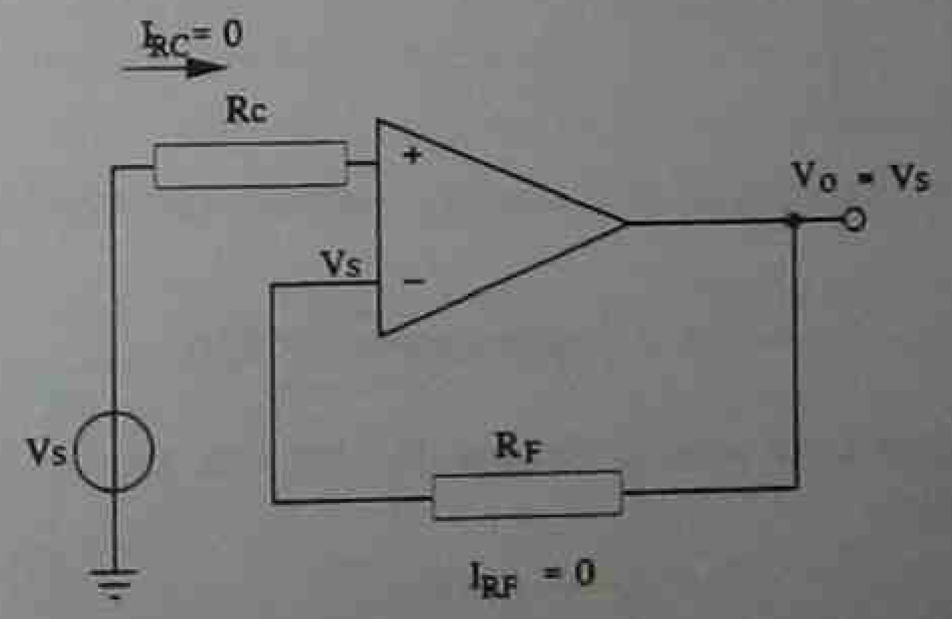


Fig. 8

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.

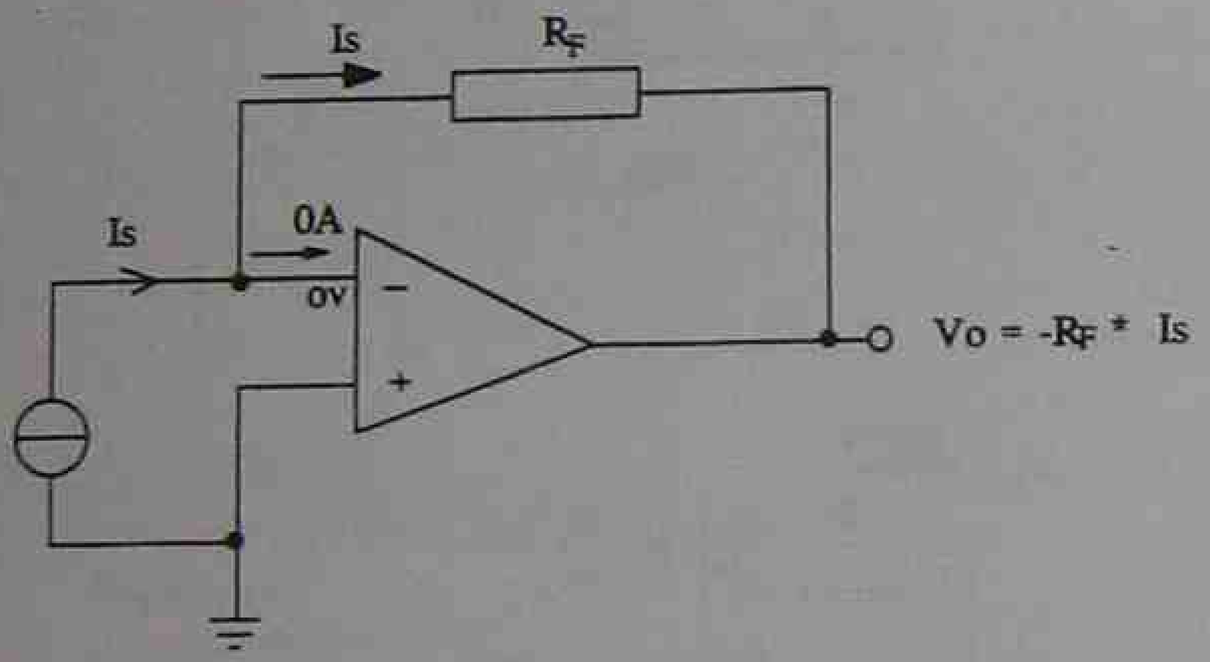


Fig. 9 Transresistance amplifier

The input current must flow around through R_F, because the op amp input does not draw any current. Also,
$V_- = V_+ = 0V$

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

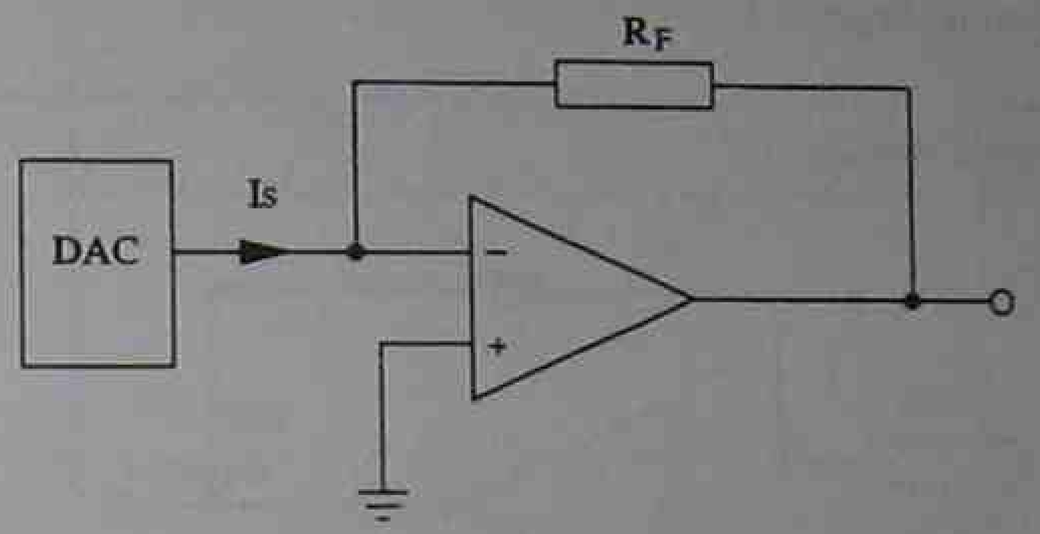
This means that $V_o \propto I_s$. The ratio V_o/I_s is called the transresistance R_T of the amplifier.

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)

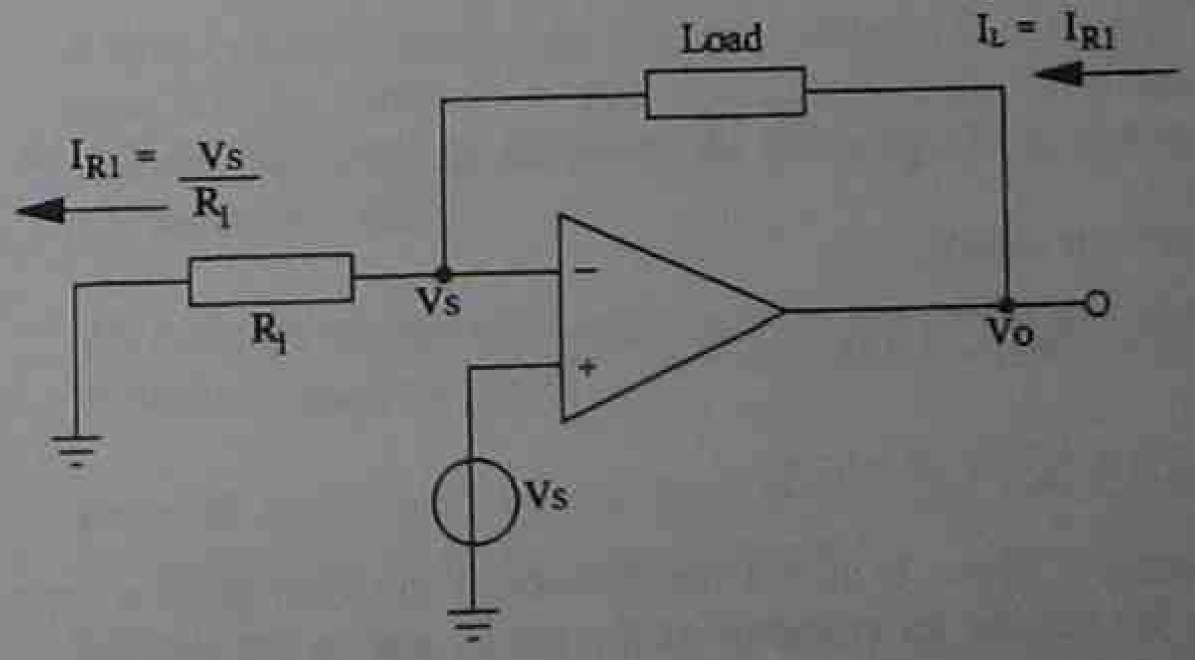


Fig. 10 Transconductance amplifier

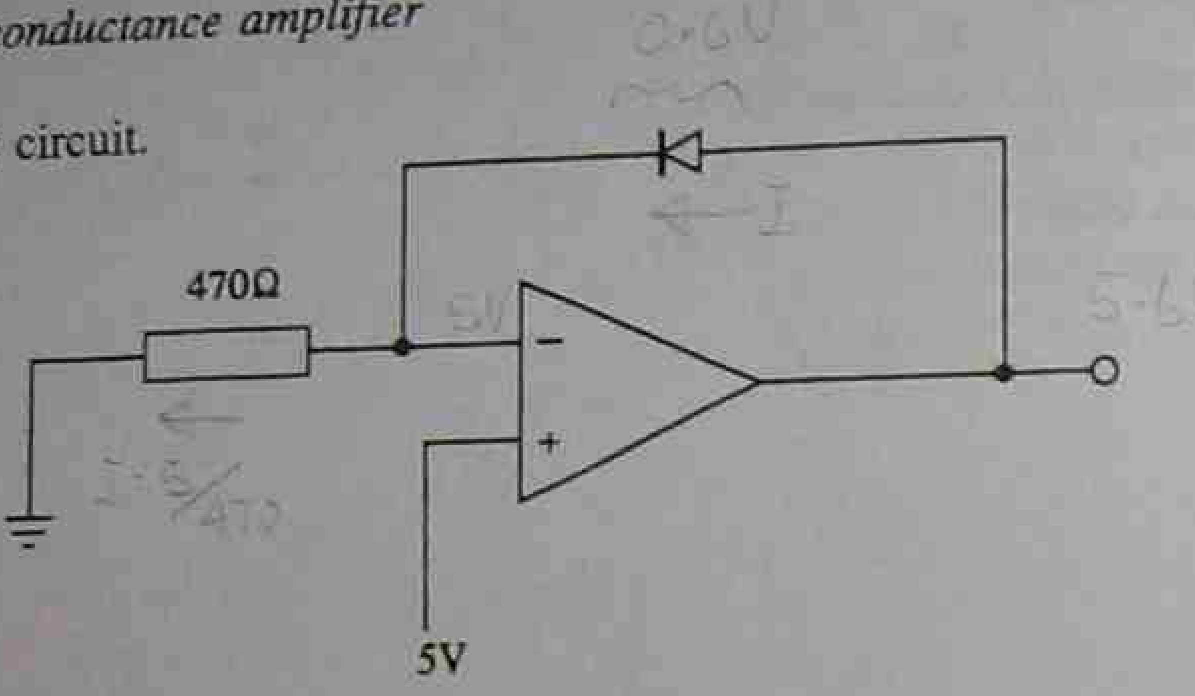
The circuit in Figure 10 is very similar to the non-inverting amplifier. Here, our aim is to pass a known current through the load, not amplifying the input voltage. The load can be any circuit components, possibly even diodes, rectifiers or meters. The purpose of the op amp is to isolate the source 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.

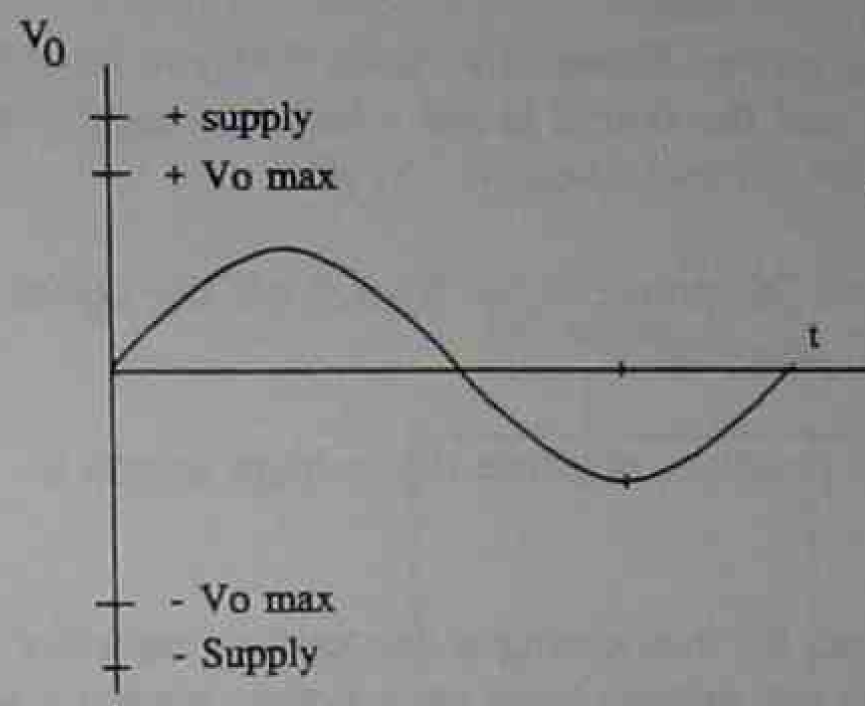


Fig. 11 Normal output

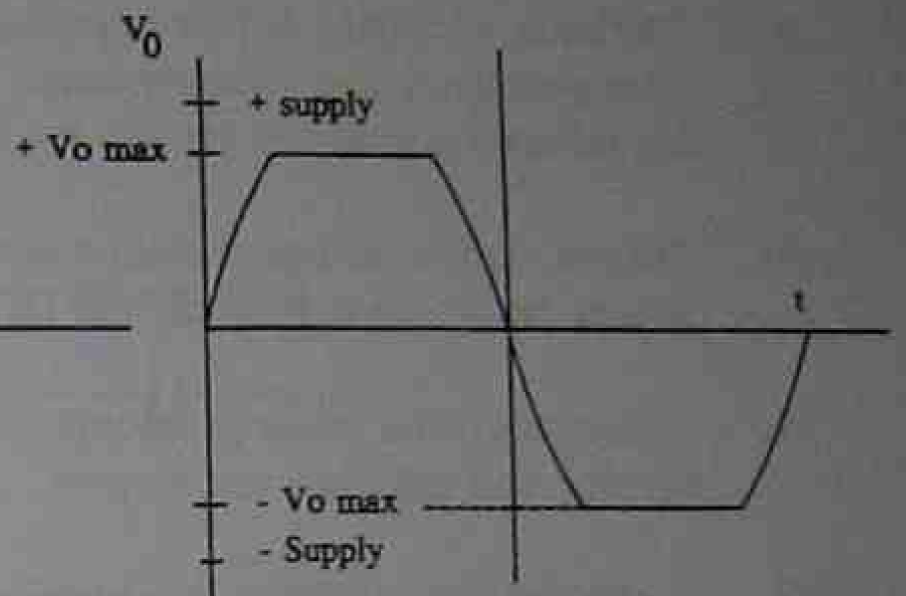


Fig.12 Clipped output

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.

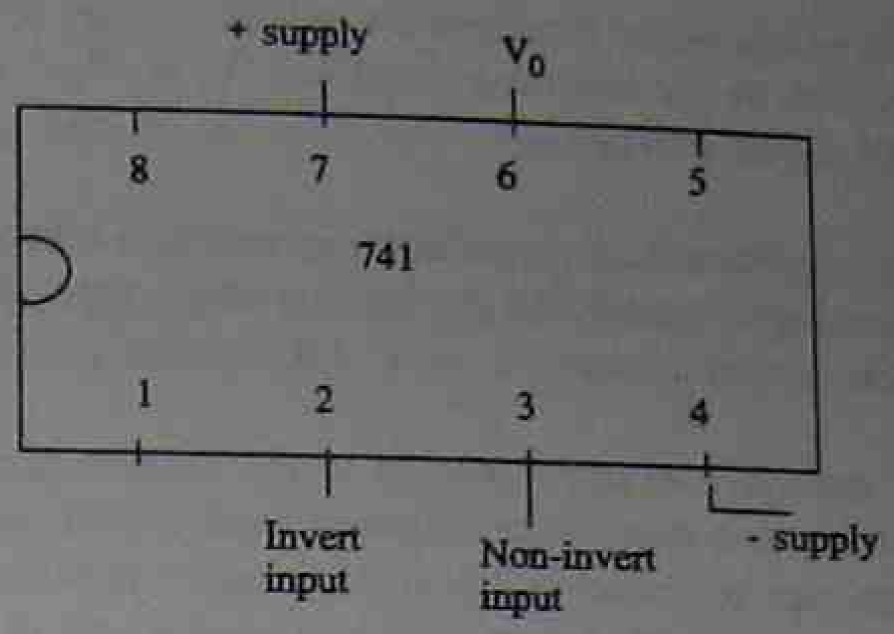


Fig.13 Pin connections for 741 and 351 type op amps

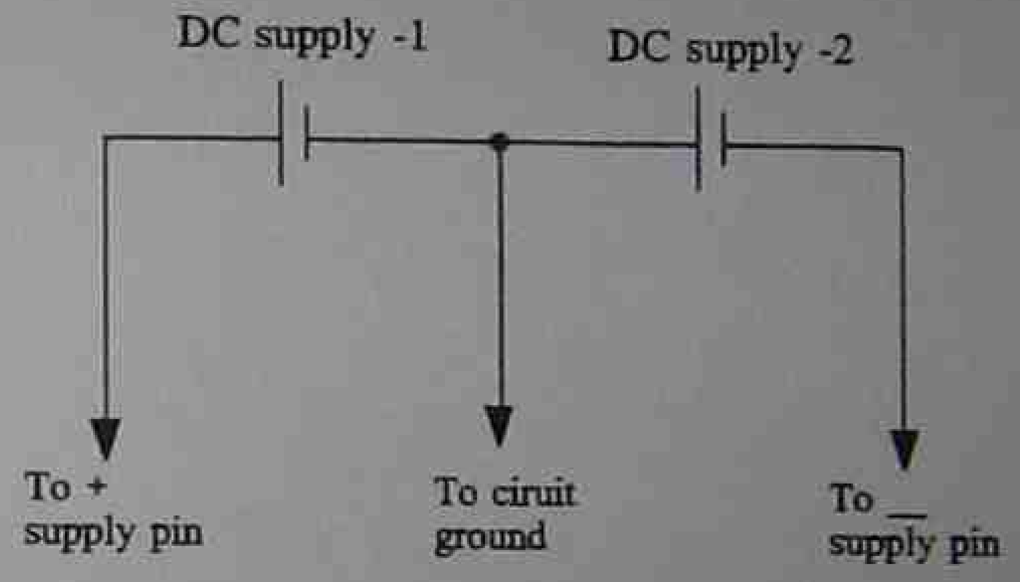


Fig.14 Dual power supply connections for op amps.

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.

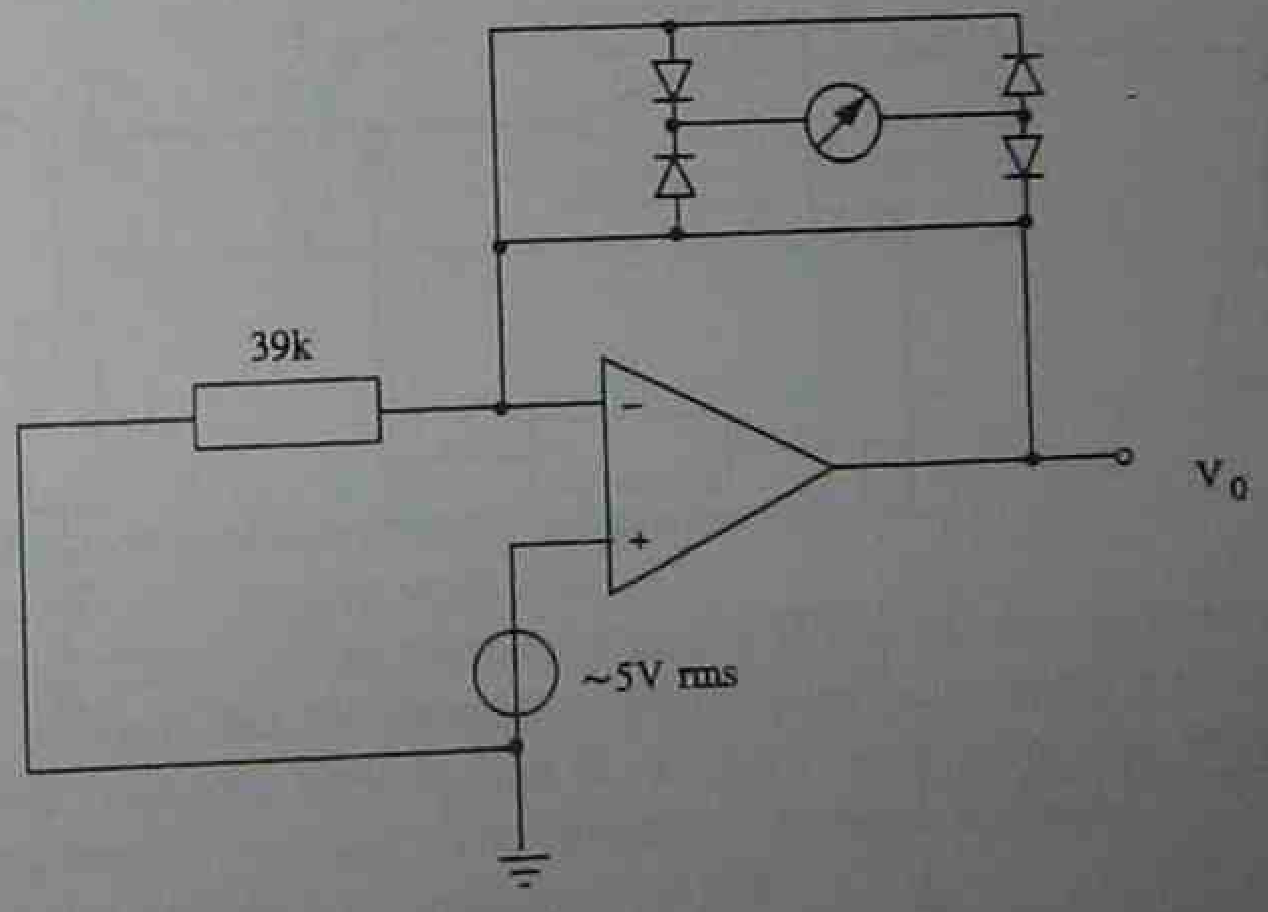
2. (a) in Q.1 how will the output voltage change if the signal source has a significant internal resistance?

(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,

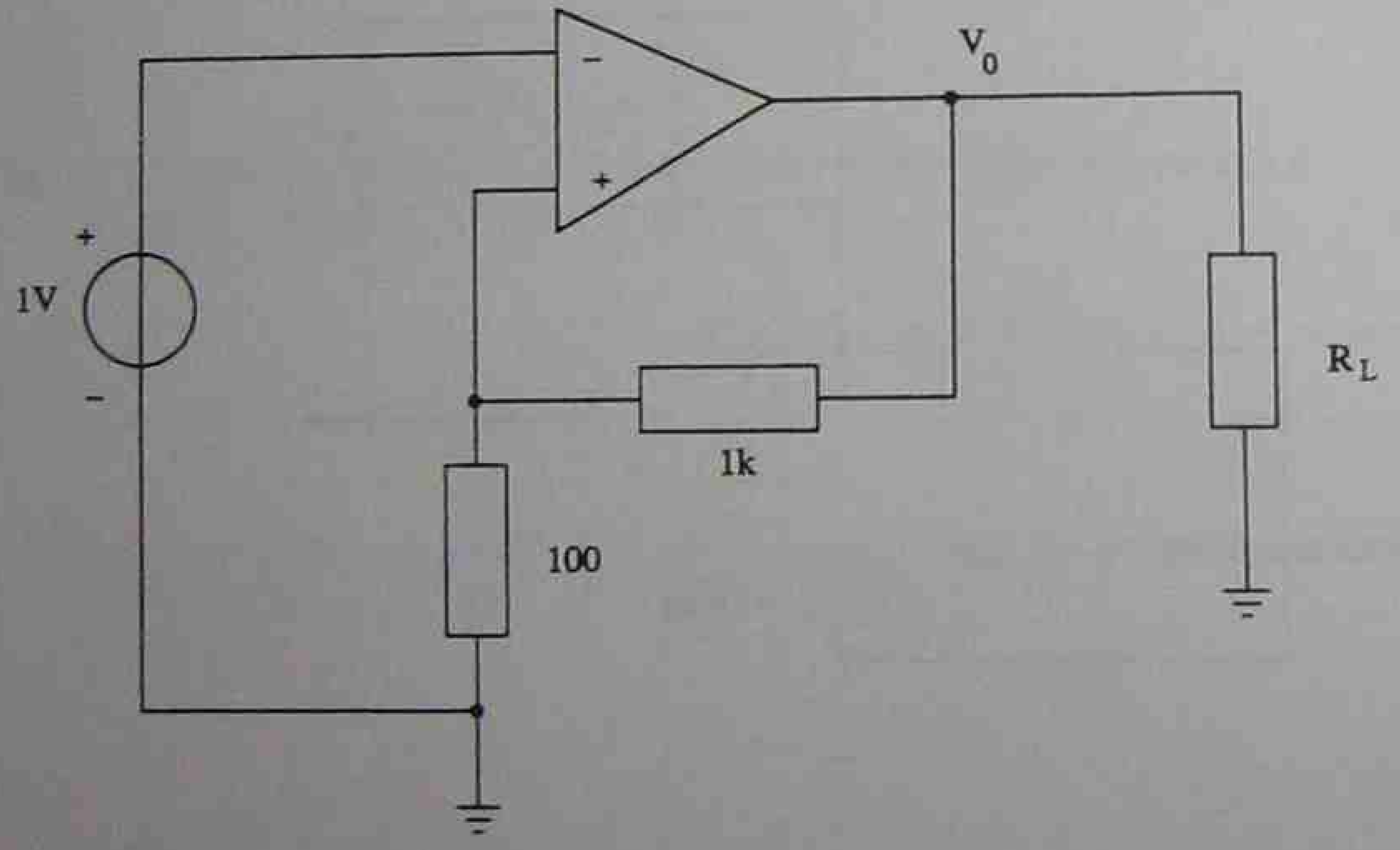
(a) What will be the average meter current if the input voltage is 5V rms sine wave? (Note: For sinewaves, full wave rectified average = 0.9 * rms)

(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

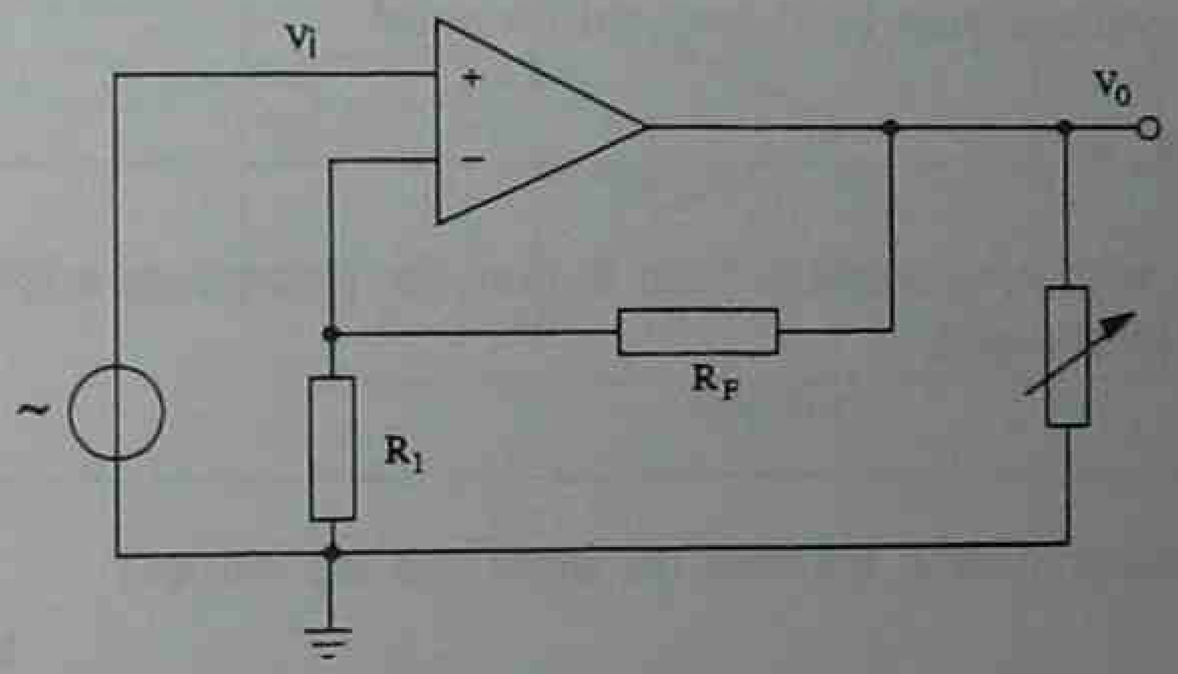
Tasks

- 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

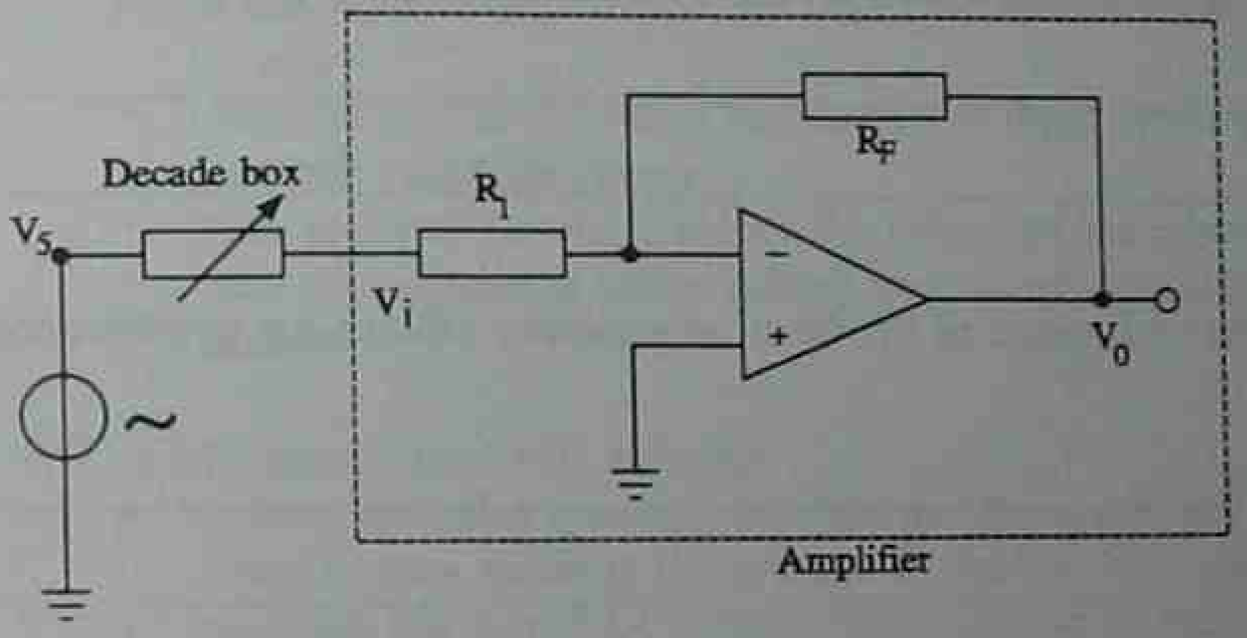
Task

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 amplifiers, 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

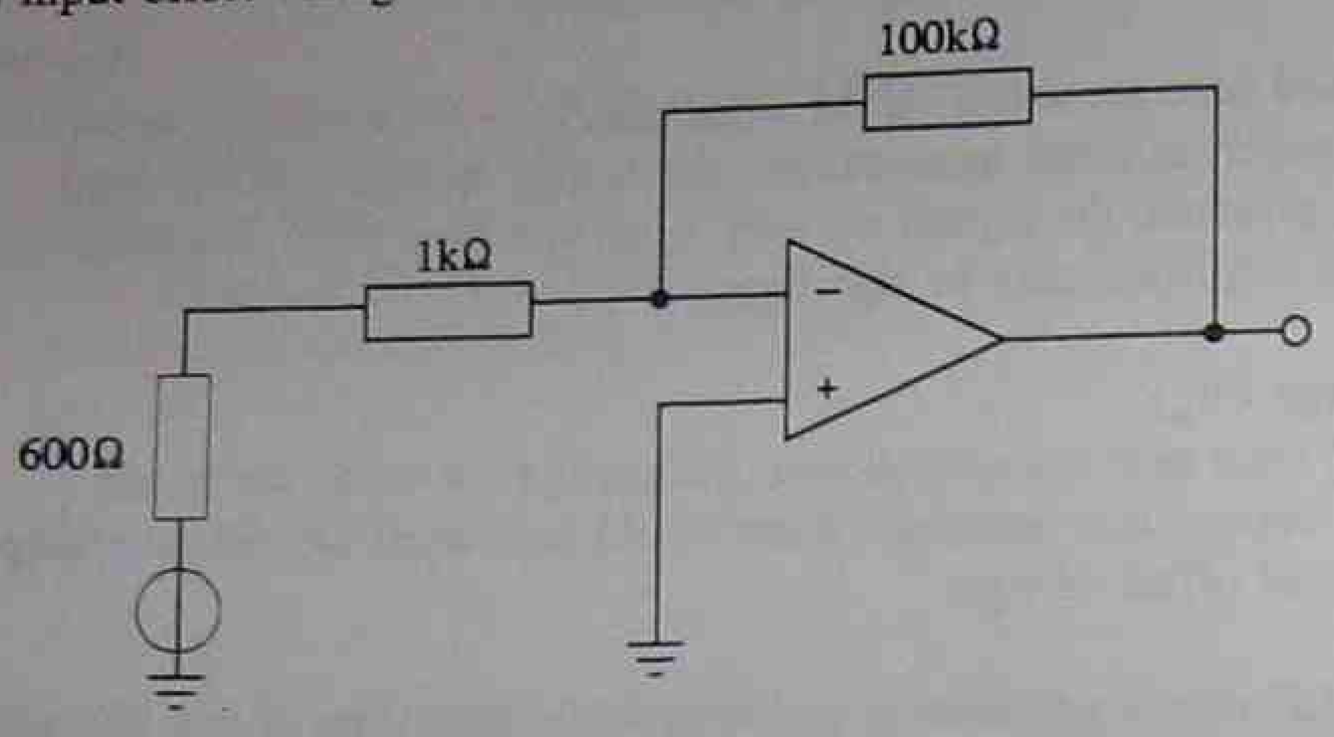


Fig. 1 Calculation of the effect of input offset voltage

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).

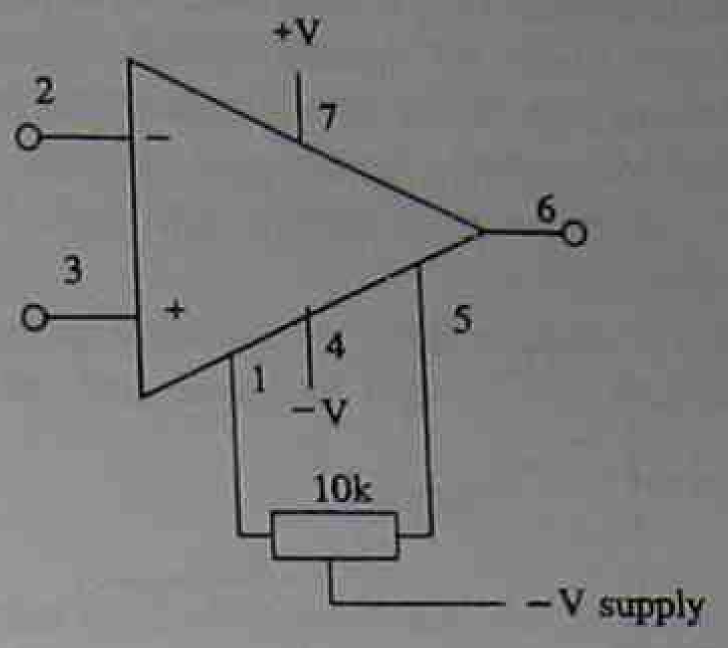


Fig. 2 Offset nulling for type 741 op amp.

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.

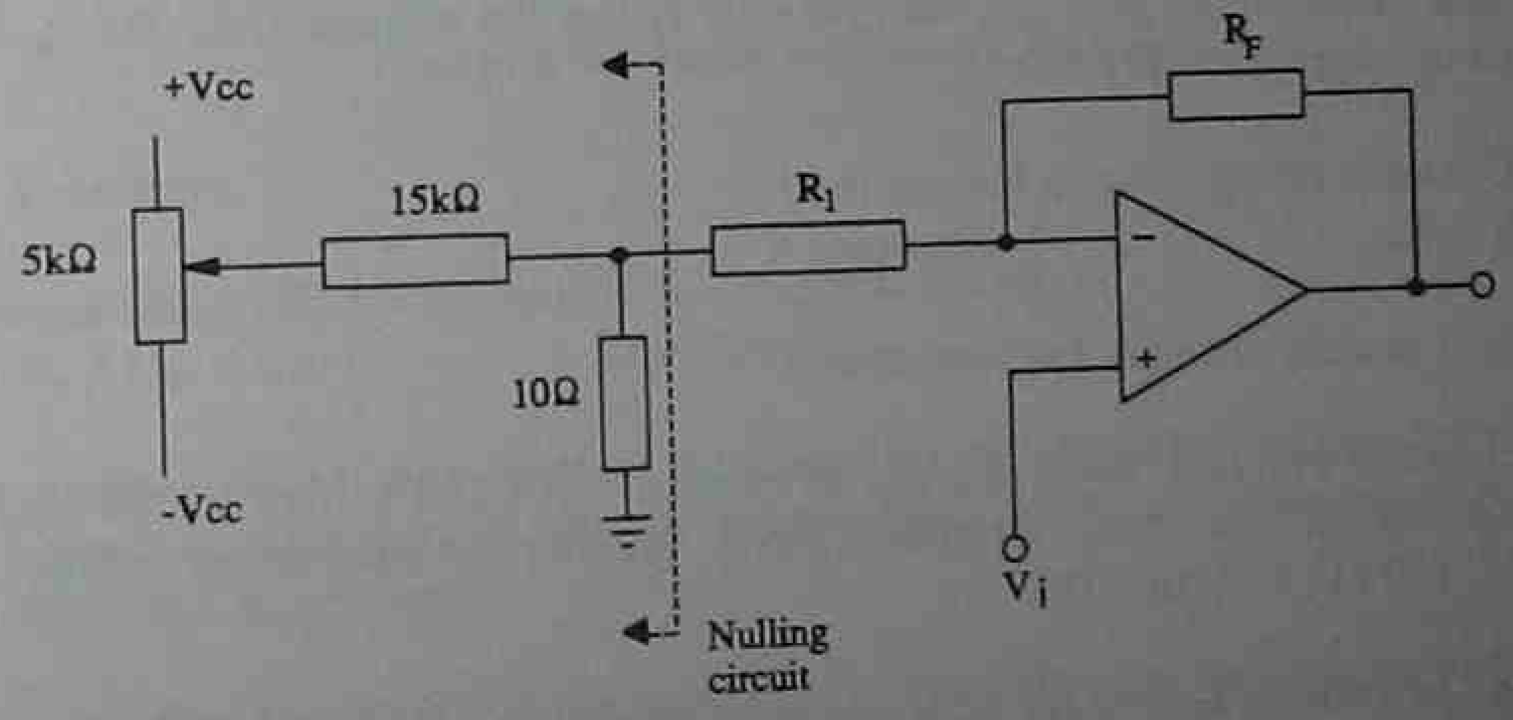


Fig. 3 Universal offset nulling circuit

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 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

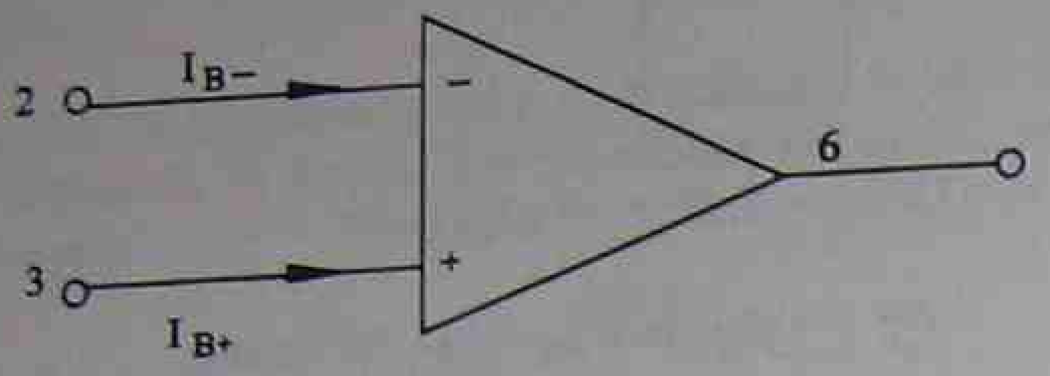


Fig. 4 Bias currents of an op amp

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).

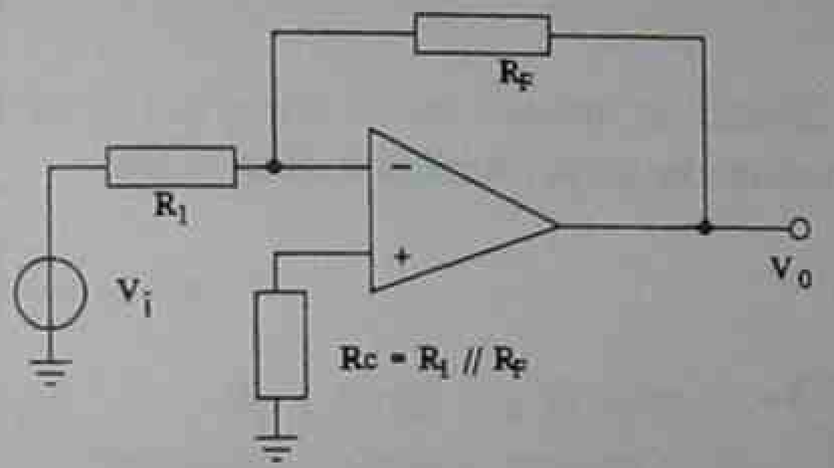


Fig. 5

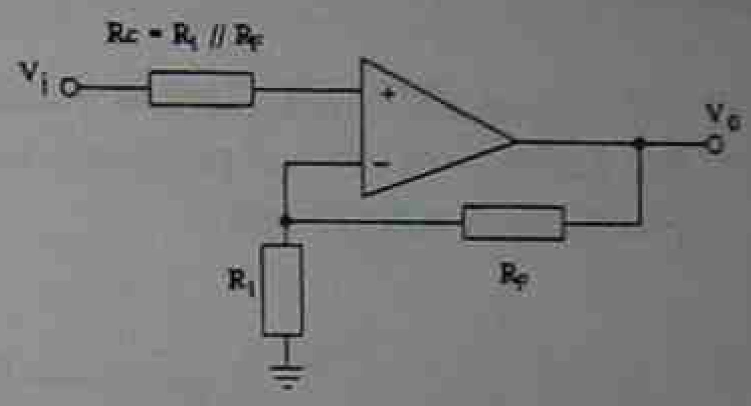


Fig. 6

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

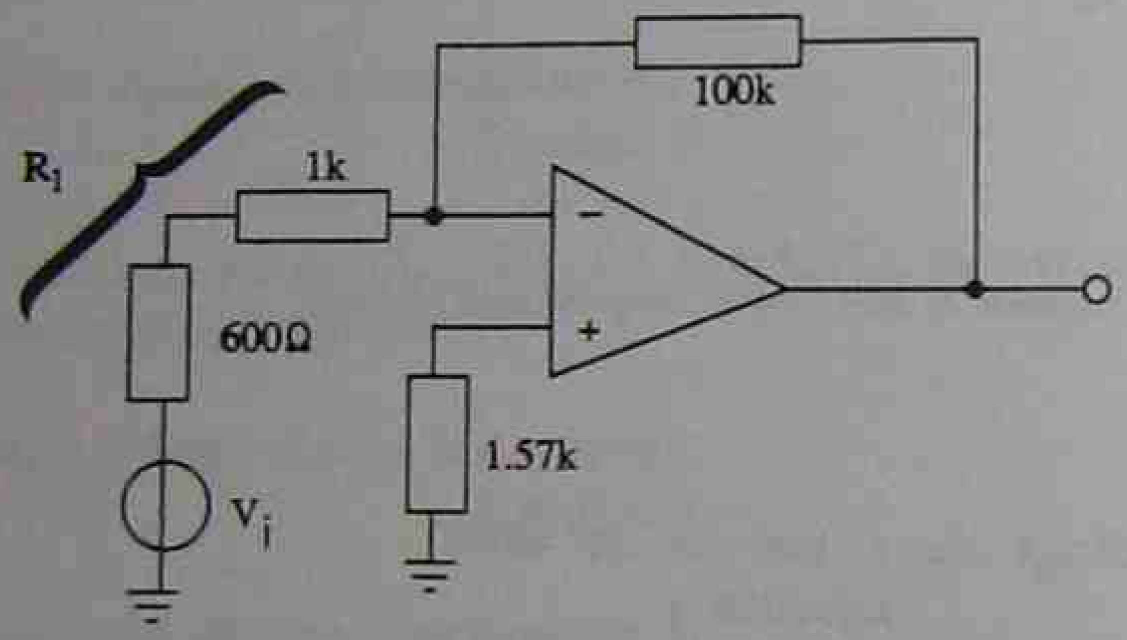
(a) For the circuit of Figure 1 design a bias current compensation.

(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) $R_c = 100\ \text{k}\Omega \parallel (1\text{k}\Omega + 600\Omega) = 1.57\ \text{k}\Omega$

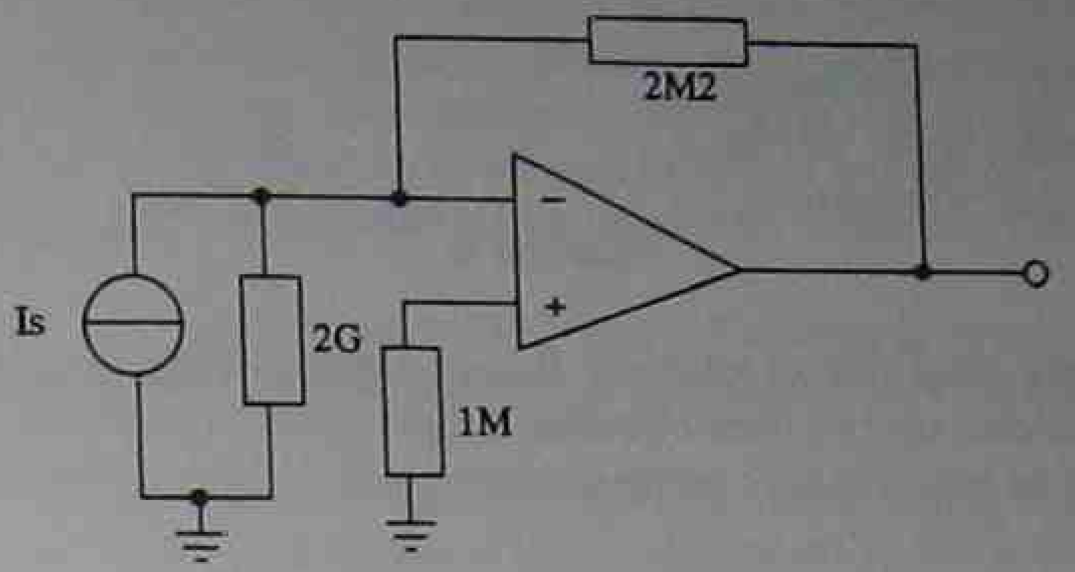
Put 1.57kΩ (or nearest preferred value) resistor in series with the +input for bias current compensation.



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

Example 2 : Bias current compensation and its effect

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



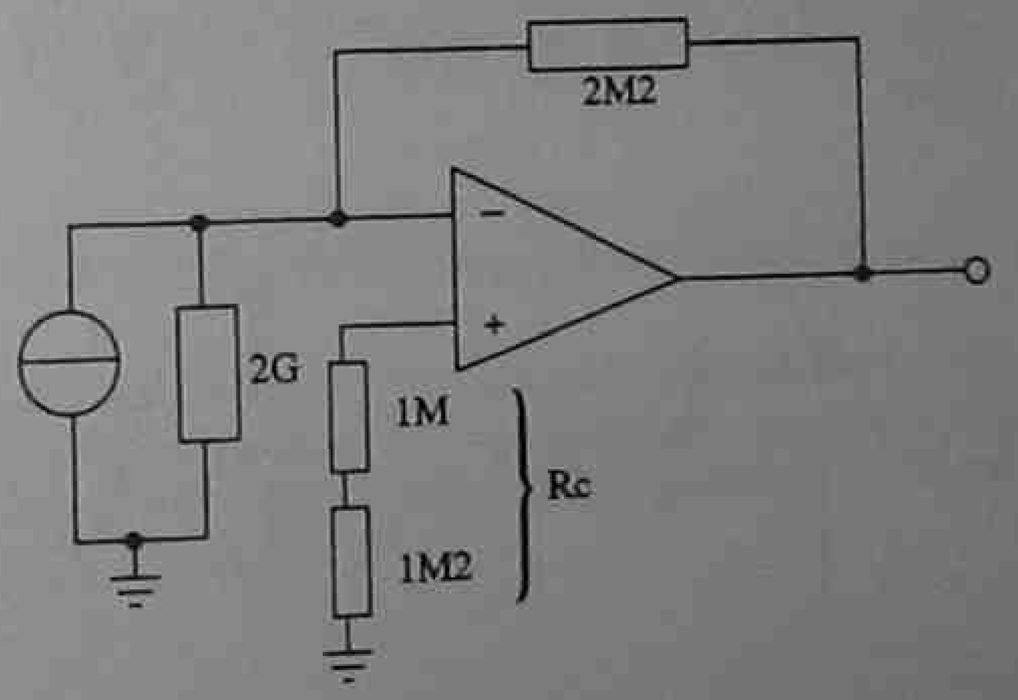
(b) If the input offset current is 10 nA, what is the output DC offset voltage due to bias currents after compensation ?

Solutions

(a) $R_c = 2\text{M}2 \parallel 2\ \text{G} = 2\text{M}2$

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

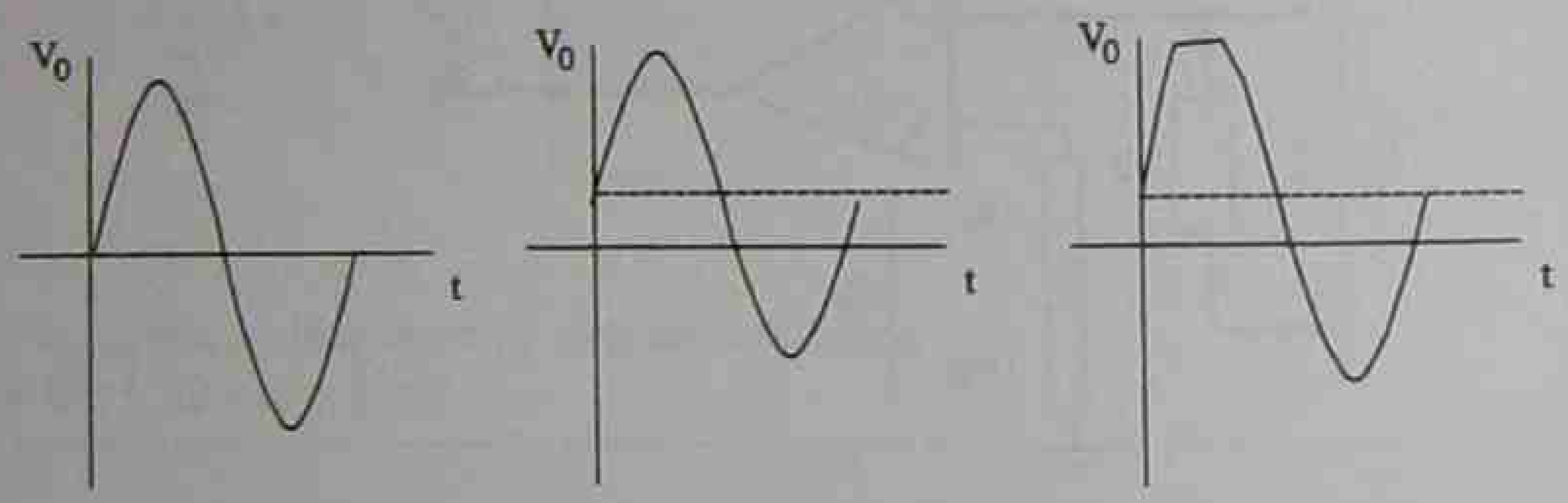
The first term on the right hand side of equation 9 is calculated using equation 1. The second term is calculated using either equation 6/equation 8 or equation 3/equation 4, depending on whether bias compensation resistor is used or not used.

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

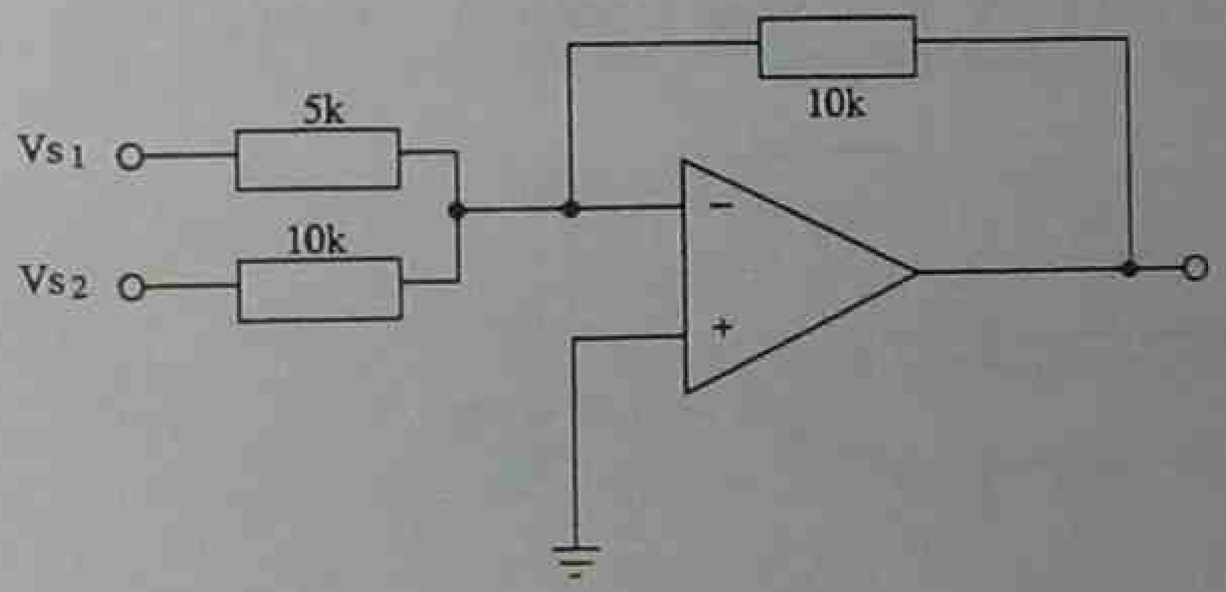
These questions will help you revise what you have learnt in Section 2.

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.

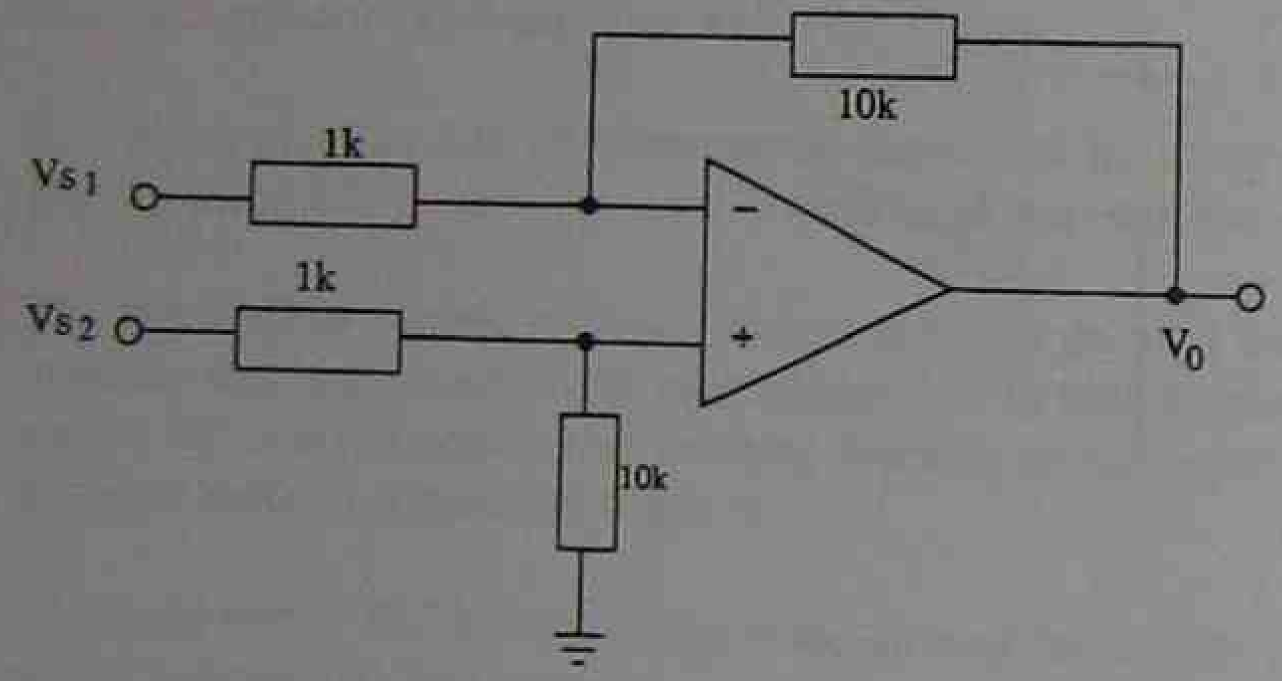


2. For the differential amplifier circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

   (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.

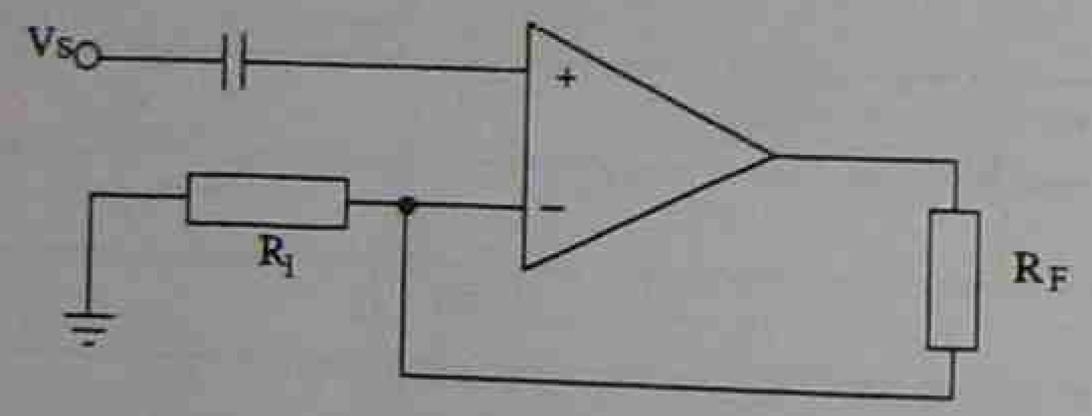
4. What are common ways to avoid errors due to drift in DC offsets in amplifiers?

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.

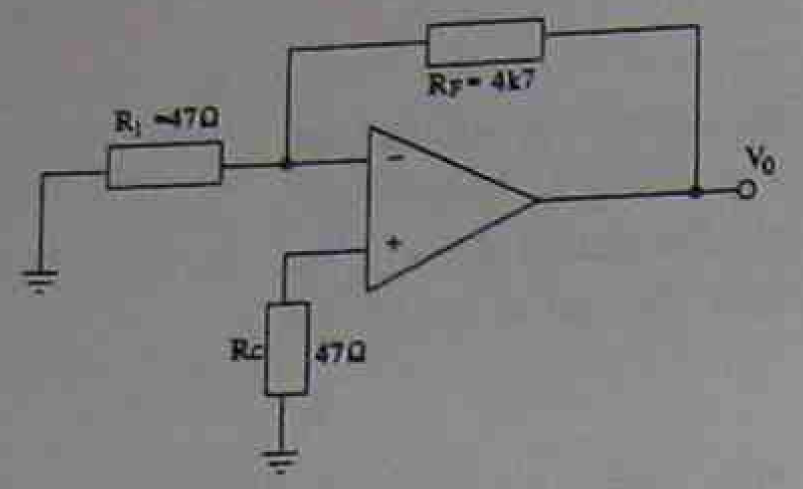
No external signal source is necessary. In the following circuits, the input is connected to ground.

(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}*

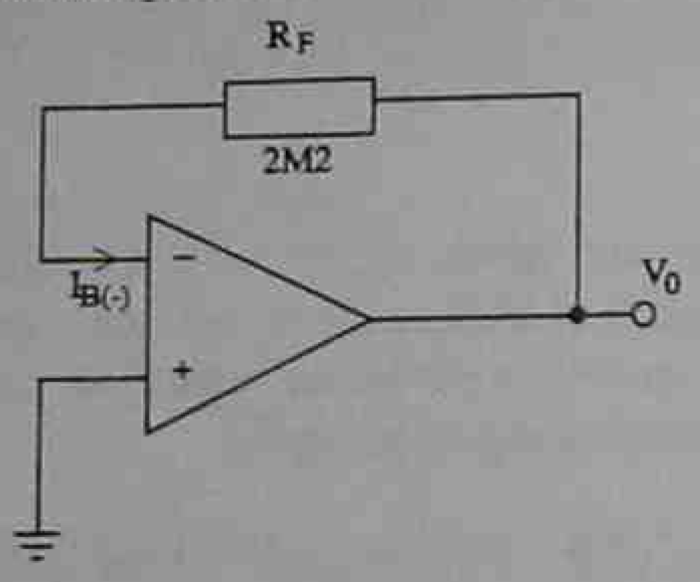
- Connect the following circuit and measure the output DC voltage V_o.



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

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

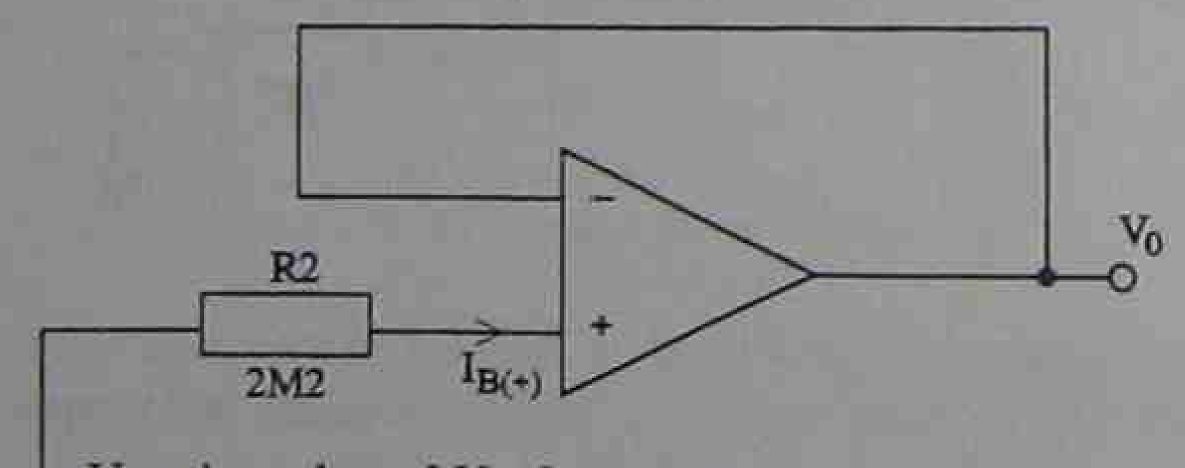
- Connect the following circuit and measure the output DC voltage V_o.



- 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.

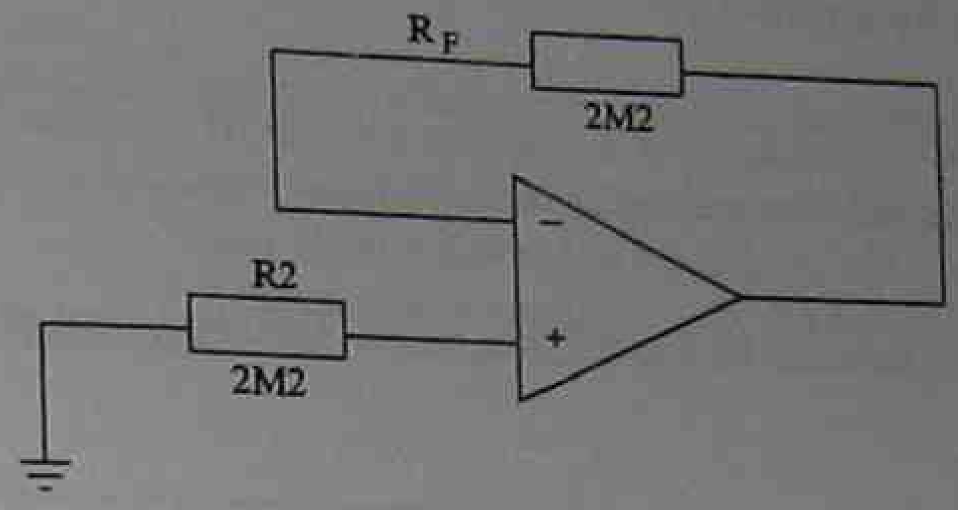


- Use the value of V_{io} from Step 1 and calculate

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

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

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



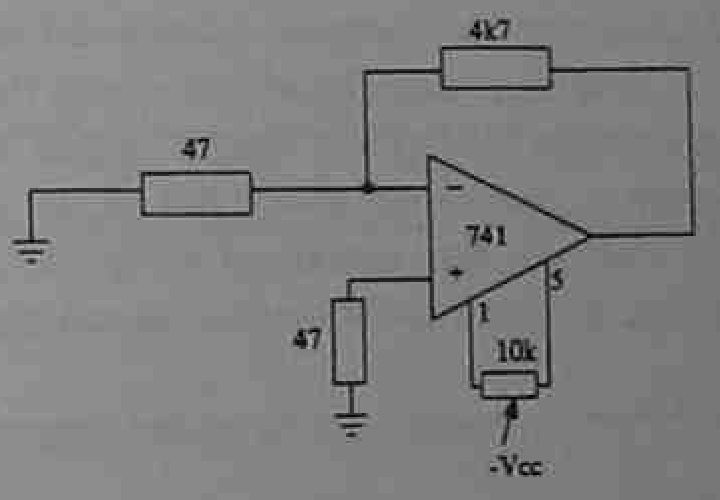
- Use the value of V_{io} from Step 1 and calculate

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

- 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).

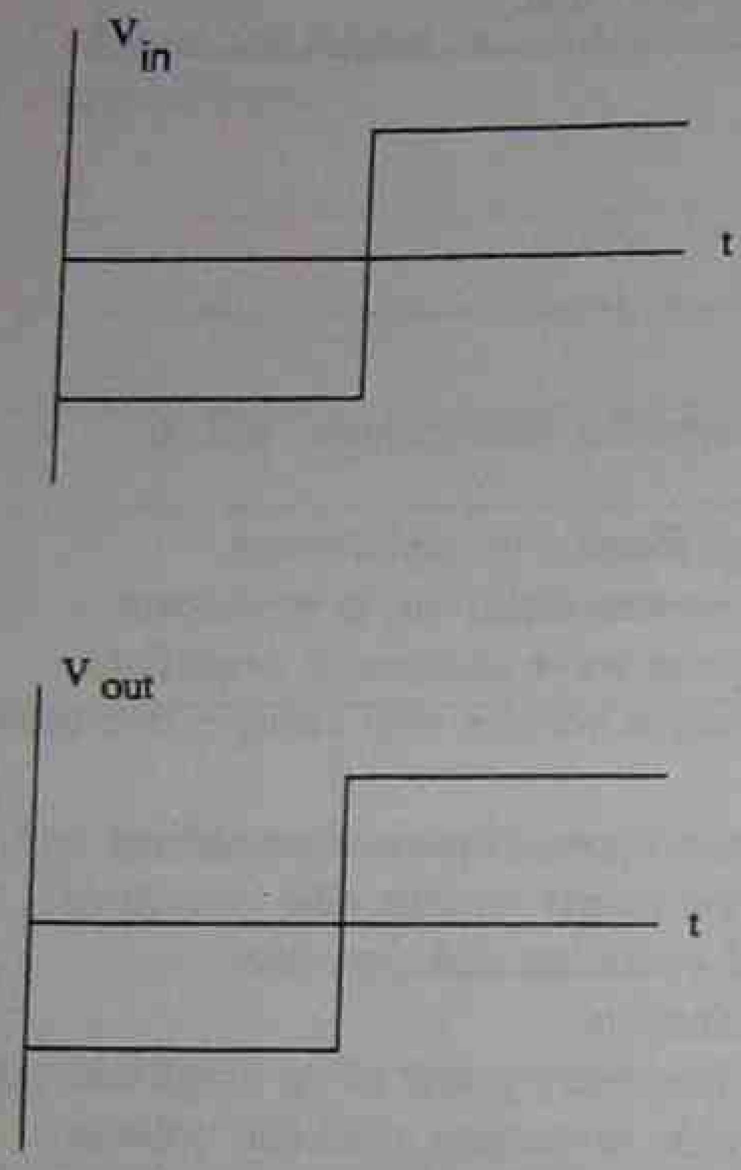


Fig. 1 (Ideal output)

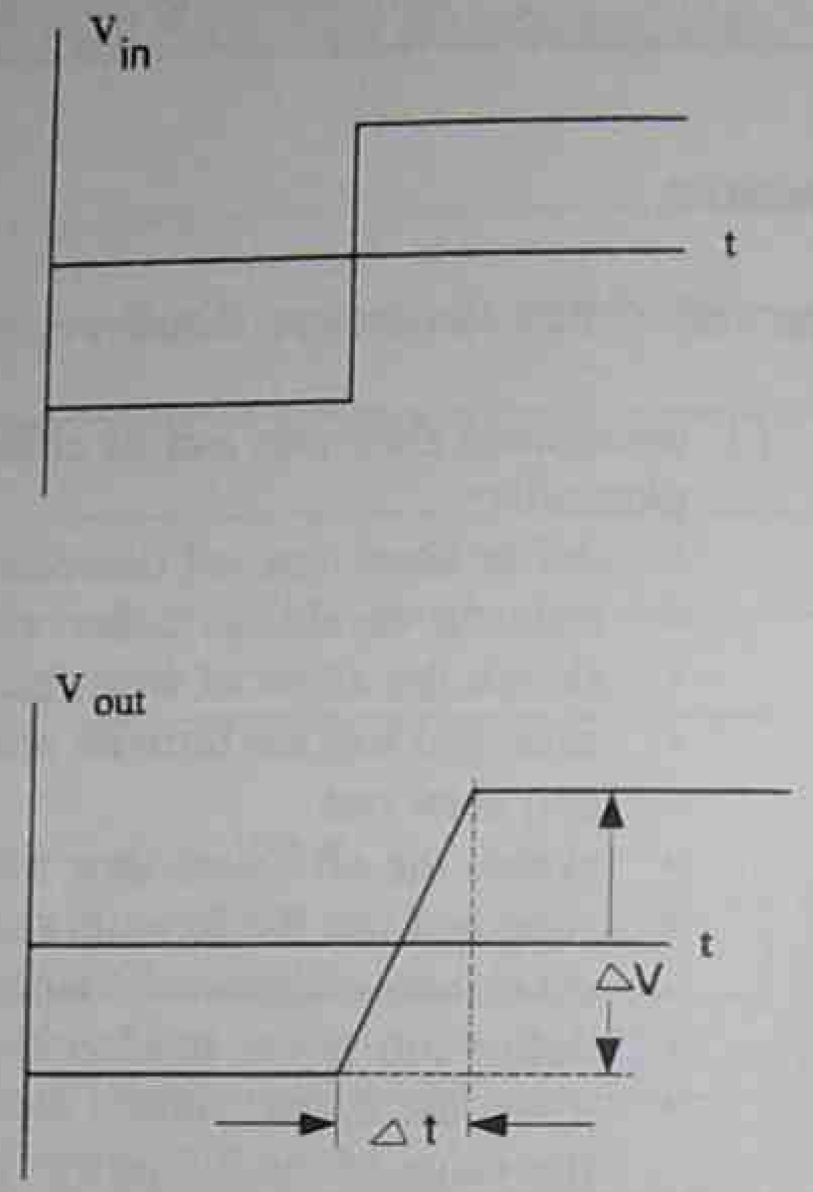


Fig.2 (Slew rate limited)

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) 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).

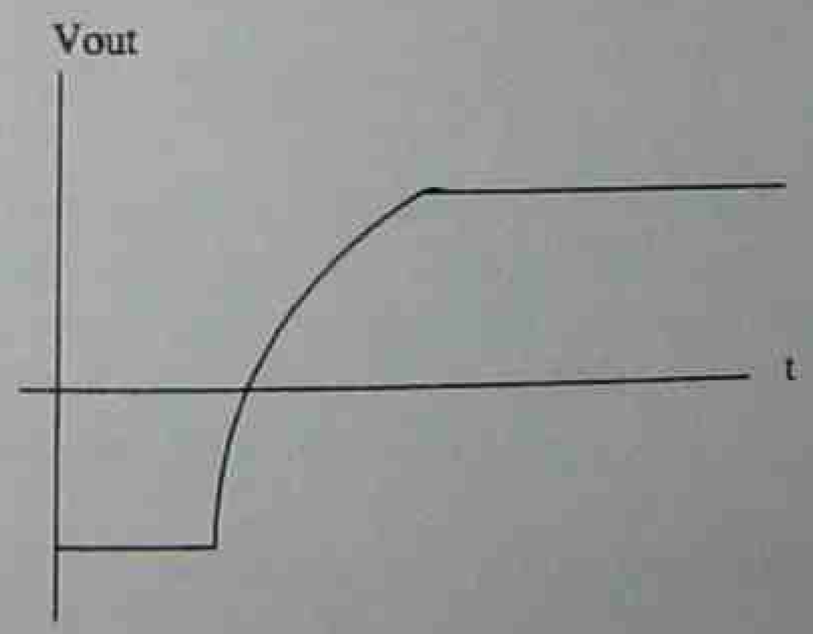


Fig.3

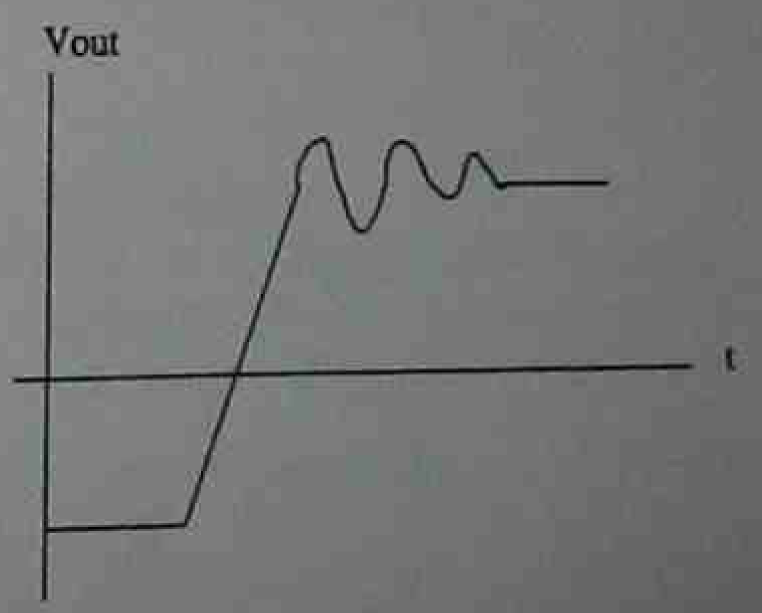


Fig.4

Small signal (bandwidth limited) square wave outputs

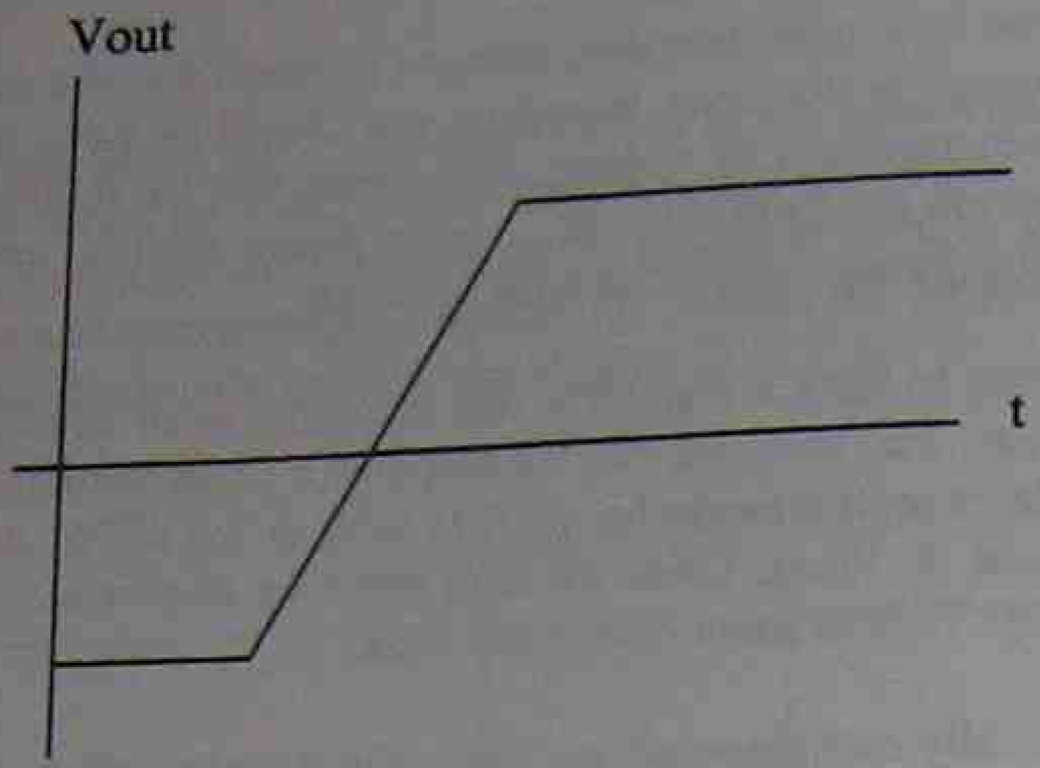


Fig.5. Large signal (slew Rate limited) square wave output

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.

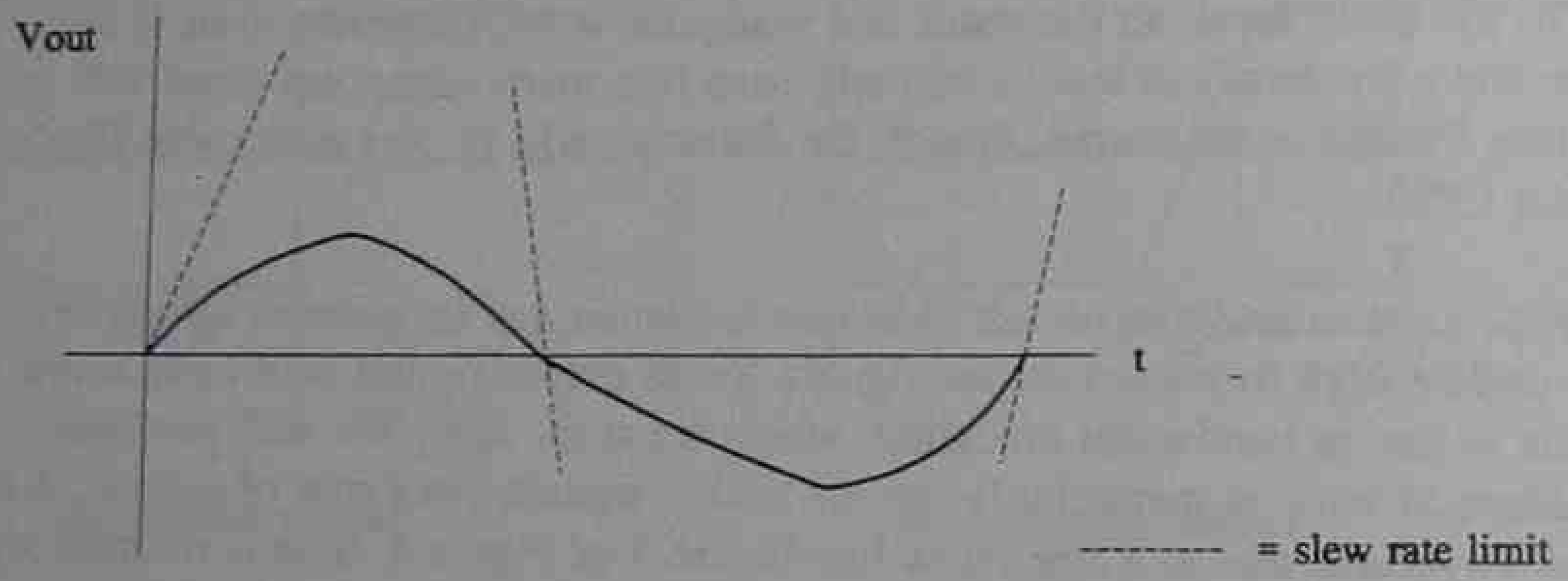


Fig.6 Undistorted sine wave output

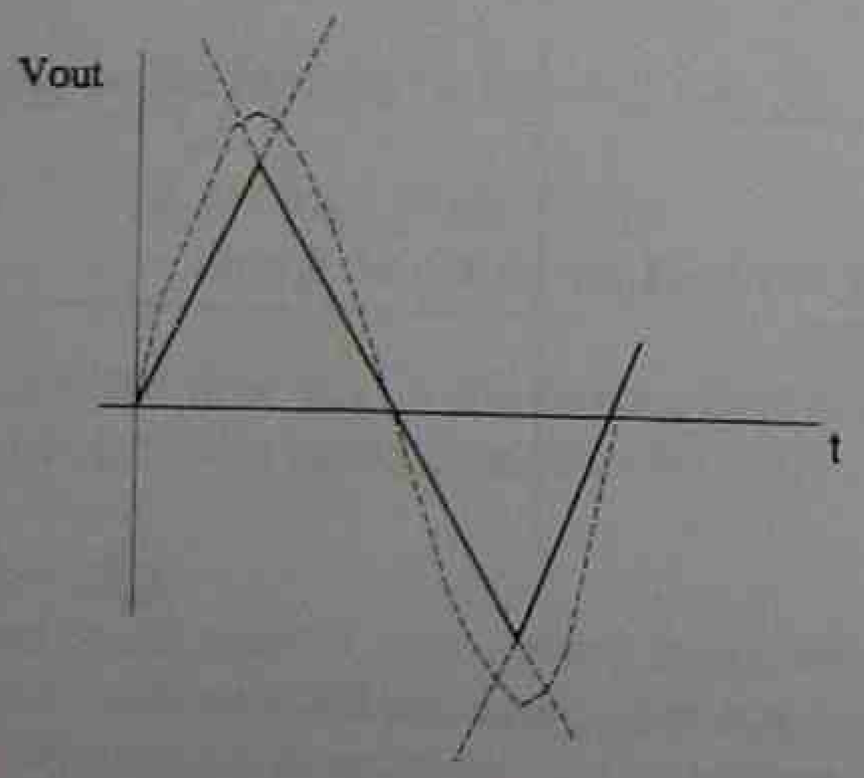


Fig. 7 High frequency sine waves are slew rate distorted

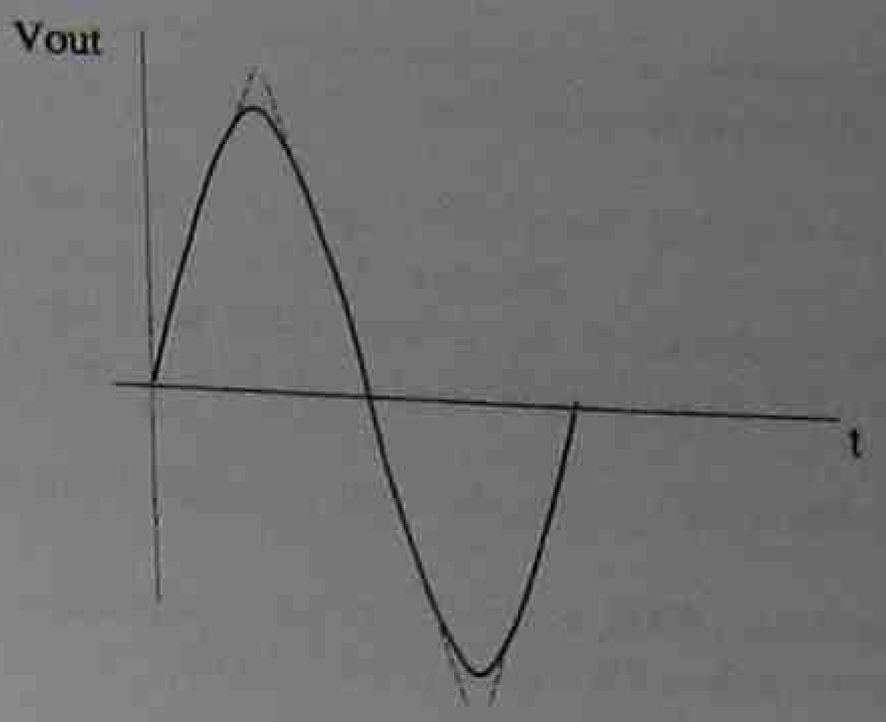


Fig.8. High amplitude sine waves are slew rate distorted

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

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

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

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

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 = $0.8\ V/\mu s\ /(2 * \pi * 14V)$

$= 9.1\ kHz$

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.

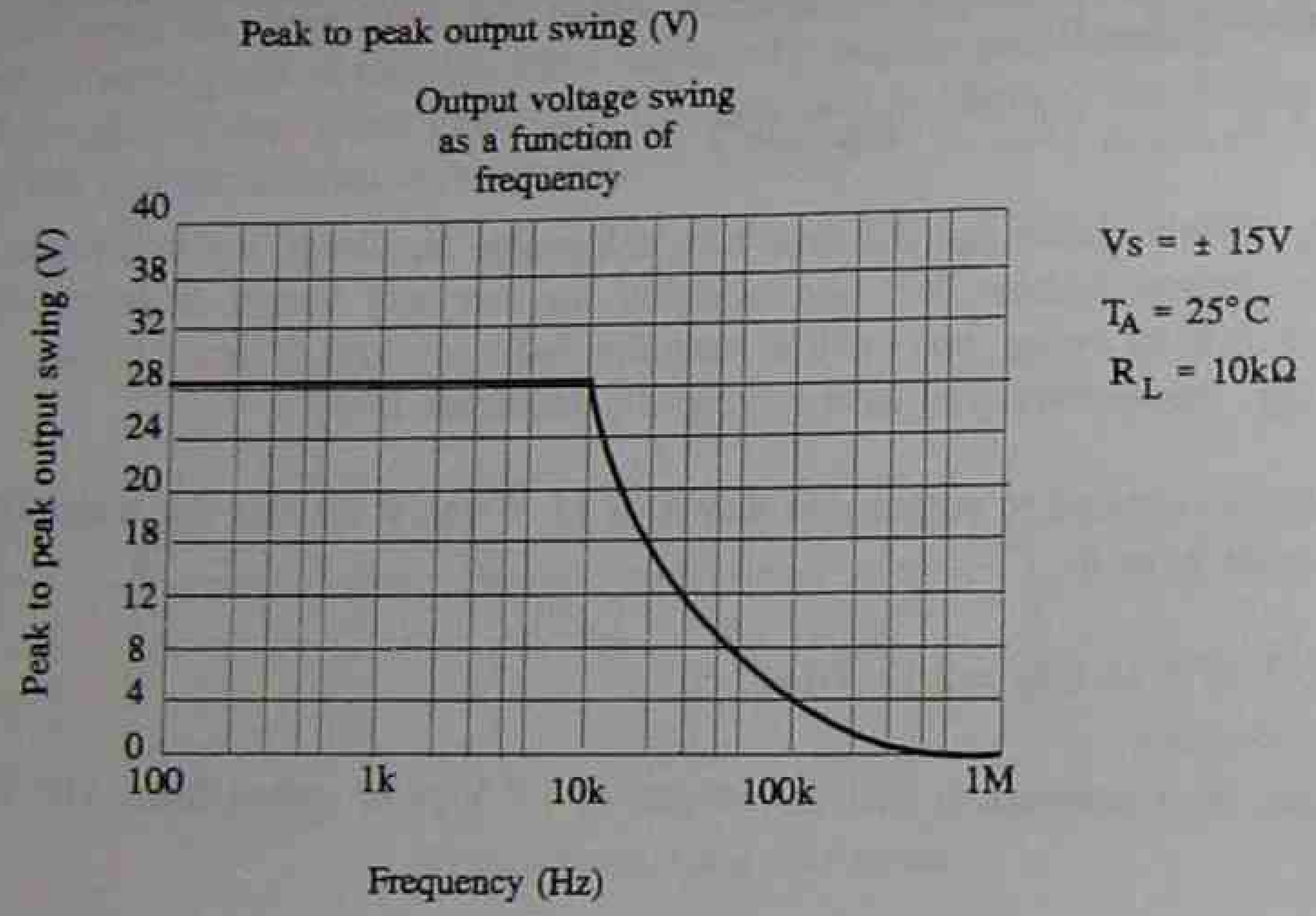


Fig. 9. Output voltage swing graph of the 741 op amp

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

$\therefore$ SR $= 2 * \pi * (28V/2) * 10\ kHz$

$= 0.88\ V/\mu s$

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

$\therefore$ max v_i rms $= 28V\ /\ (2\sqrt{2})\ /\ gain = 1\ V$ 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 $= 14V\ /\ (2\sqrt{2})\ /\ gain = 0.5\ V$ rms

The output is slew rate limited above FPBW

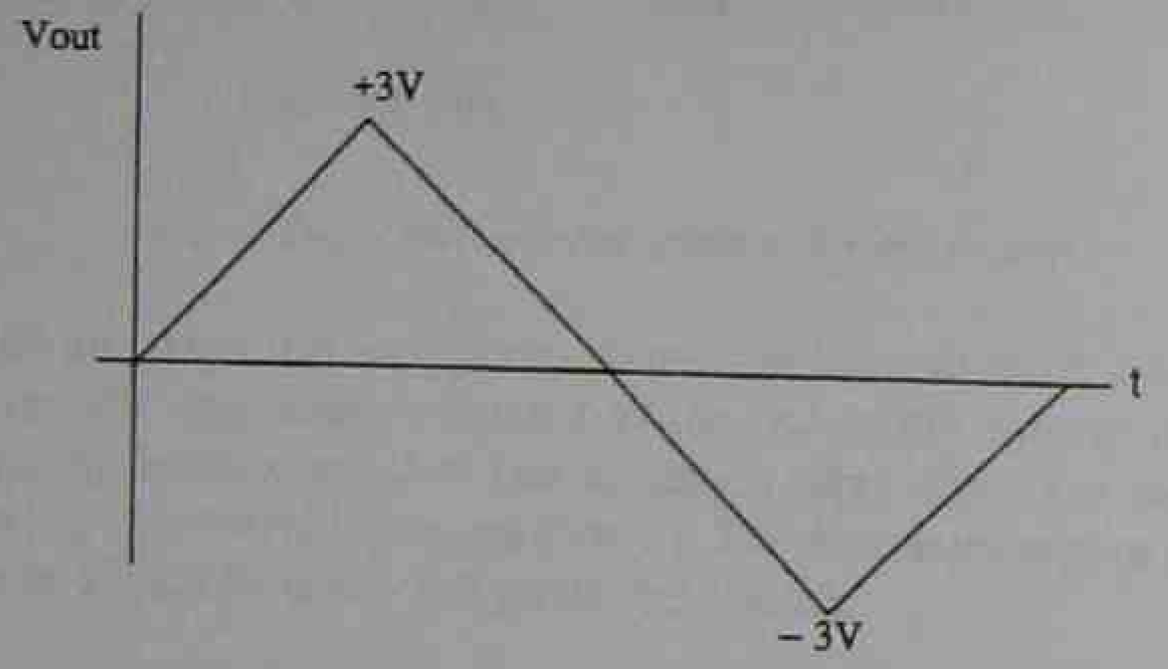
(Note : you could have calculated the answer to part (c) using the slew rate and equation 2. The answer would be about the same, except for the human error in reading graphs.)

Review questions

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

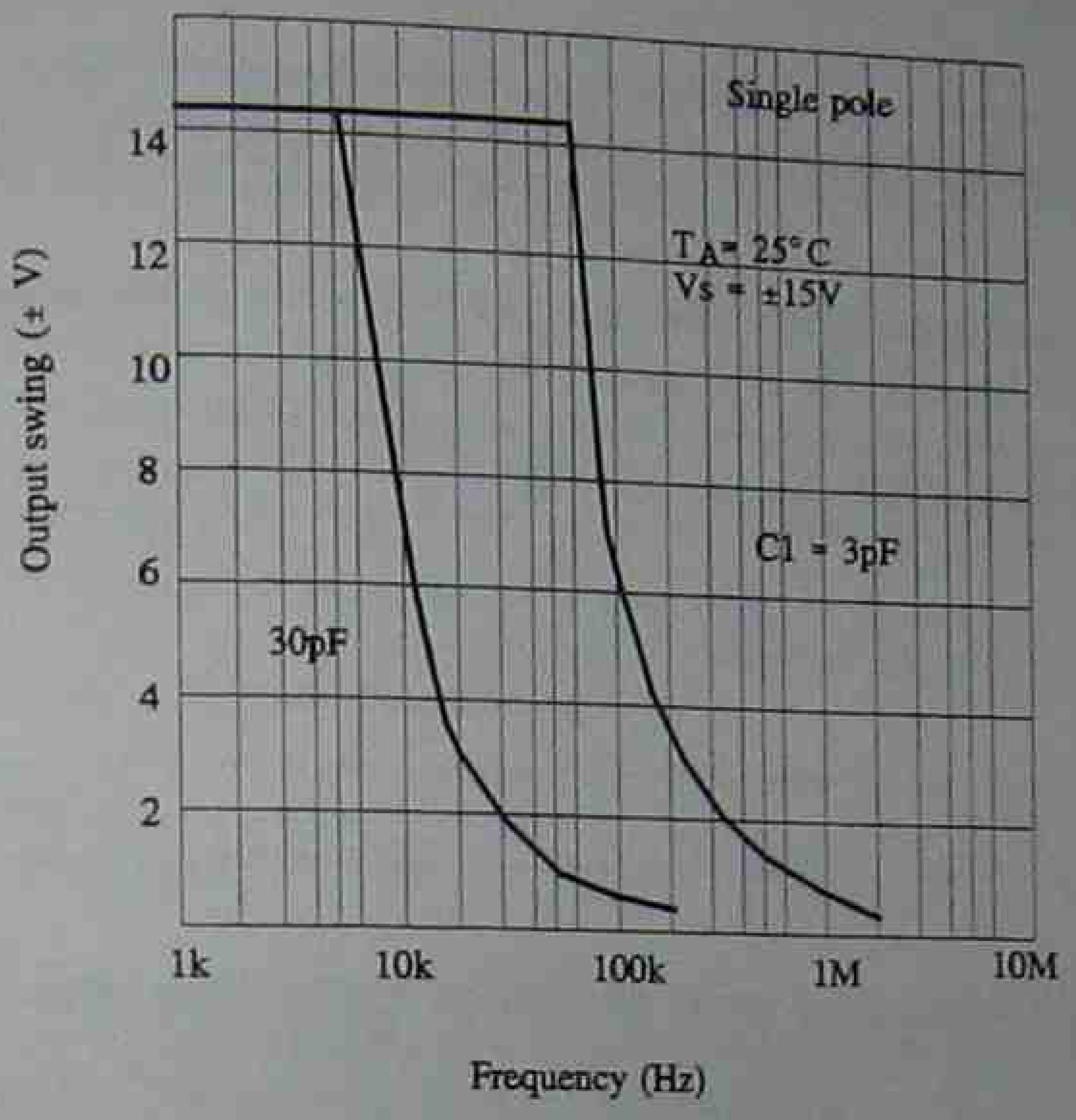
1. An operational amplifier with a slew rate of 0.5 V/μs is connected as a voltage follower and a square wave input at 62.5 kHz, 6V peak to peak is applied. Sketch the output waveform, carefully marking important time and voltage values.

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

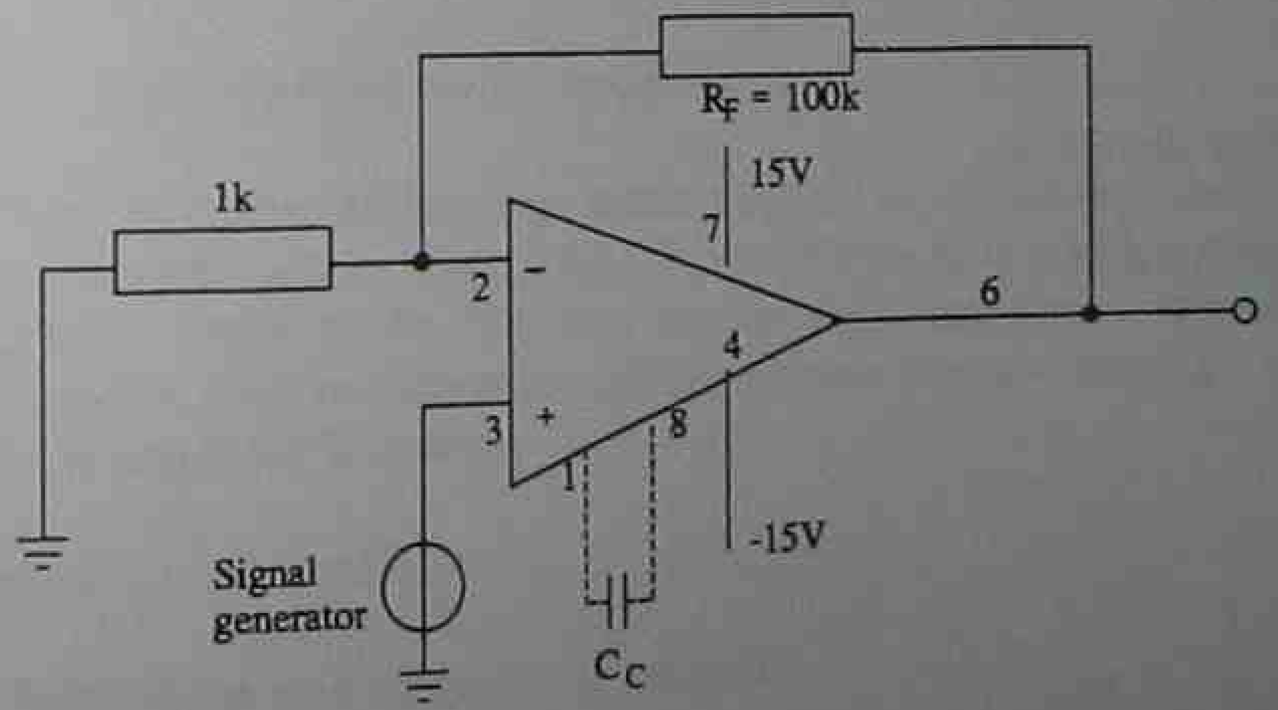
- Either one op amp type LM301 and a 30 pF capacitor OR one op amp type 741. (741 is not suitable for Step 6.)
- General purpose 15 MHz oscilloscope (one with time-base x5 or x10 switch preferable)
- AC signal source (e.g.function generator)
- Dual DC power supply (fixed ±15V supplies can be used)
-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

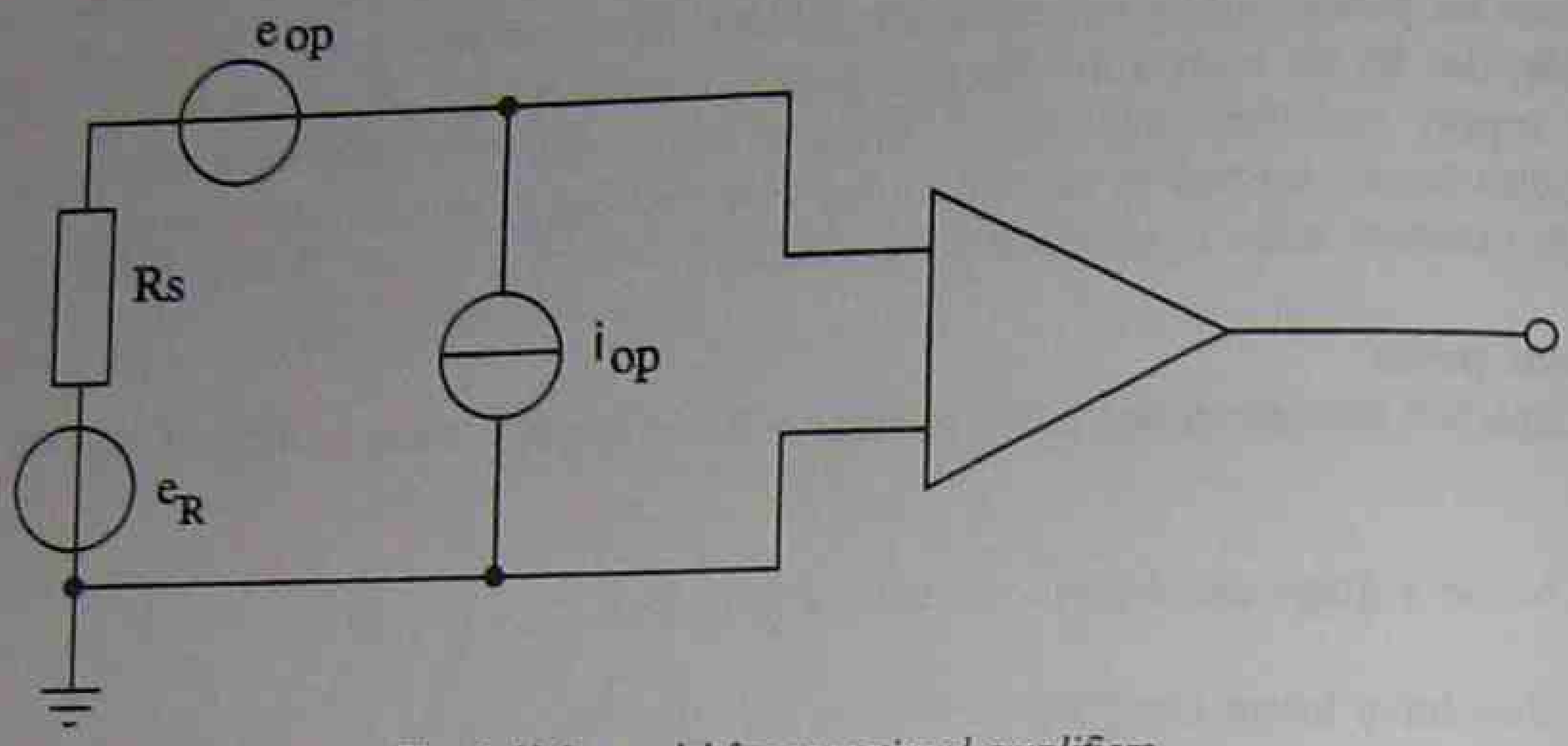


Fig. 1 Noise model for operational amplifiers

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 $\mathrm{nV}/\sqrt{Hz}$, its rms noise voltage over a bandwidth of 30 kHz

$= 15\ \mathrm{nV}/\sqrt{Hz} * \sqrt{30kHz} = 2.6\ \mu\mathrm{V}$ rms.

Example 2

The noise current density of a 741 op amp is specified as $3 * 10^{-25}\ \mathrm{A}^2/\mathrm{Hz}$. 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 =

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

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

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

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.

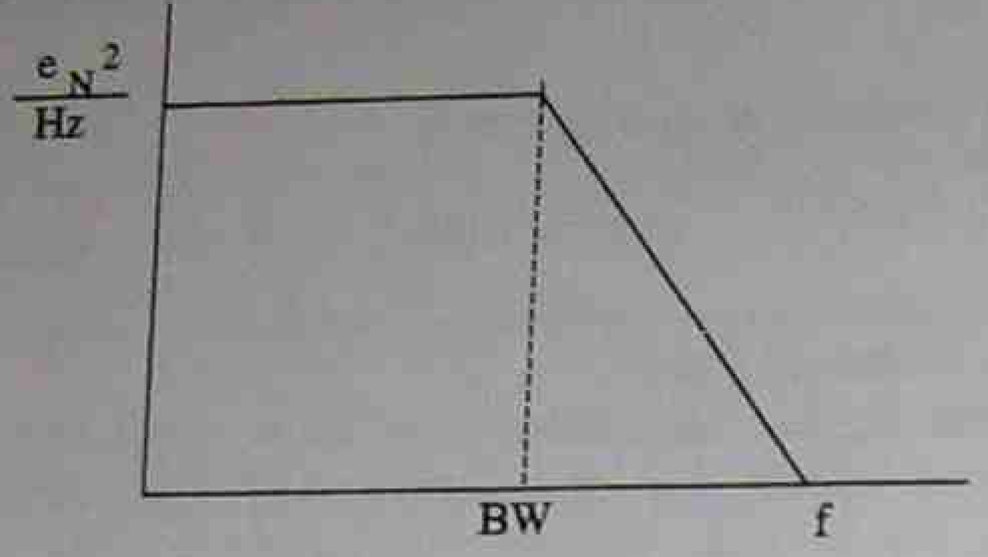


Fig. 2 Practical Bandlimiting Filter response

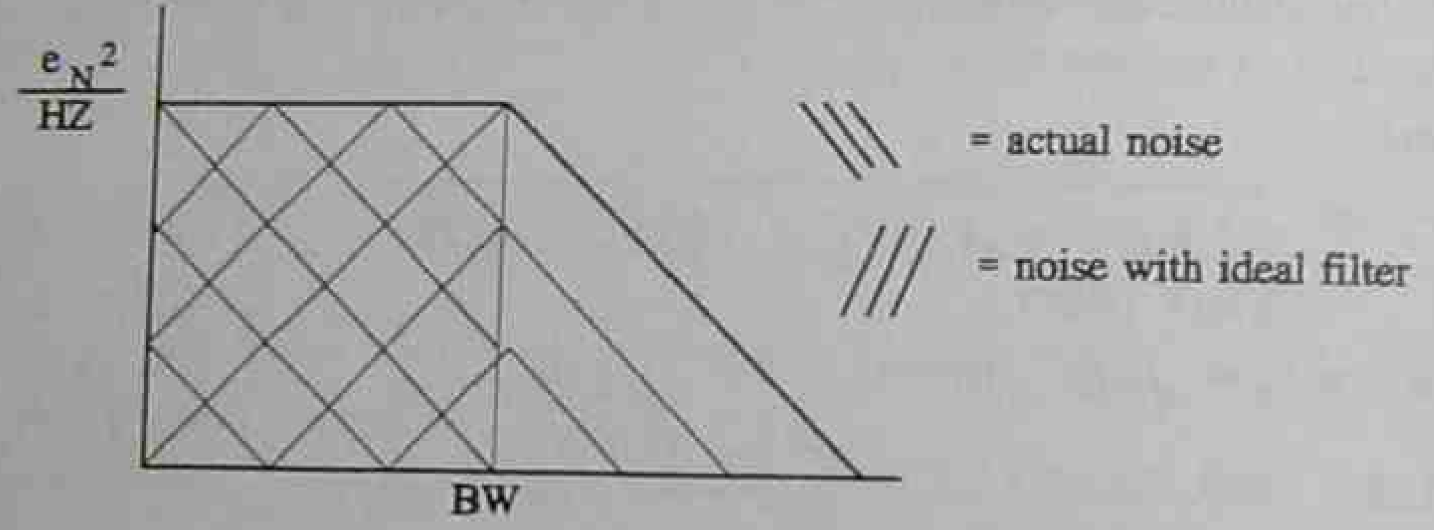


Fig. 3 Comparison of noise allowed in by ideal and practical bandlimiting filters

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).

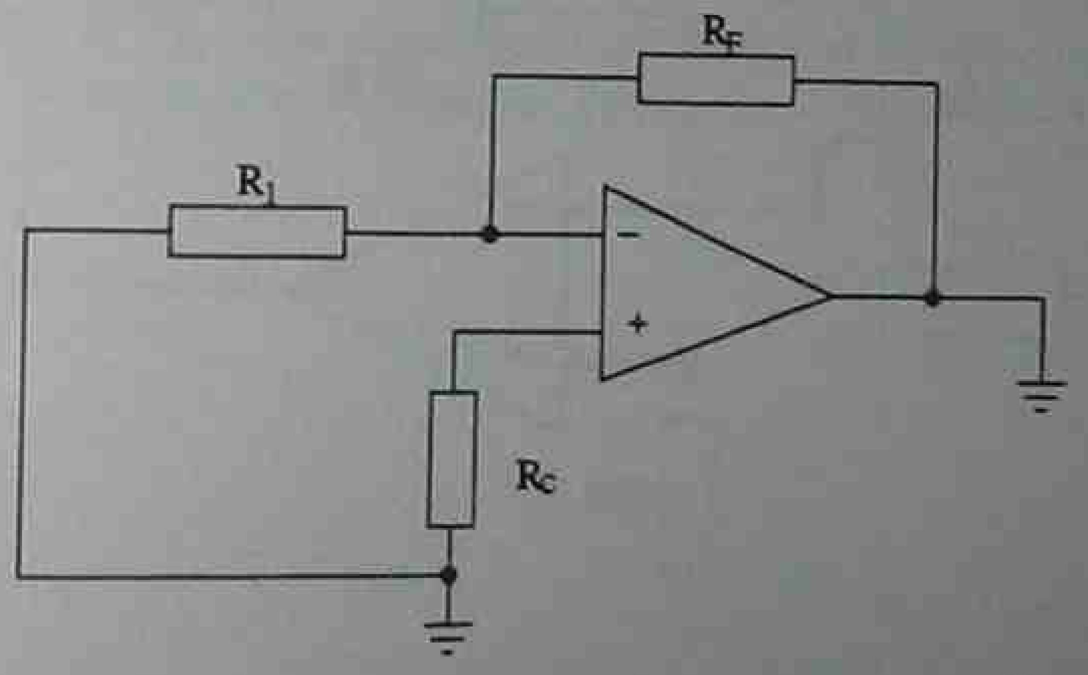


Fig. 4 Negative feedback amplifier

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

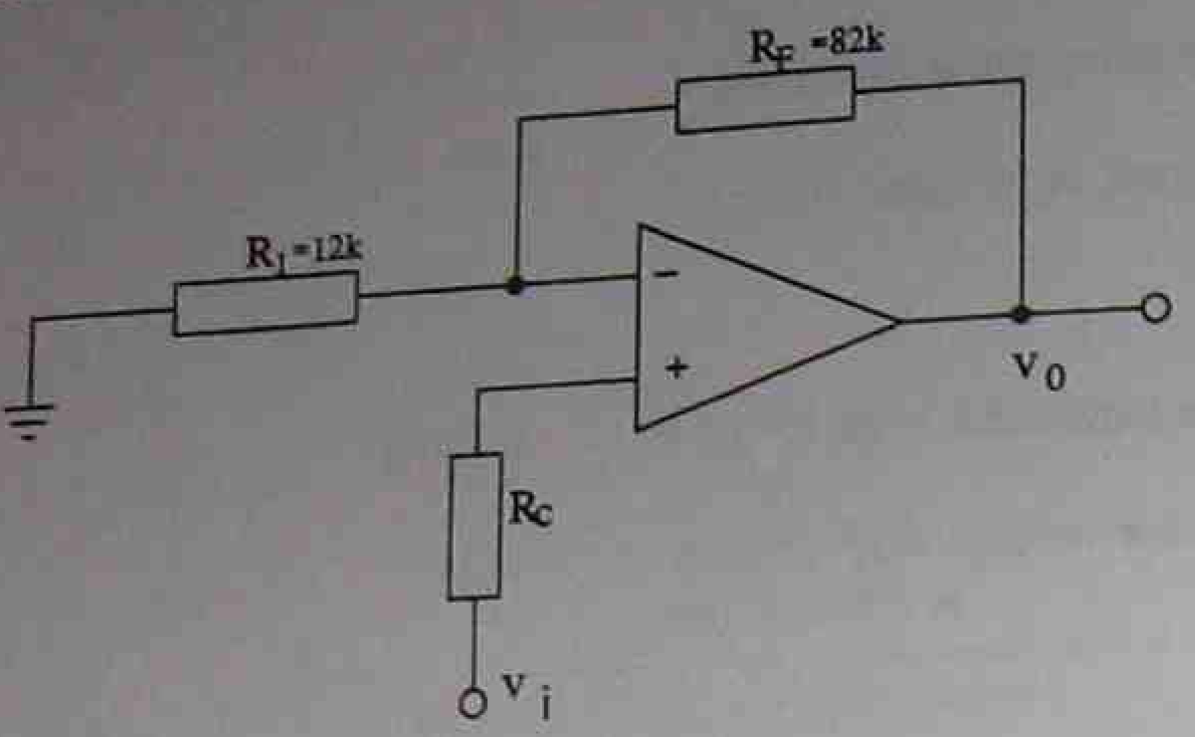
As before, R_c and R_1 are the effective resistances from the +input and -input to ground respectively, after simplifying any series and parallel combinations.(To be exact, the answer in equation 8 must be paralleled with the input resistance of the op amp. Practically, the input resistance is very large 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



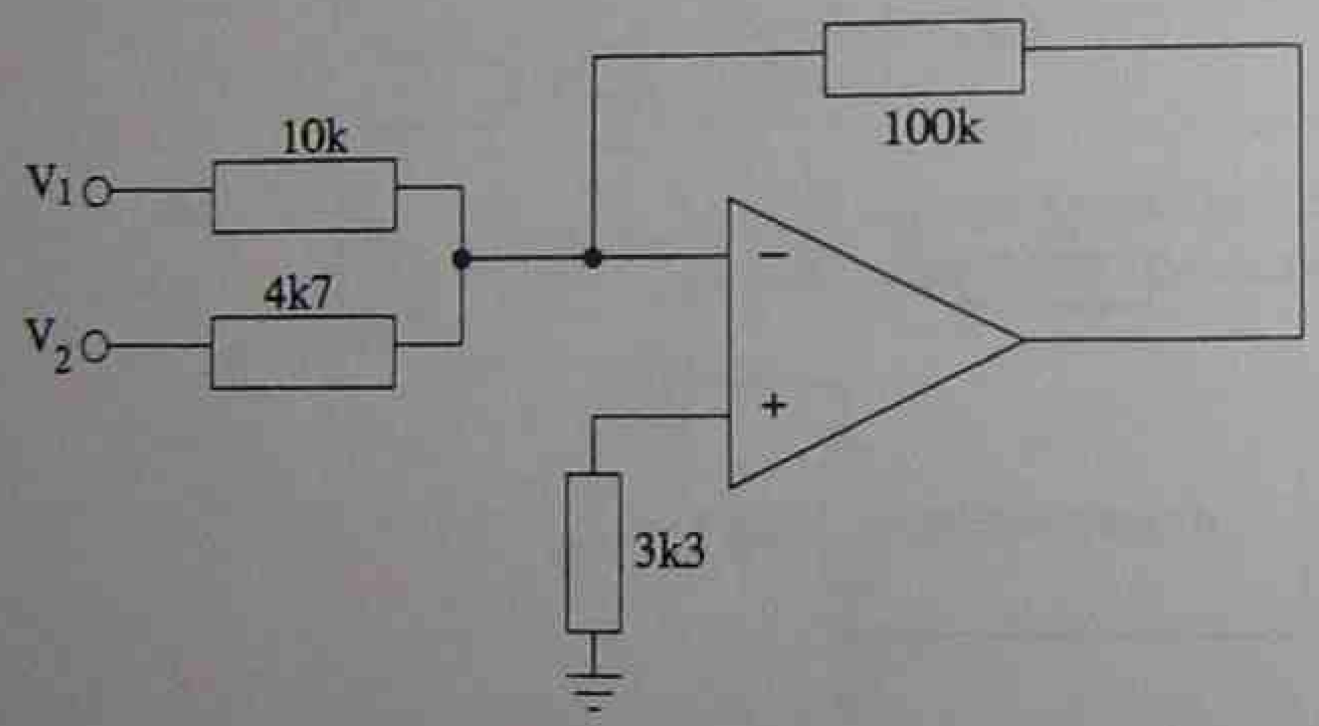
Solutions

(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.

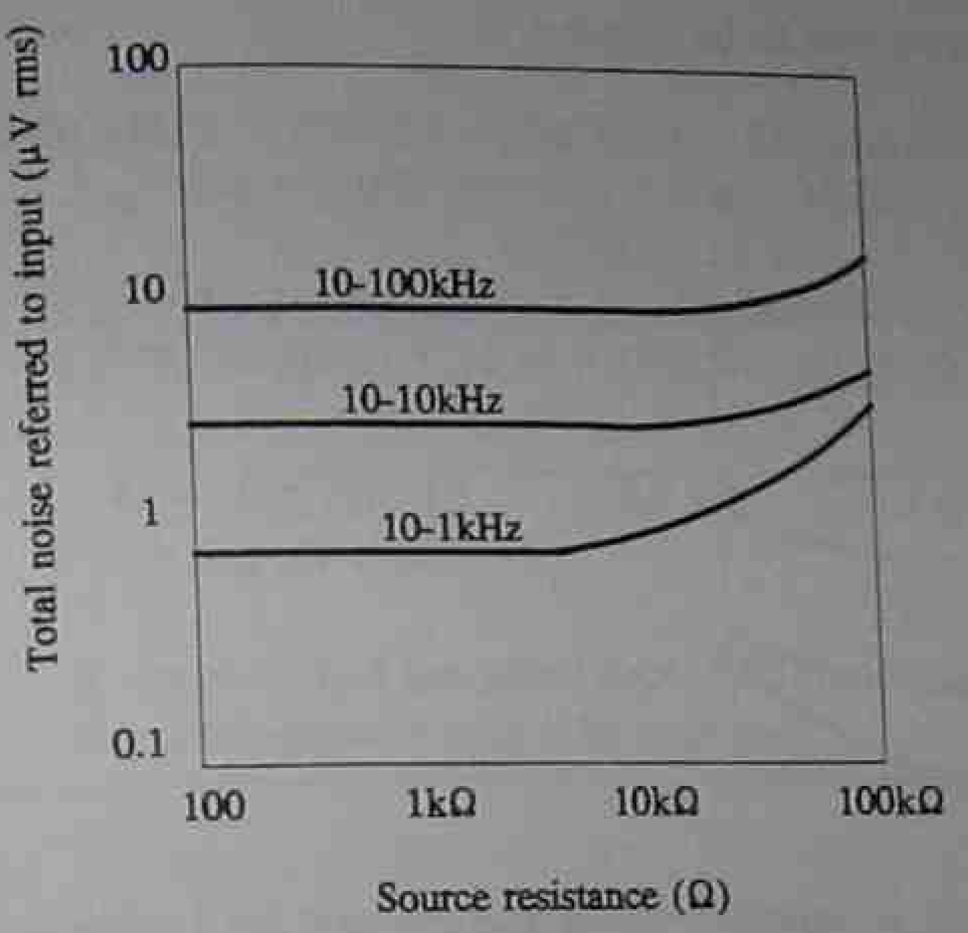


Fig. 5 : Broadband noise graph for 741 op amp

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.

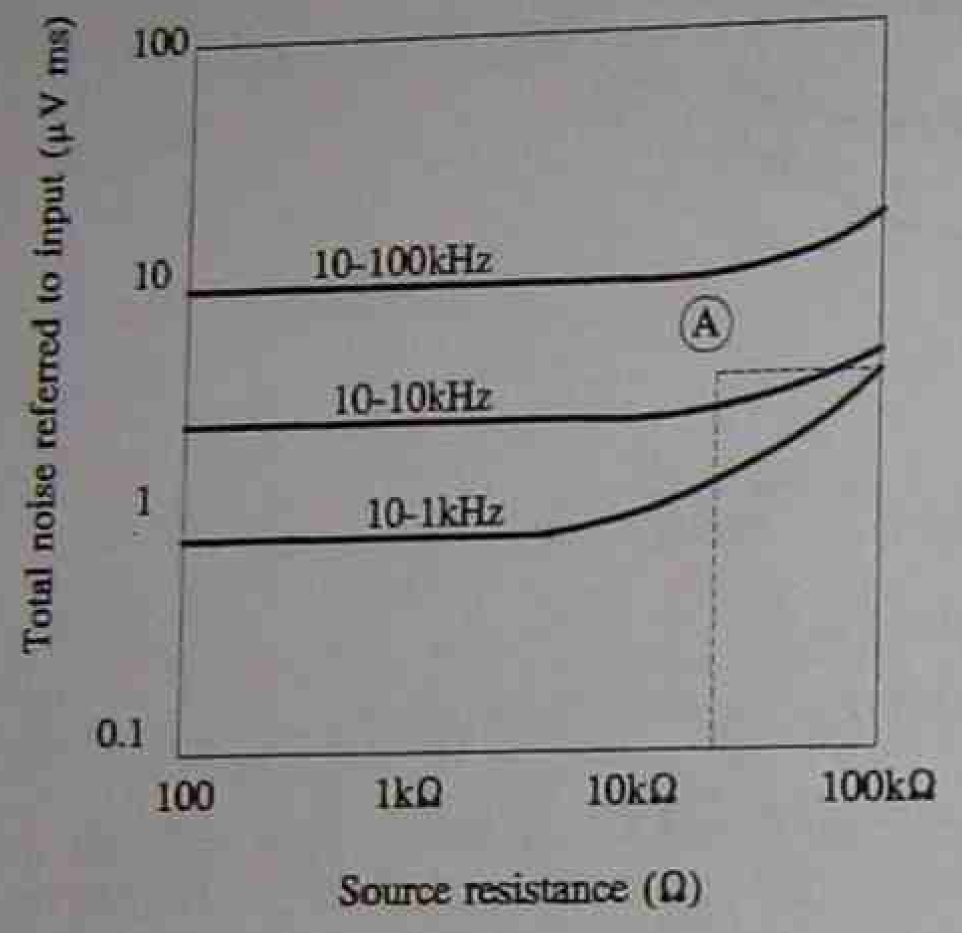
Example 8: Using the broadband noise graph

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

$$R_s = 20\ k\Omega;\ T = 273+25 = 298°K;\ B = 31.4kHz$$

$$\therefore e_R^{\,2} = 4 * 1.38E\text{-}23 * 298 * 20k * 31.4k$$

$$= 10.33E\text{-}12\ V^2$$

$$e_{op}^{\,2} = 4E\text{-}16 * 31.4k \qquad = 12.56E\text{-}12\ V^2$$

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

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

$$= 5.2\ \mu V \text{ rms.}$$

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.

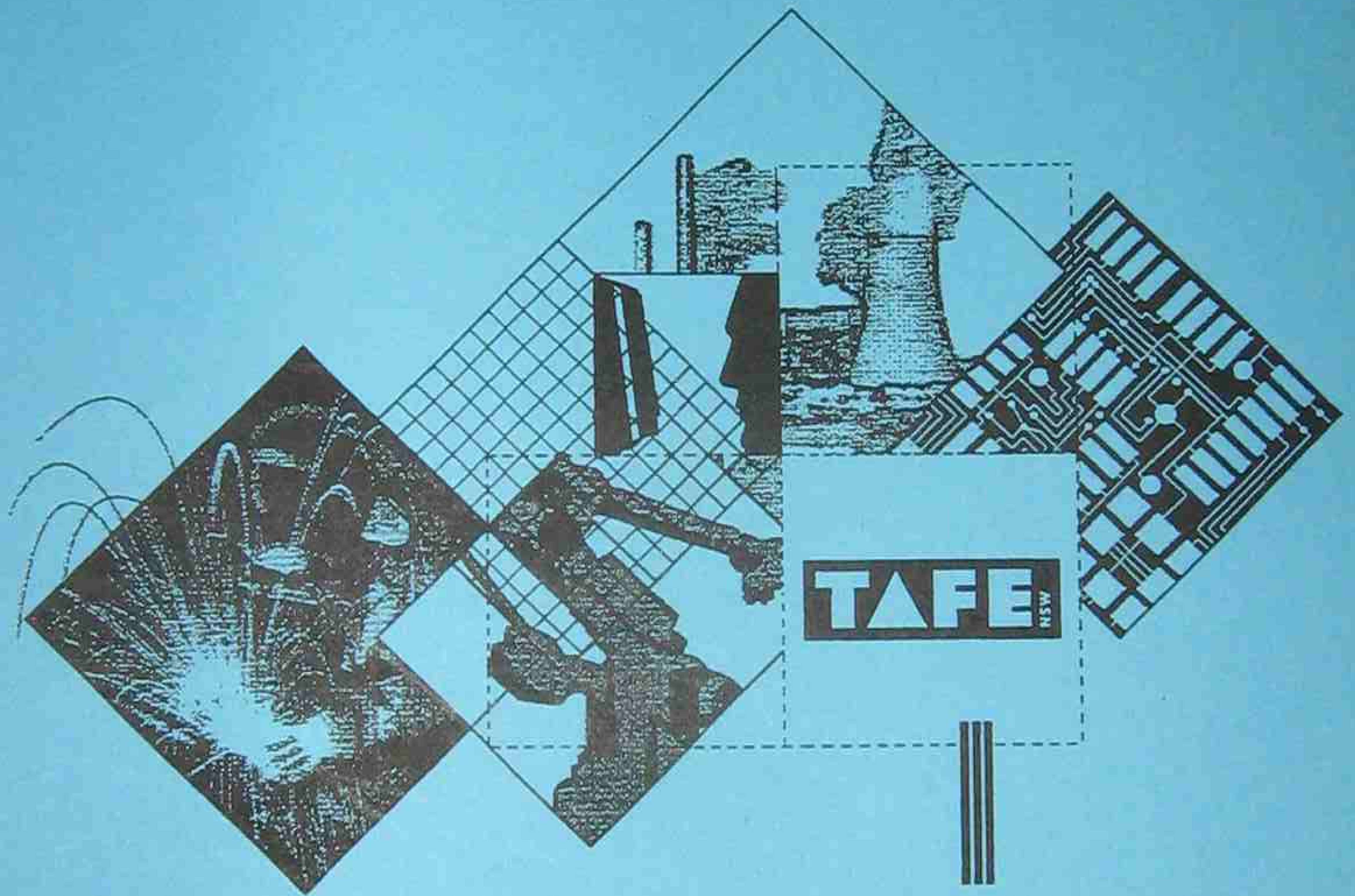
Using a CRO, display the signal with a slow sweep (you can use AC or DC signal coupling), observe the p-p noise voltage and then estimate :

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

Introduction

Module sections

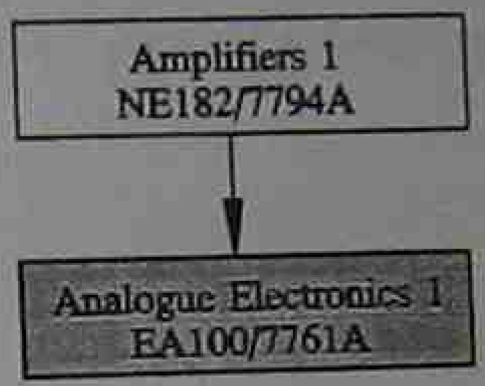
Notes

INTRODUCTION

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.

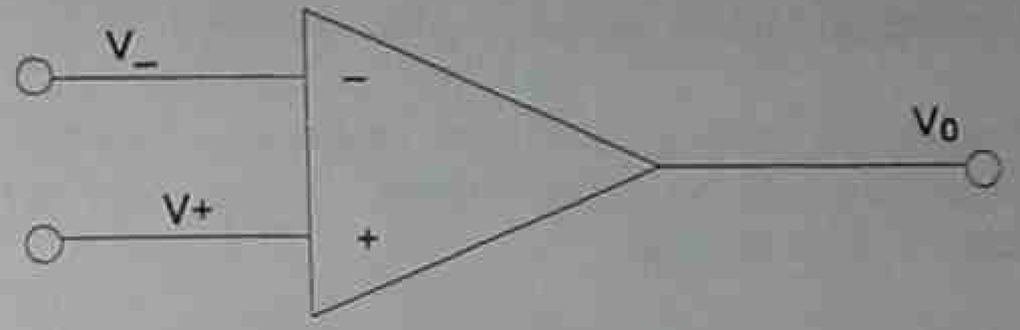


Fig. 1 Operational amplifier

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.
  $\therefore V_+ = V_-$ 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

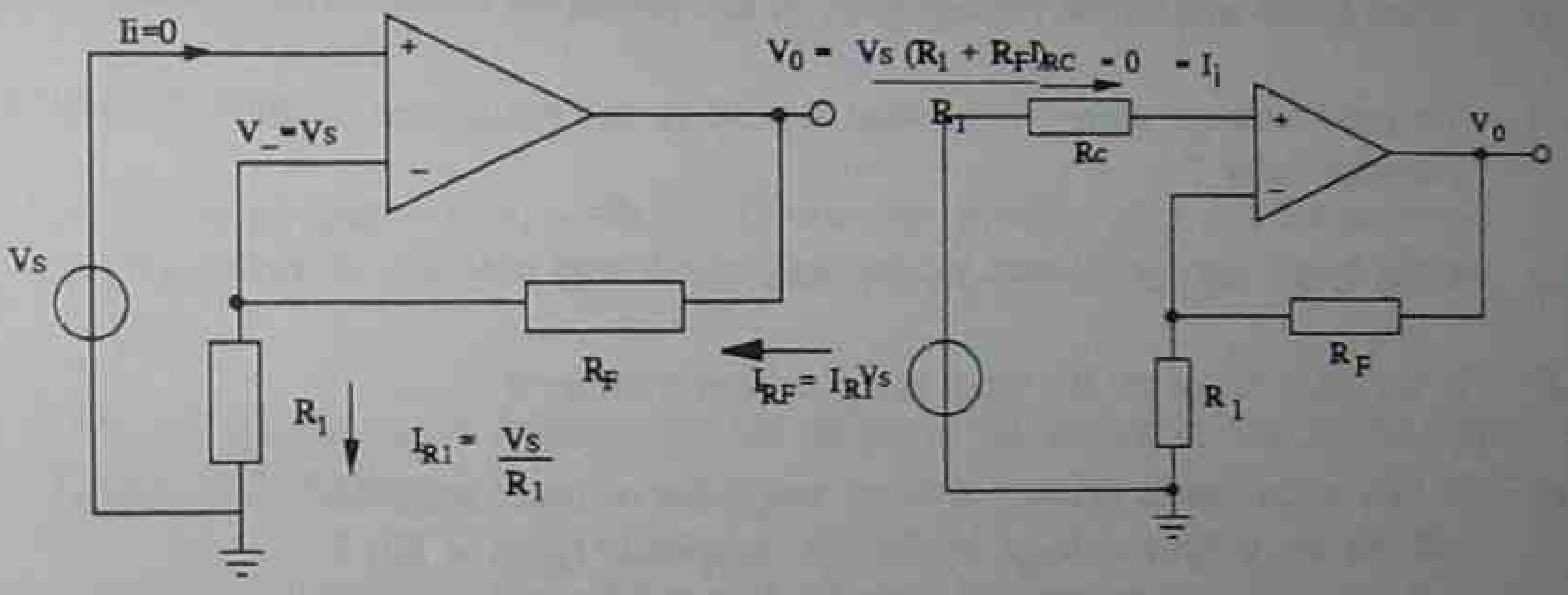


Fig. 2 *Fig. 3*

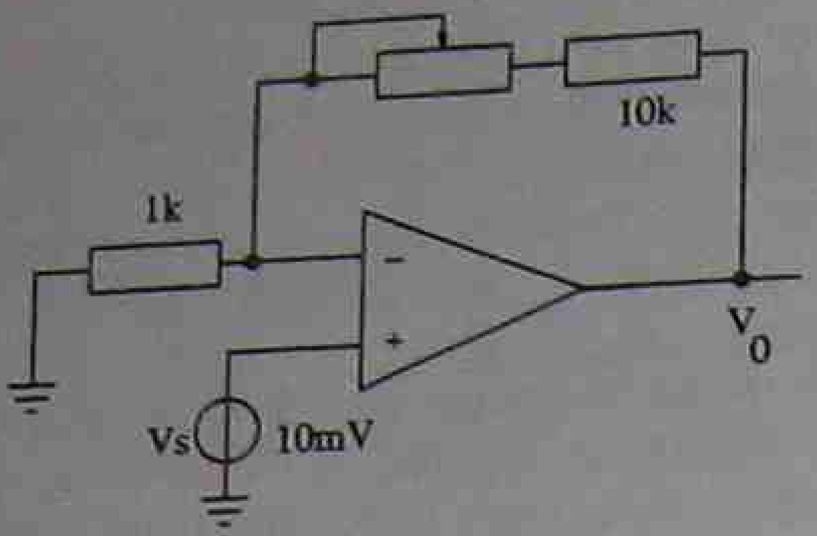
Non-inverting amplifier configurations

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) $v_o = 30 * 10\text{ mV} = 300\text{ mV}$

(e) unchanged

(f) unchanged

Inverting amplifier

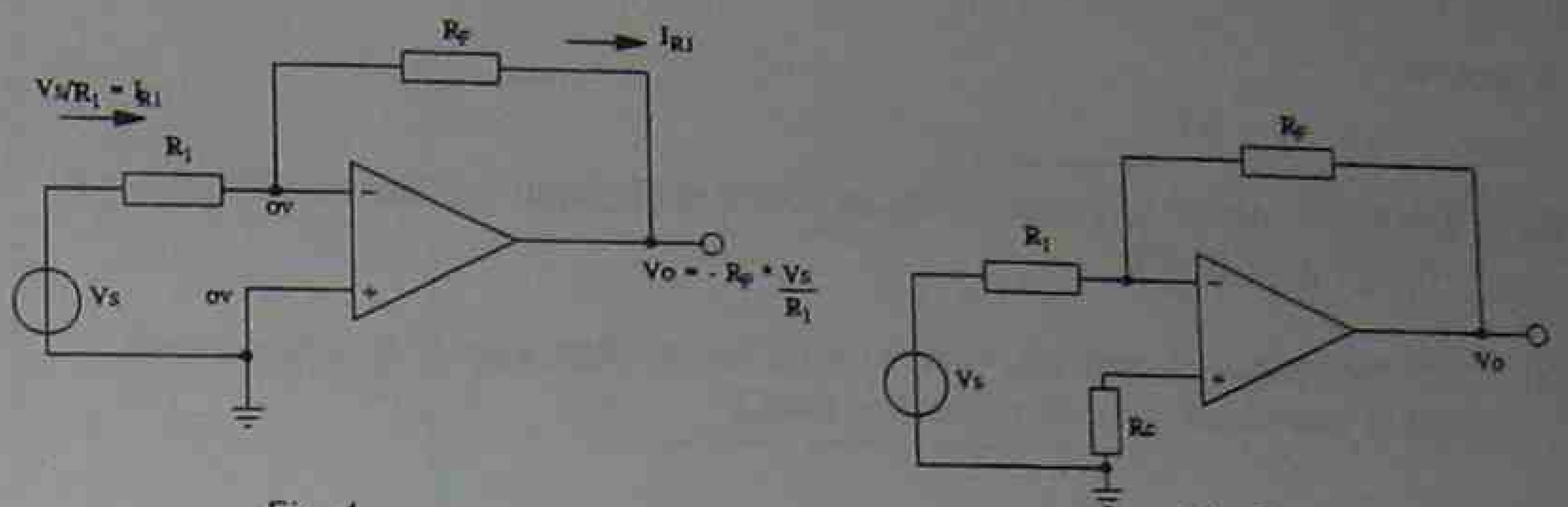


Fig. 4

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

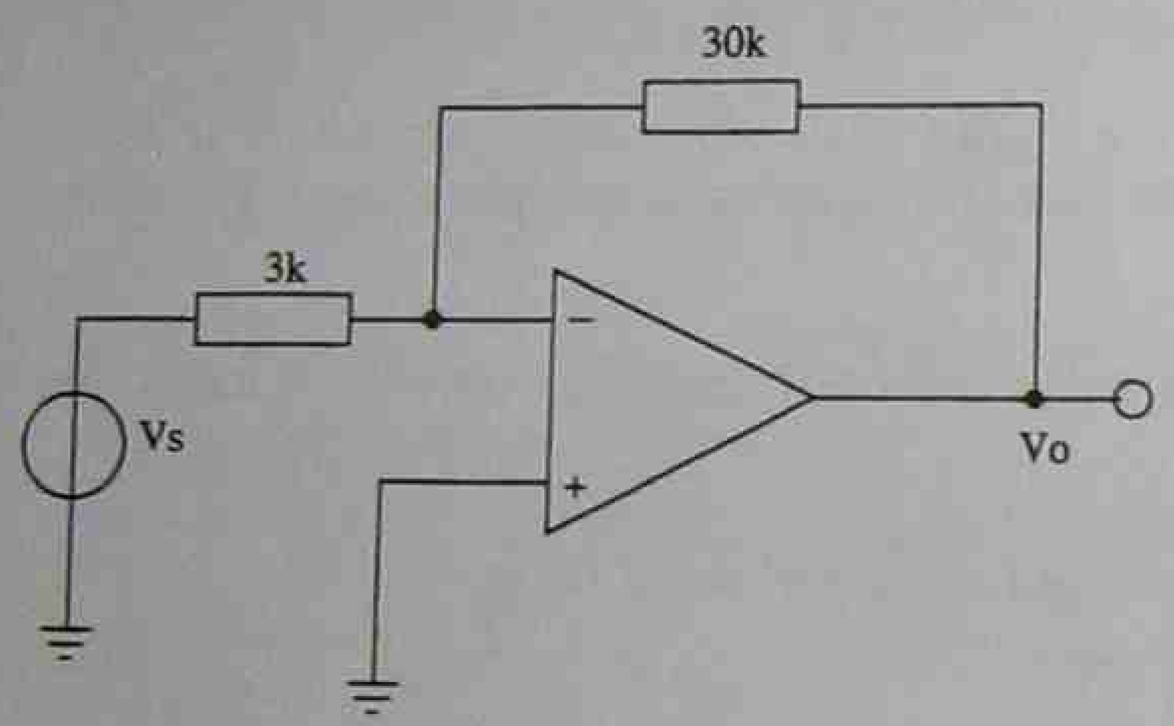
Questions

(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

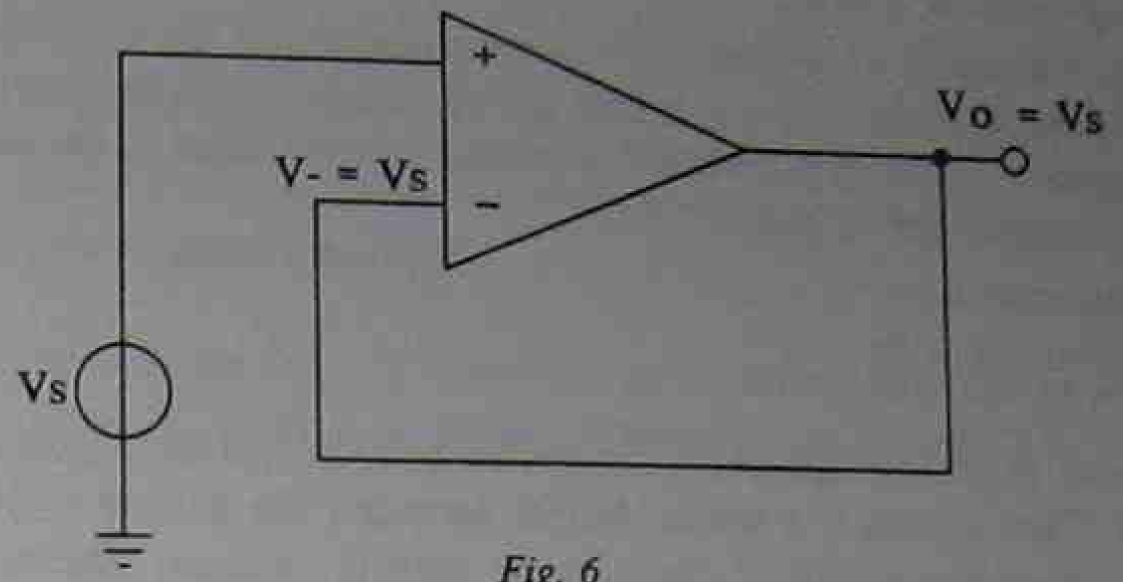


Fig. 6

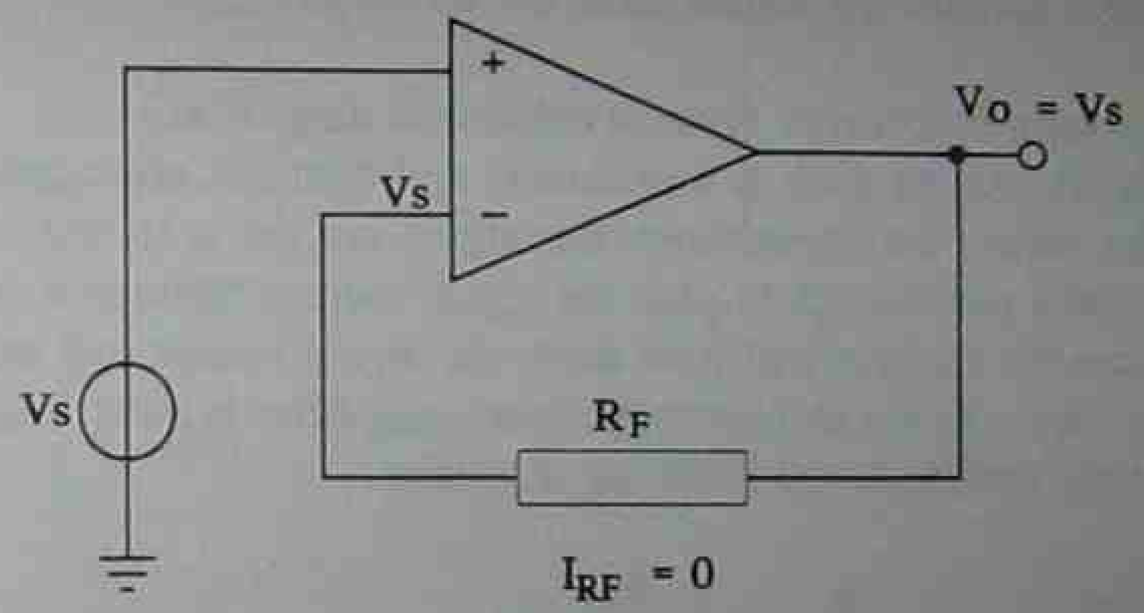


Fig. 7

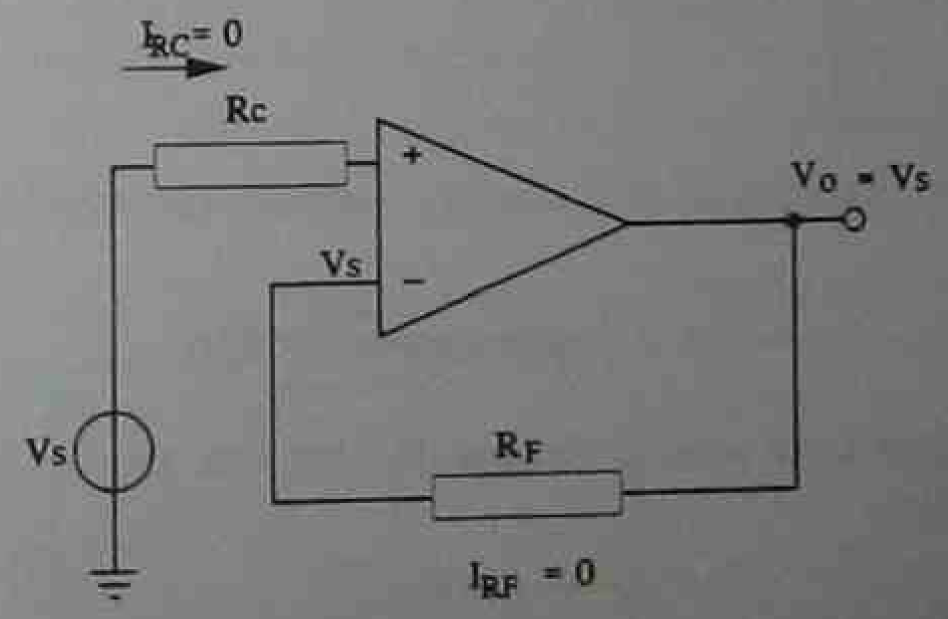


Fig. 8

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.

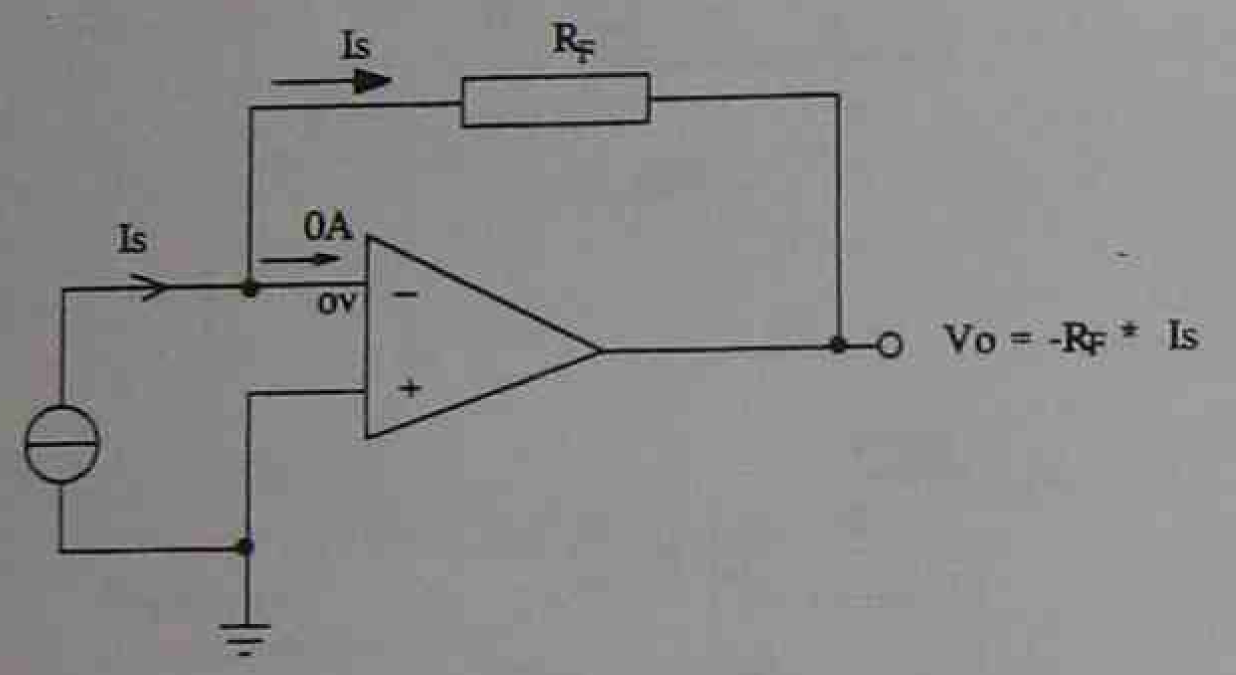


Fig. 9 Transresistance amplifier

The input current must flow around through R_F, because the op amp input does not draw any current. Also,
$V_- = V_+ = 0V$

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

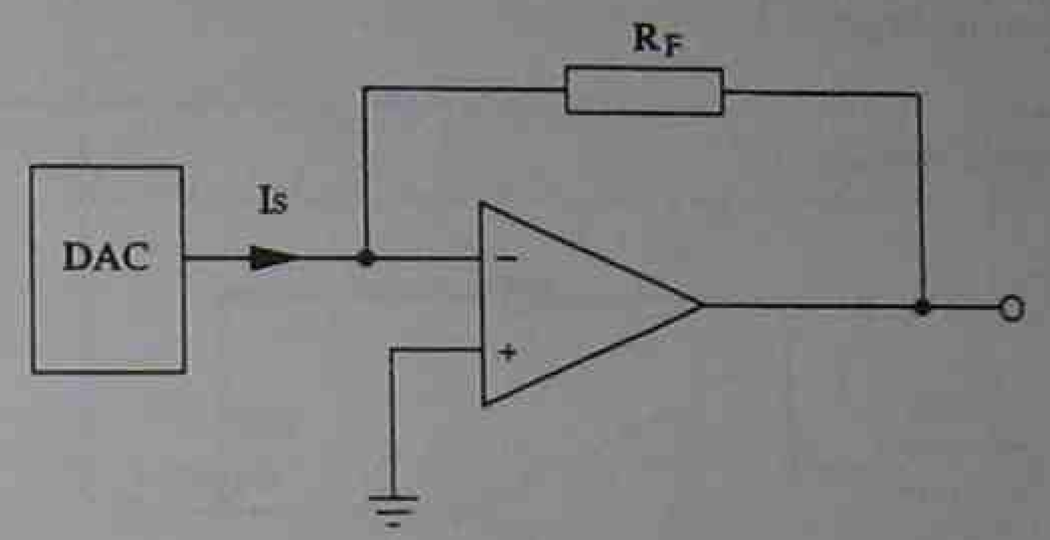
This means that $V_o \propto I_s$. The ratio V_o/I_s is called the transresistance R_T of the amplifier.

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)

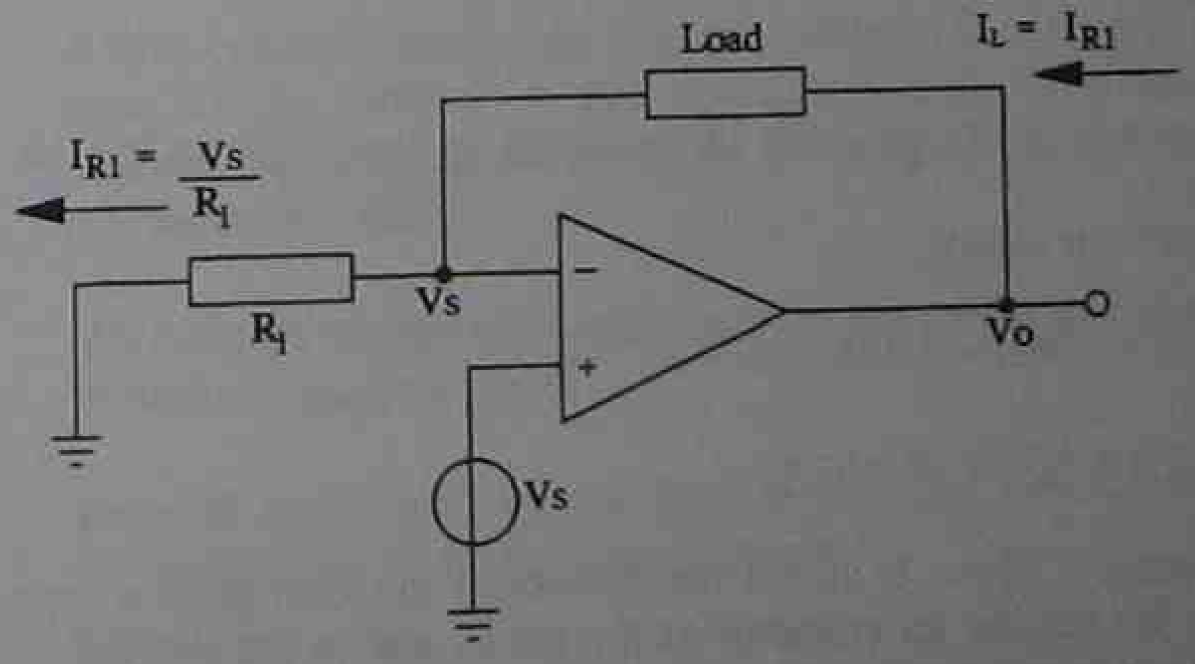


Fig. 10 Transconductance amplifier

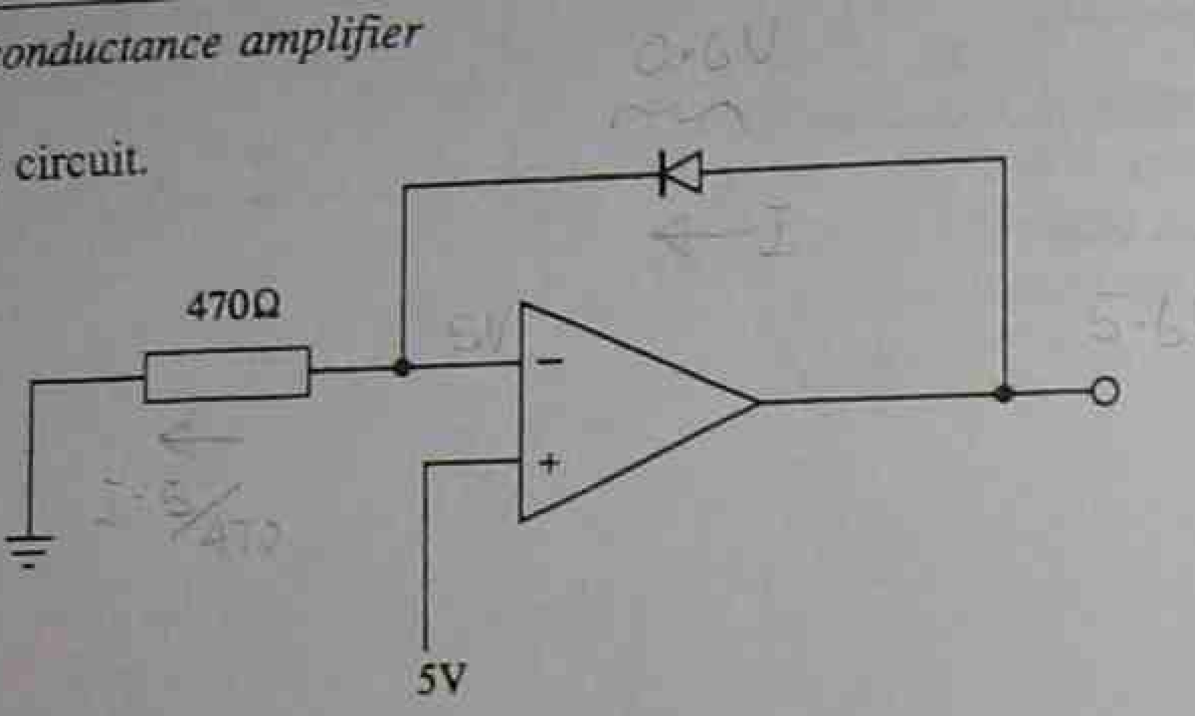
The circuit in Figure 10 is very similar to the non-inverting amplifier. Here, our aim is to pass a known current through the load, not amplifying the input voltage. The load can be any circuit components, possibly even diodes, rectifiers or meters. The purpose of the op amp is to isolate the source 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.

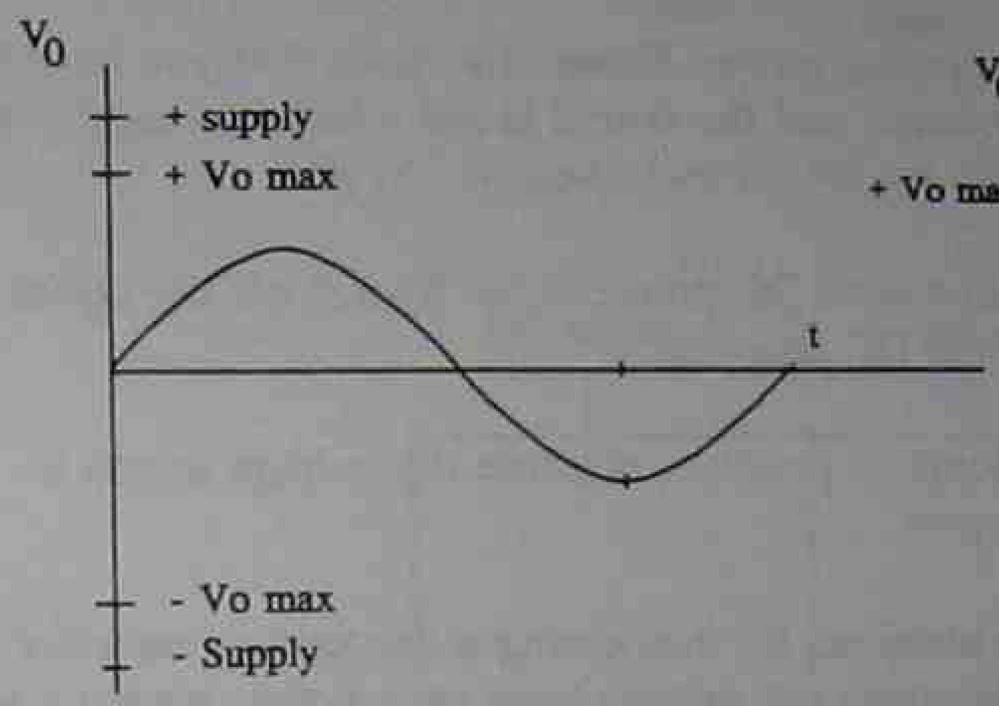


Fig. 11 Normal output

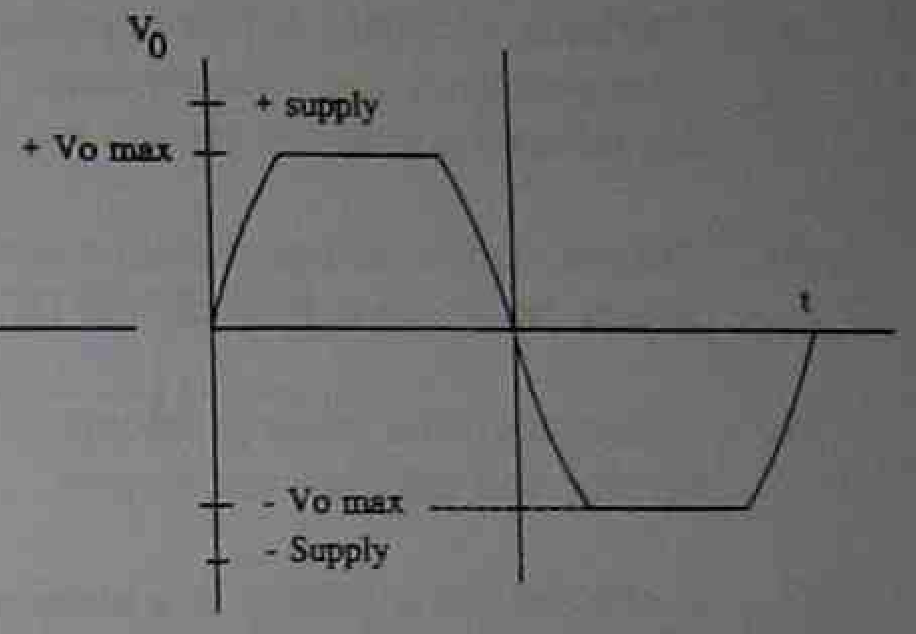


Fig.12 Clipped output

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.

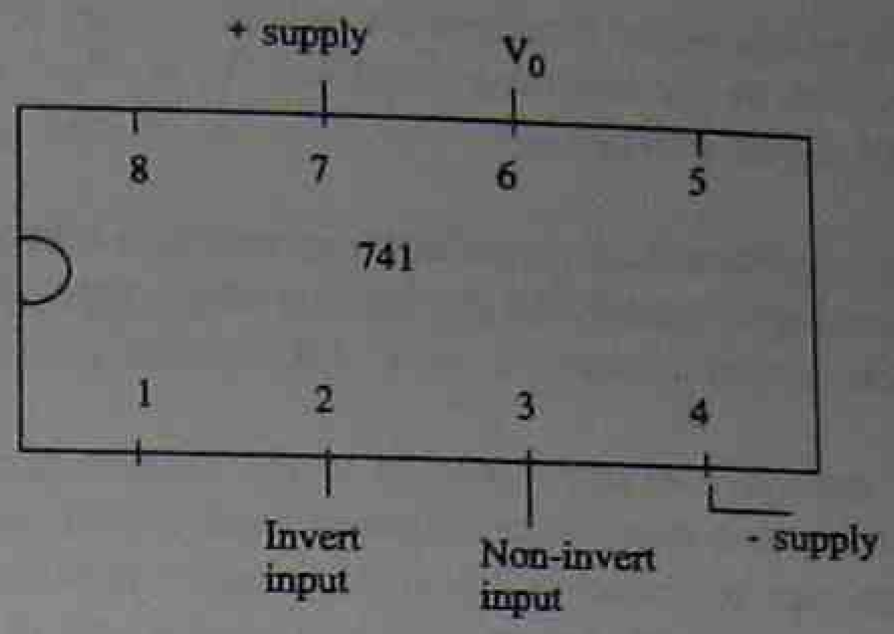


Fig.13 Pin connections for 741 and 351 type op amps

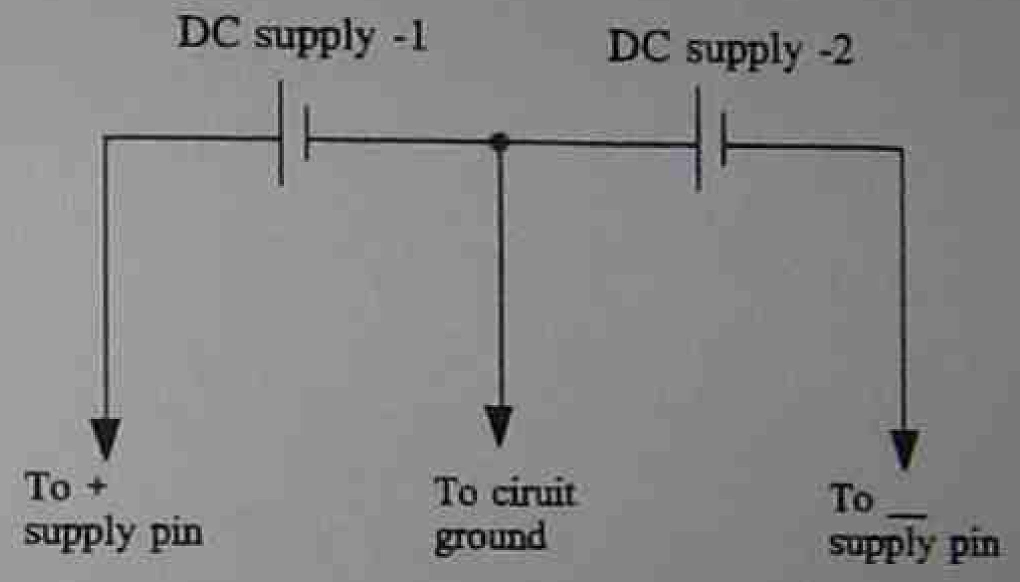


Fig.14 Dual power supply connections for op amps.

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?

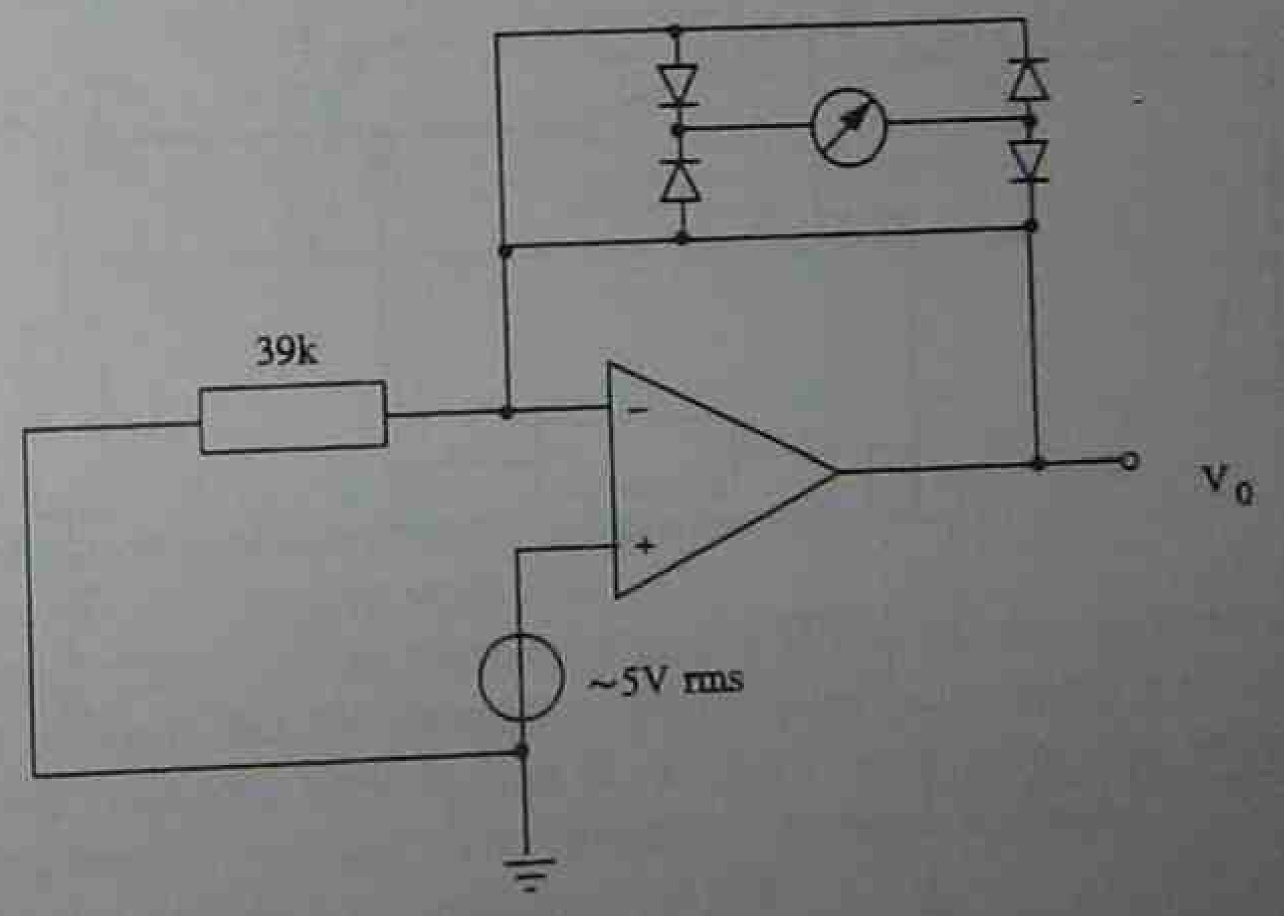
(b) Sketch an addition to the circuit in Q.1 to overcome the problem mentioned in part 2(a) above.

---

3. In the following circuit,

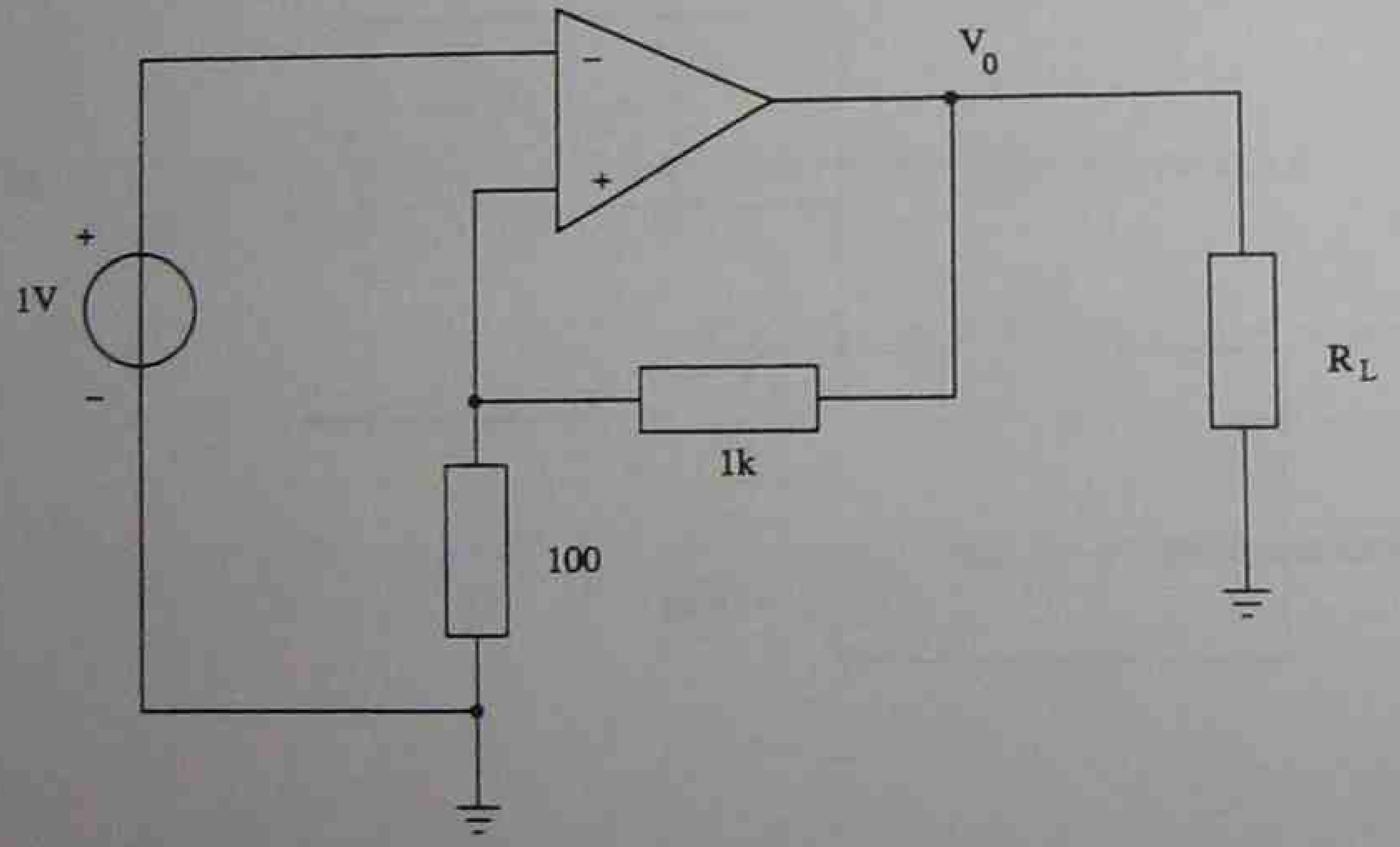
(a) What will be the average meter current if the input voltage is 5V rms sine wave? (Note: For sinewaves, full wave rectified average = 0.9 * rms)

(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

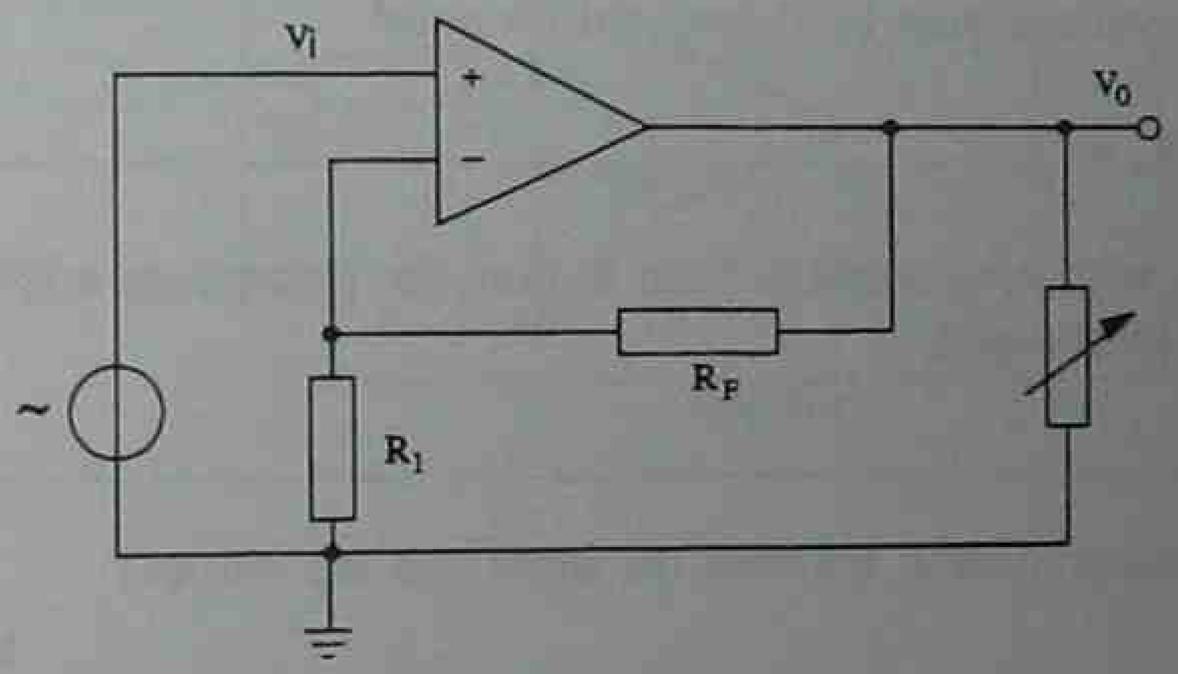
Tasks

- 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

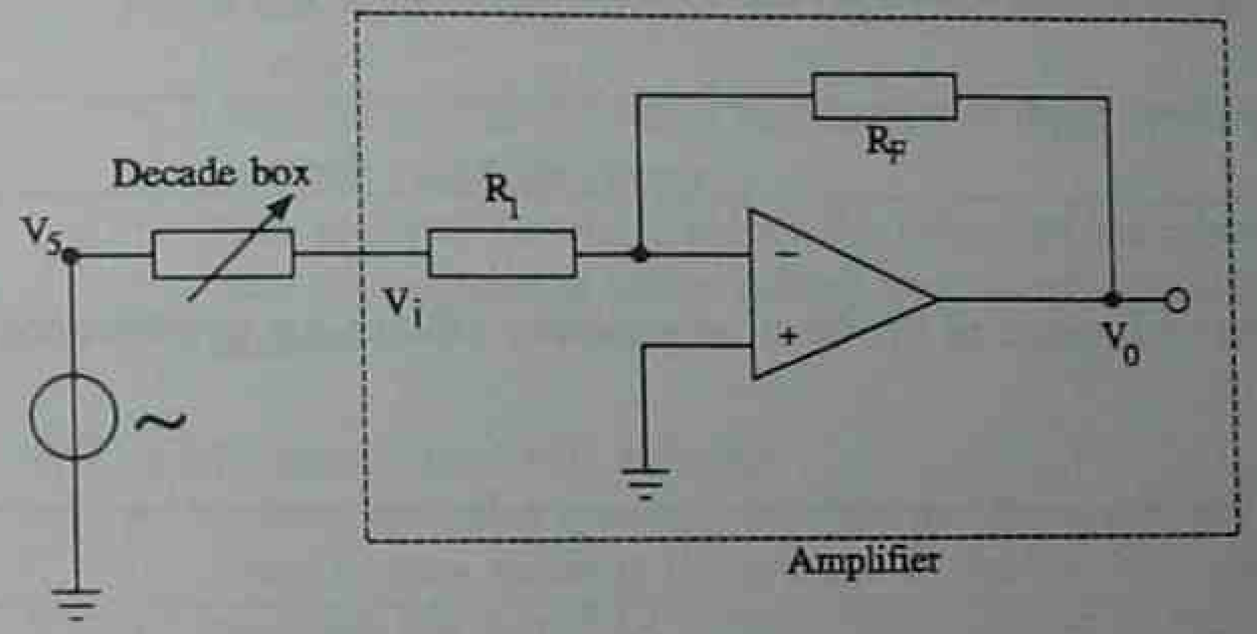
Task

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 amplifiers, 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

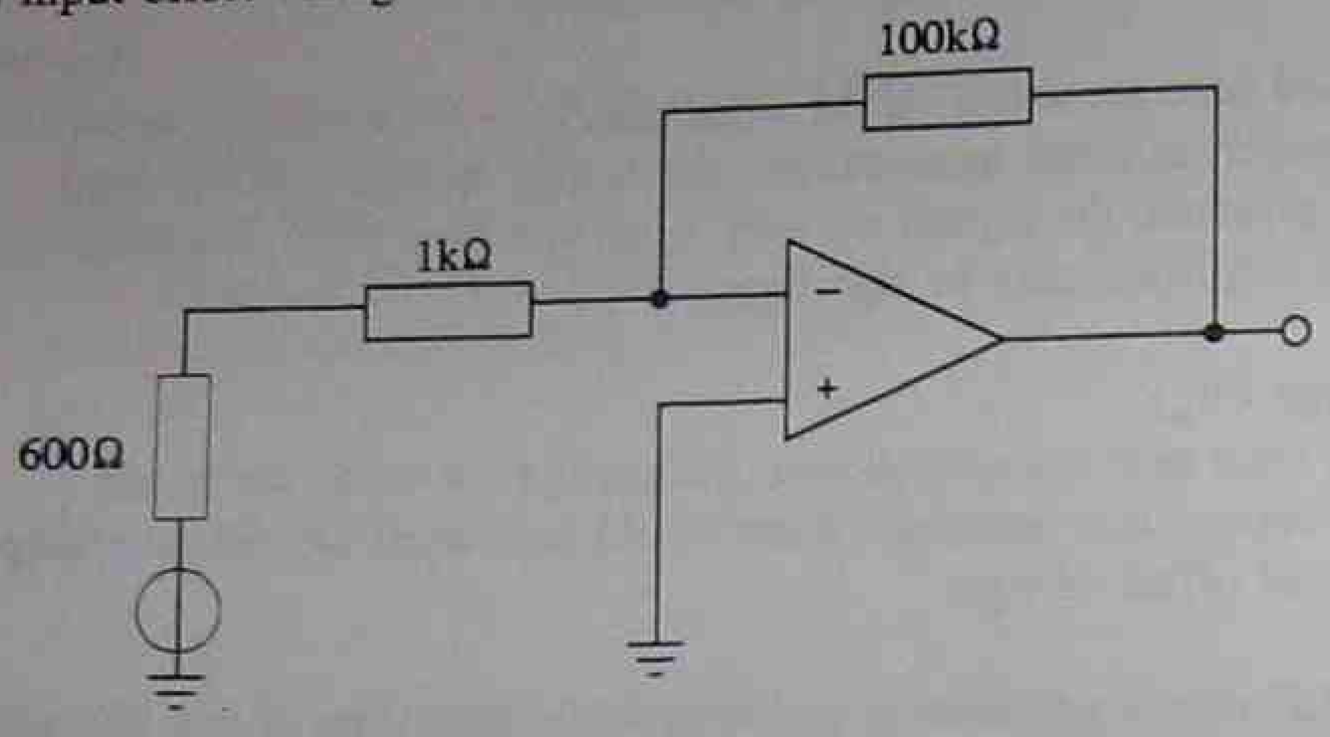


Fig. 1 Calculation of the effect of input offset voltage

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).

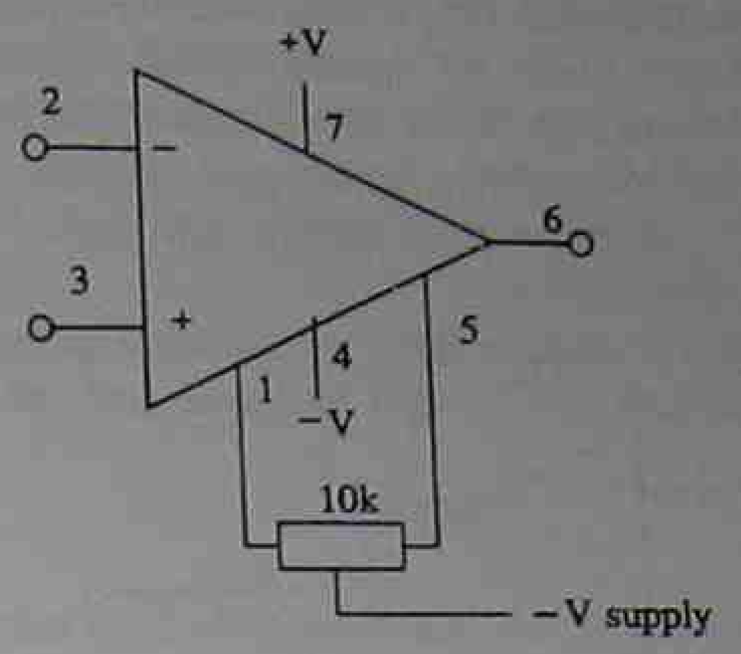


Fig. 2 Offset nulling for type 741 op amp.

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.

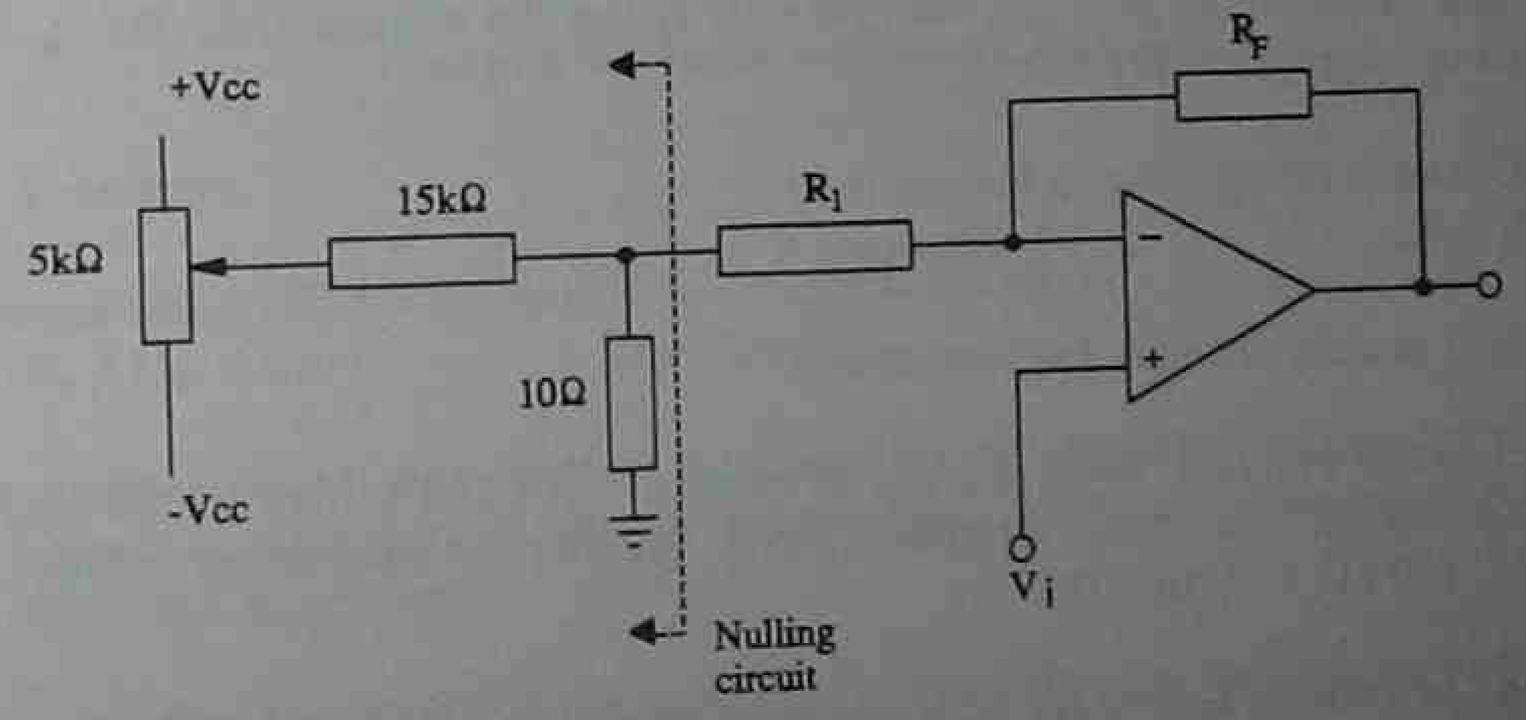


Fig. 3 Universal offset nulling circuit

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

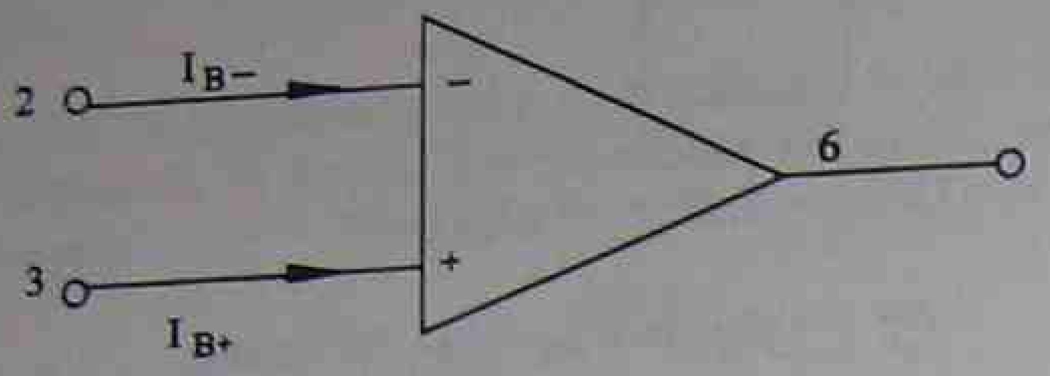


Fig. 4 Bias currents of an op amp

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).

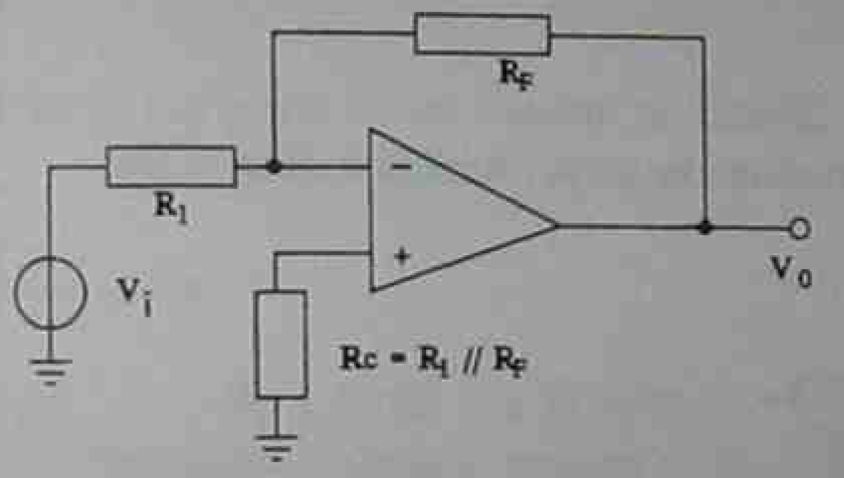


Fig. 5

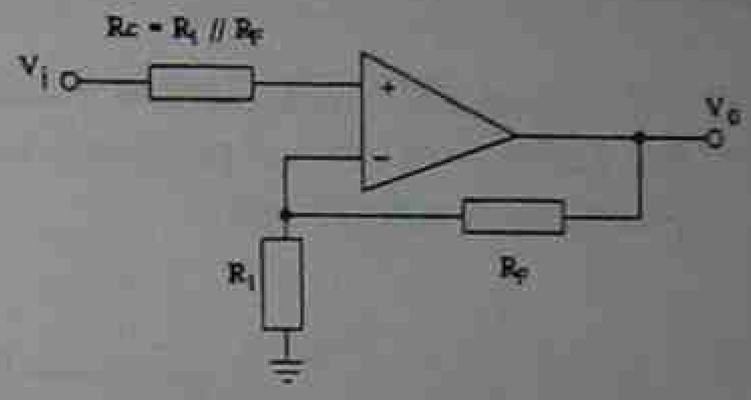


Fig. 6

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

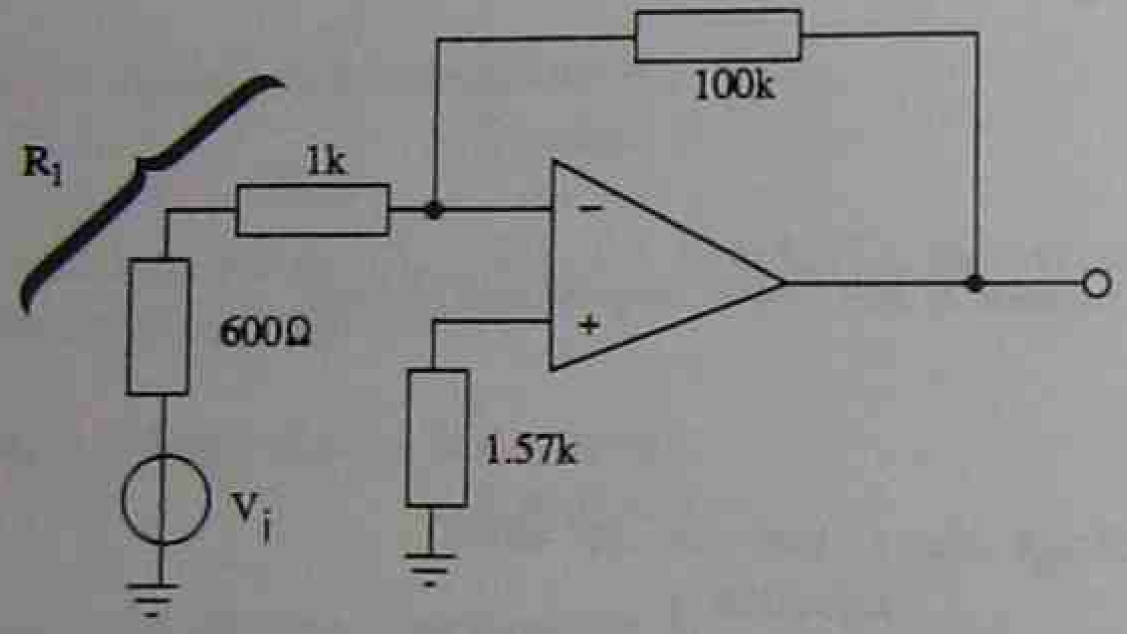
(a) For the circuit of Figure 1 design a bias current compensation.

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

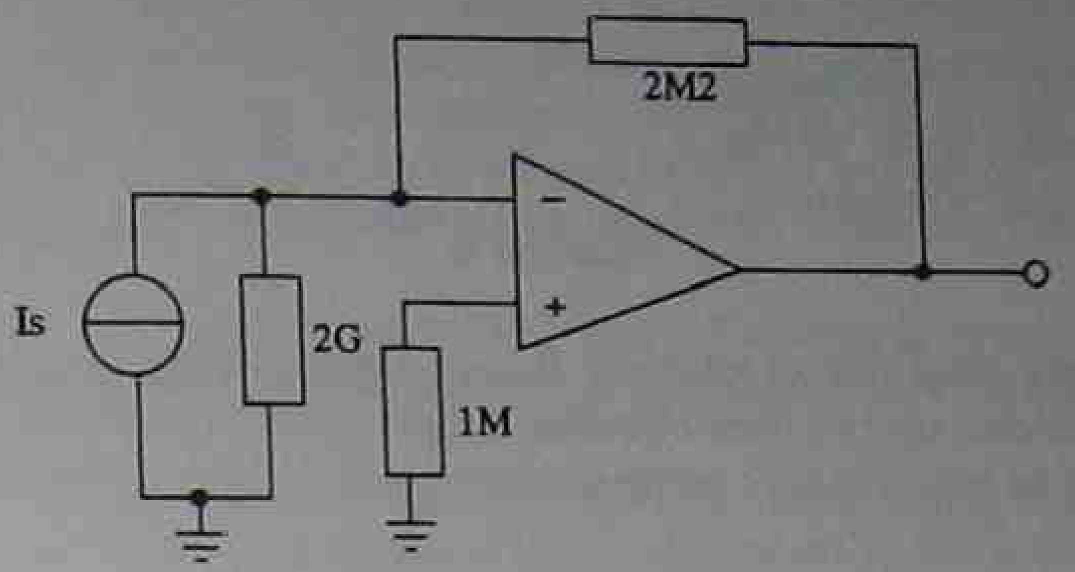
Put 1.57kΩ (or nearest preferred value) resistor in series with the +input for bias current compensation.



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

Example 2 : Bias current compensation and its effect

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



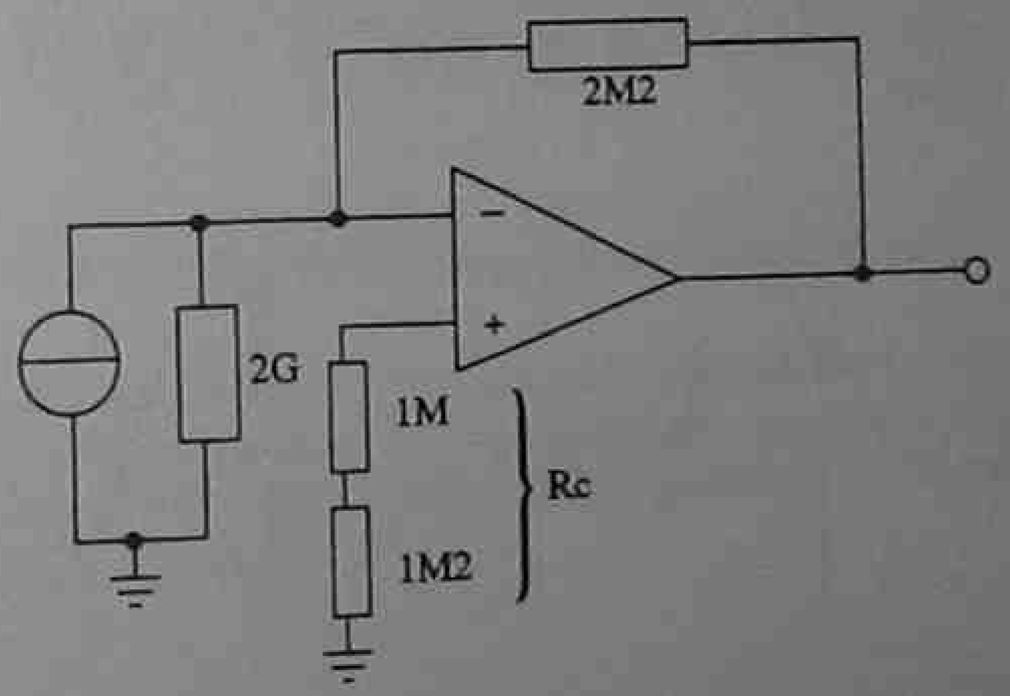
(b) If the input offset current is 10 nA, what is the output DC offset voltage due to bias currents after compensation ?

Solutions

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

Resistance already present from +input to ground = 1M

$\therefore$ 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

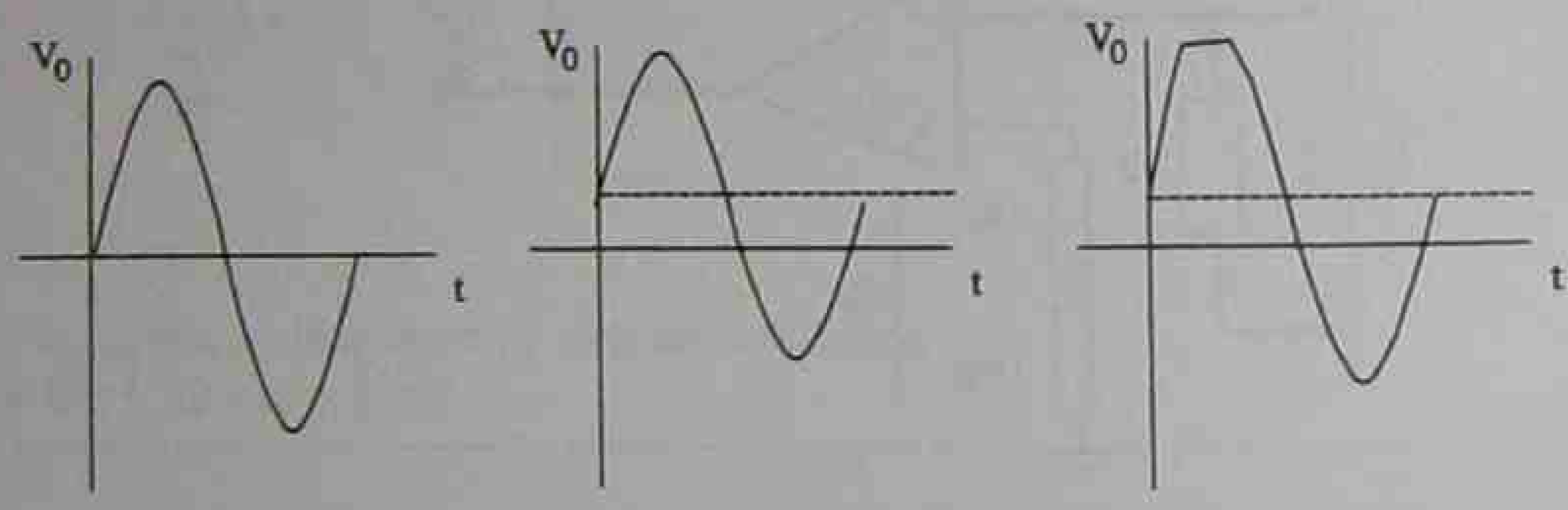
The first term on the right hand side of equation 9 is calculated using equation 1. The second term is calculated using either equation 6/equation 8 or equation 3/equation 4, depending on whether bias compensation resistor is used or not used.

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

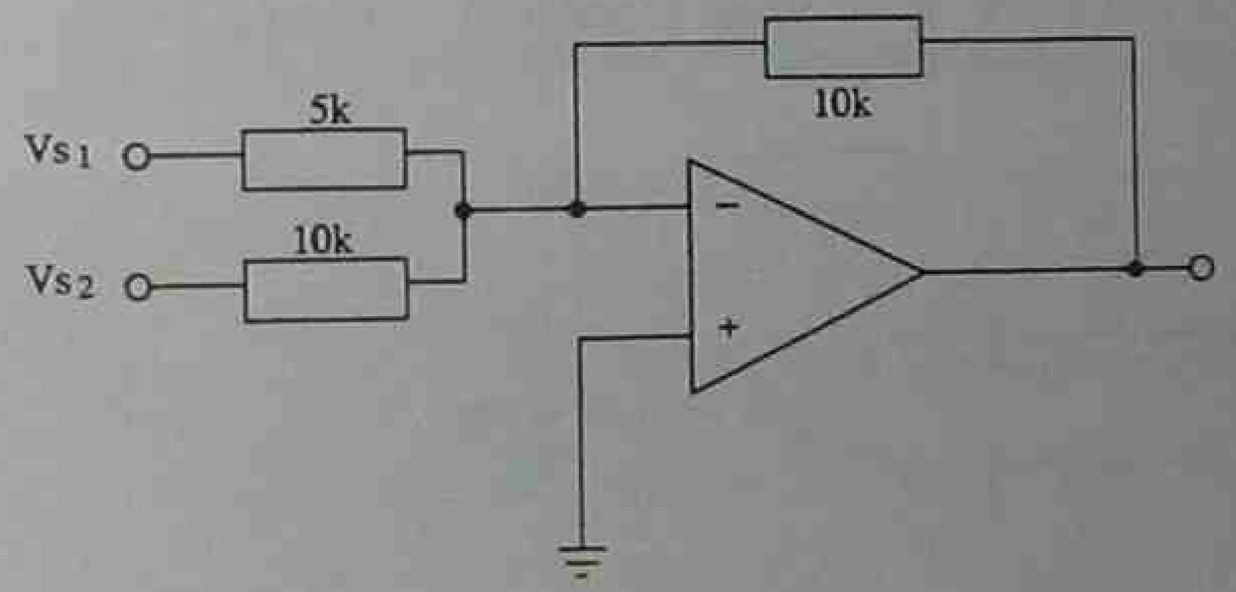
These questions will help you revise what you have learnt in Section 2.

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.

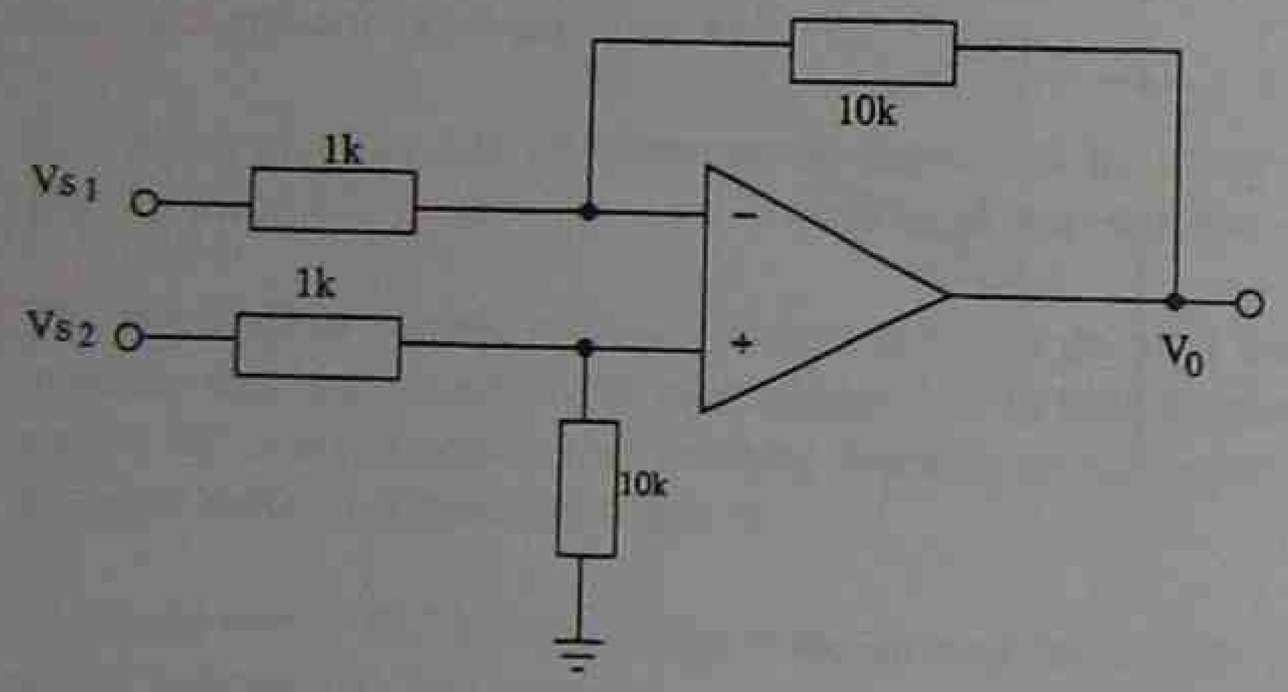


2. For the differential amplifier circuit shown below, the amplifier has input offset voltage 2 mV, input bias current 80 nA and input offset current 20 nA.

   (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.

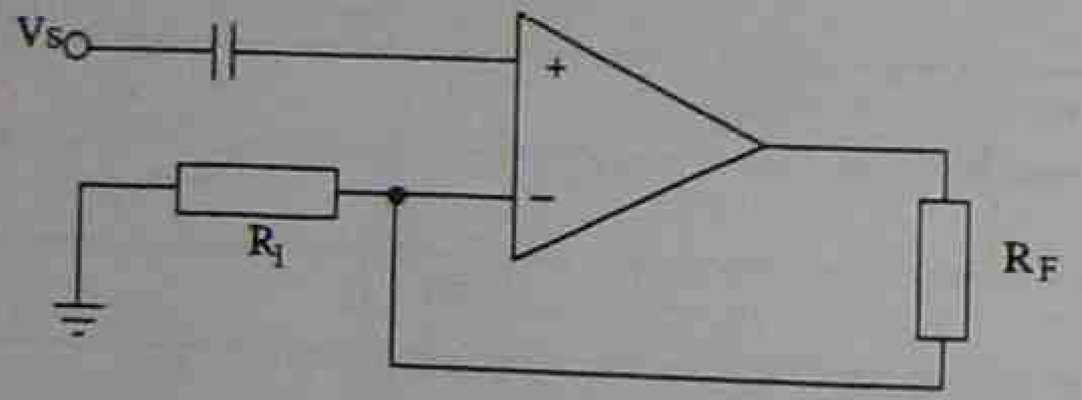
4. What are common ways to avoid errors due to drift in DC offsets in amplifiers?

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.

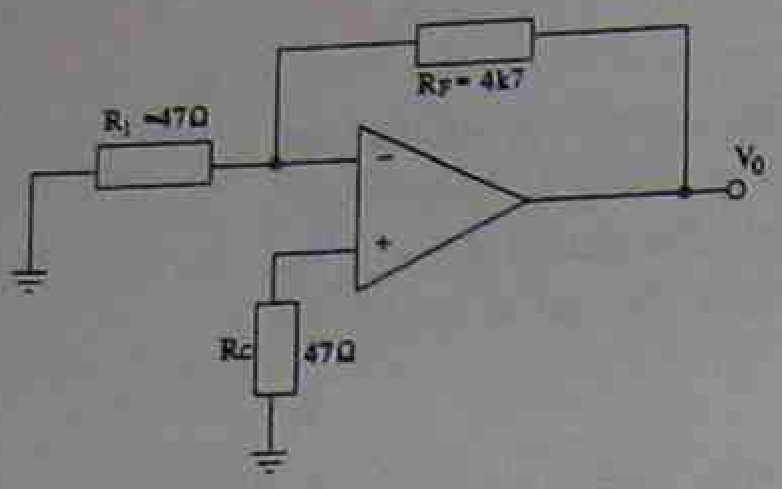
No external signal source is necessary. In the following circuits, the input is connected to ground.

(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}

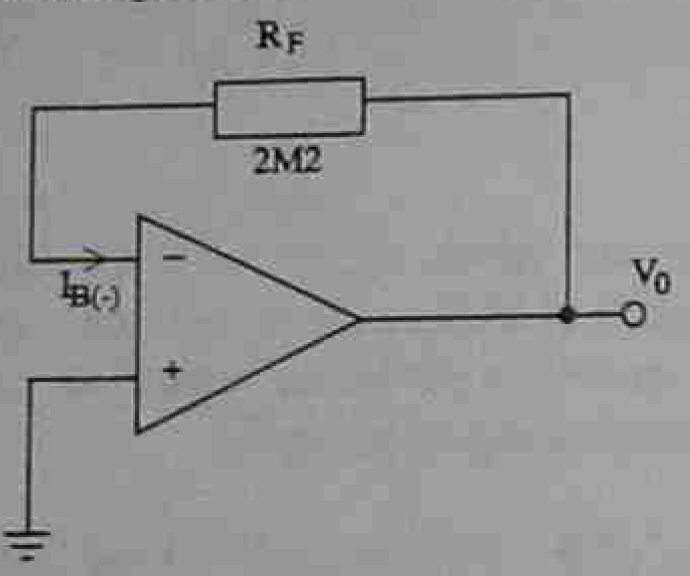
- Connect the following circuit and measure the output DC voltage V_o.



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

Step 2 Measurement of bias current I_{B-}

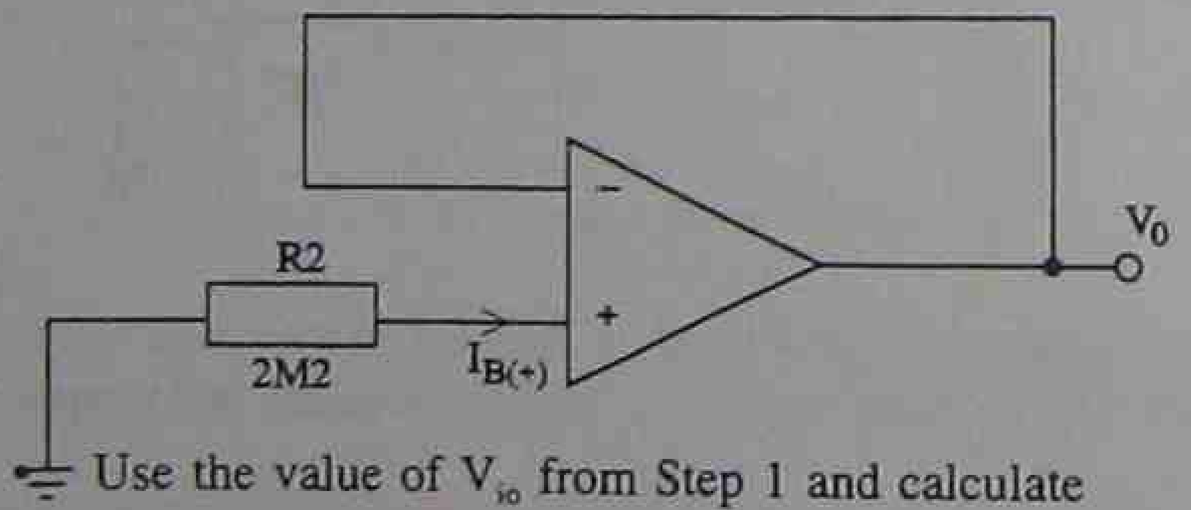
- Connect the following circuit and measure the output DC voltage V_o.



- 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.

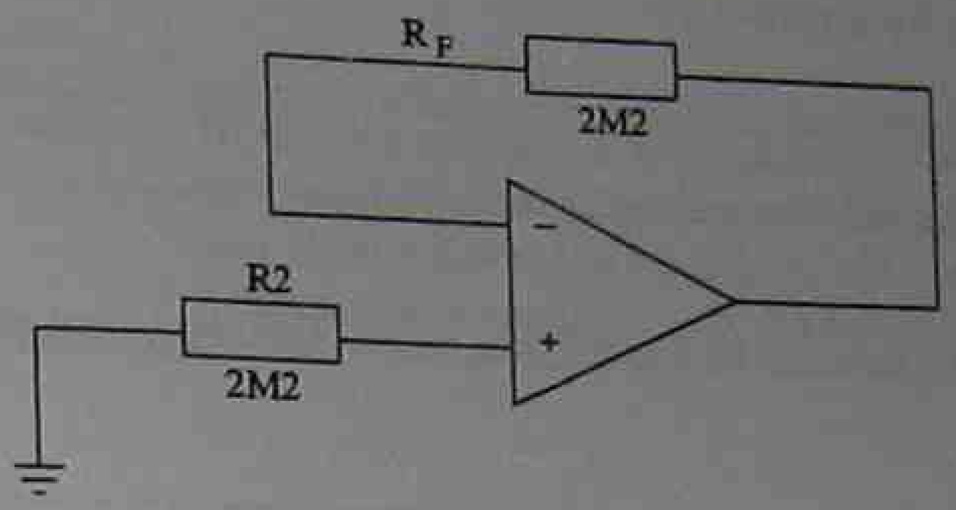


- Use the value of V_{io} from Step 1 and calculate

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

Step 4 Measurement of input offset current I_{os}

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



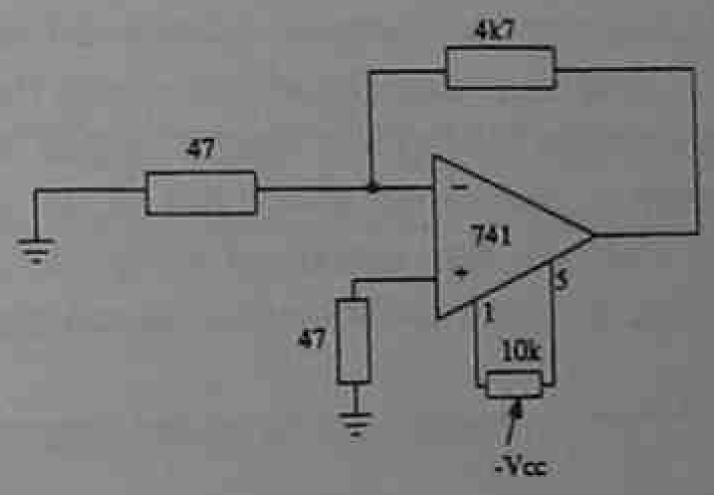
- Use the value of V_{io} from Step 1 and calculate

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

- 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.

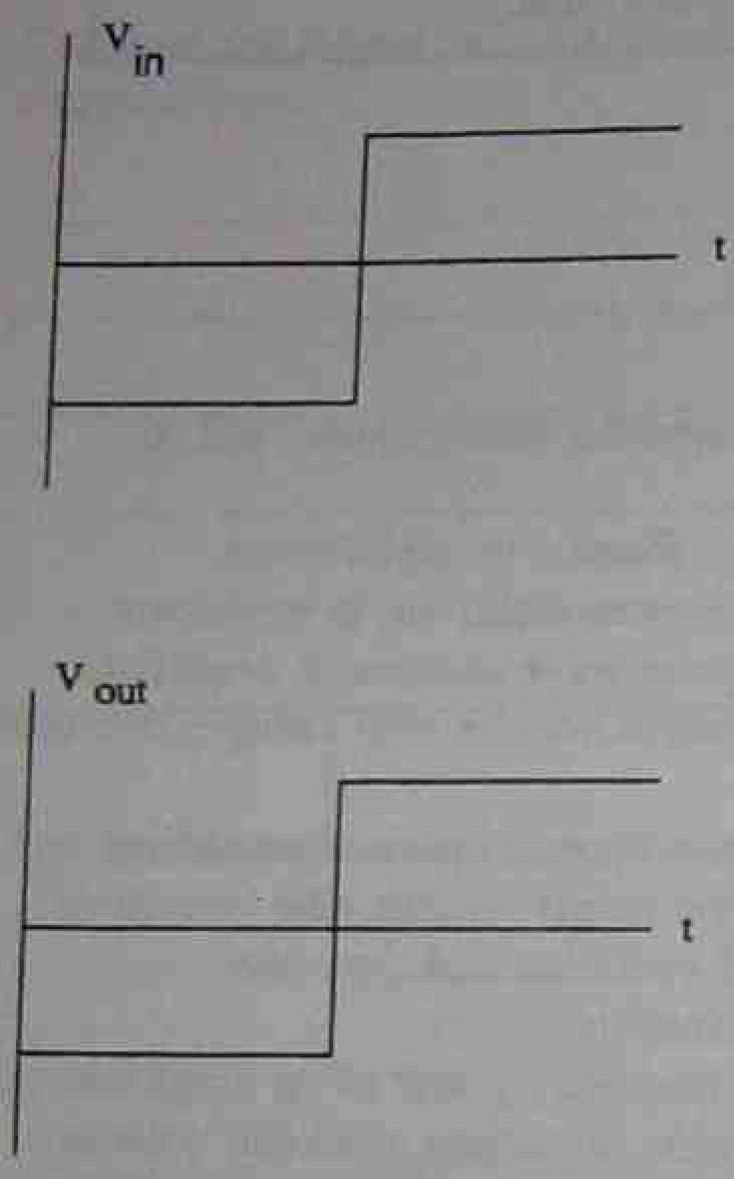
References

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).



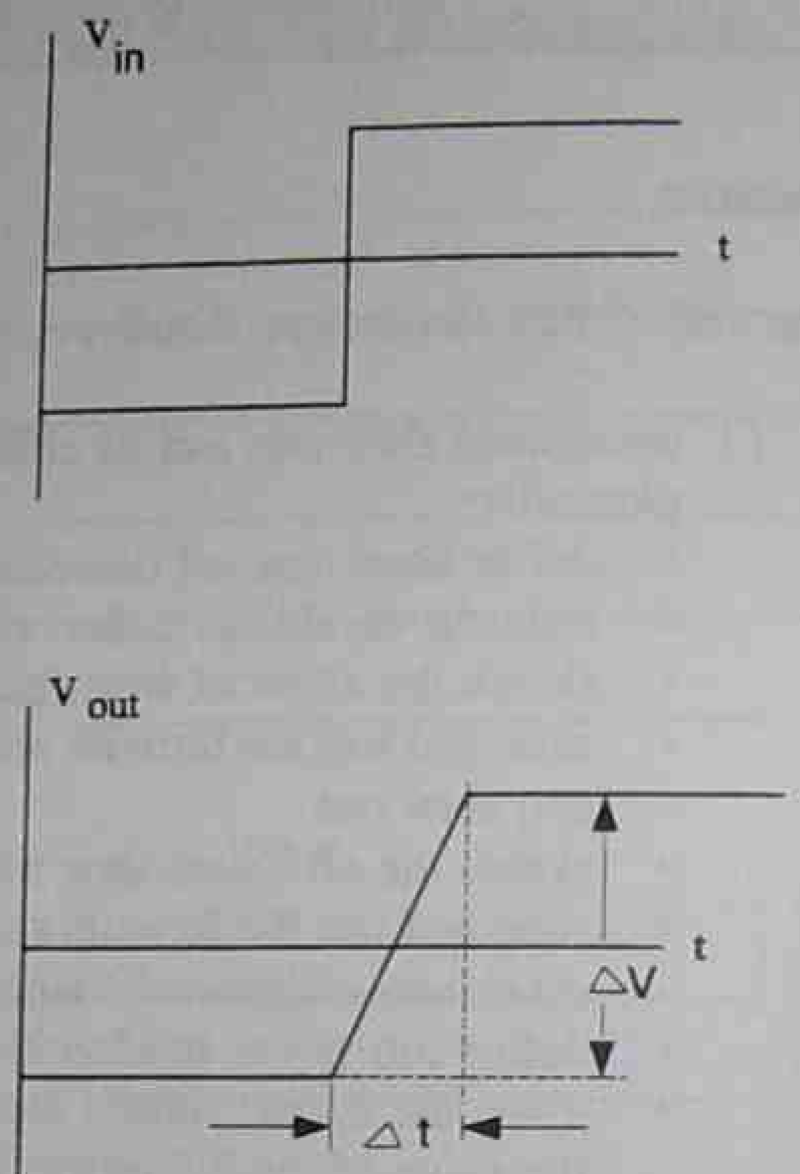


Fig. 1 (Ideal output)

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) 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).

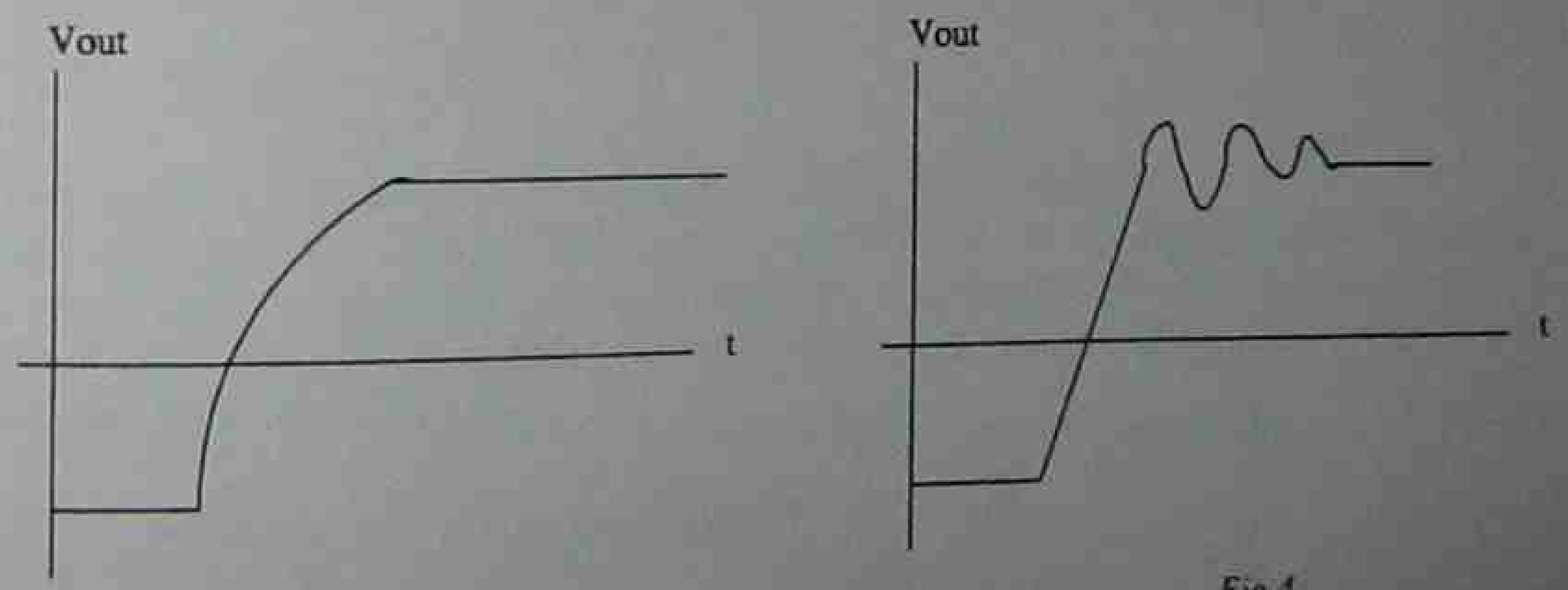


Fig.3

Fig.4

Small signal (bandwidth limited) square wave outputs

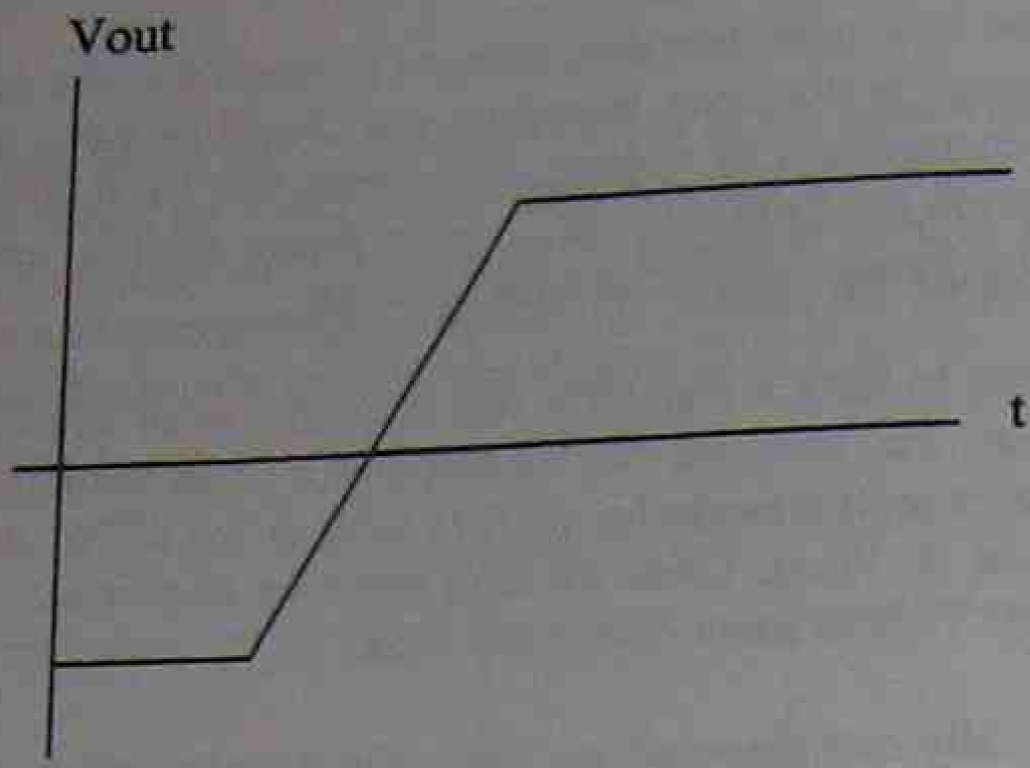


Fig.5. Large signal (slew Rate limited) square wave output

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.

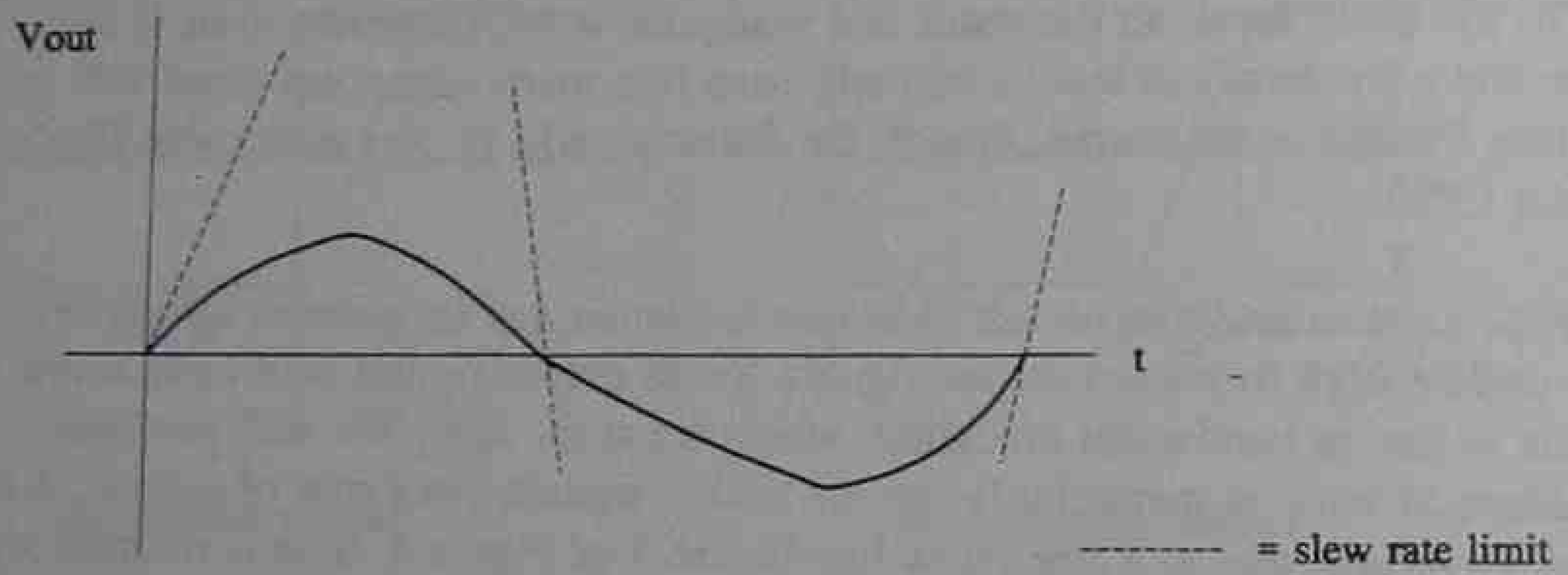


Fig.6 Undistorted sine wave output

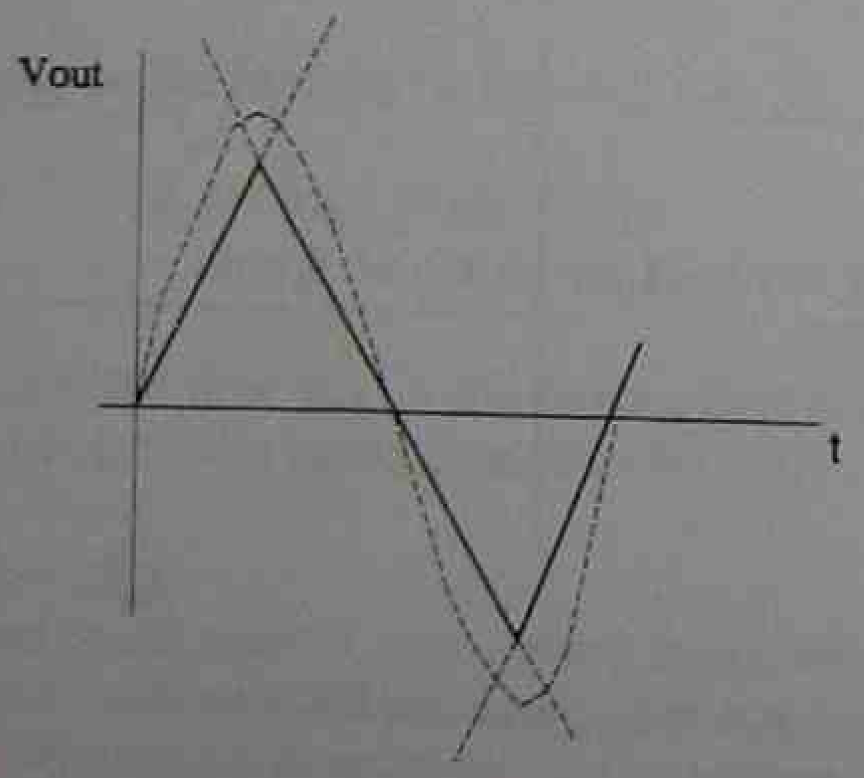


Fig. 7 High frequency sine waves are slew rate distorted

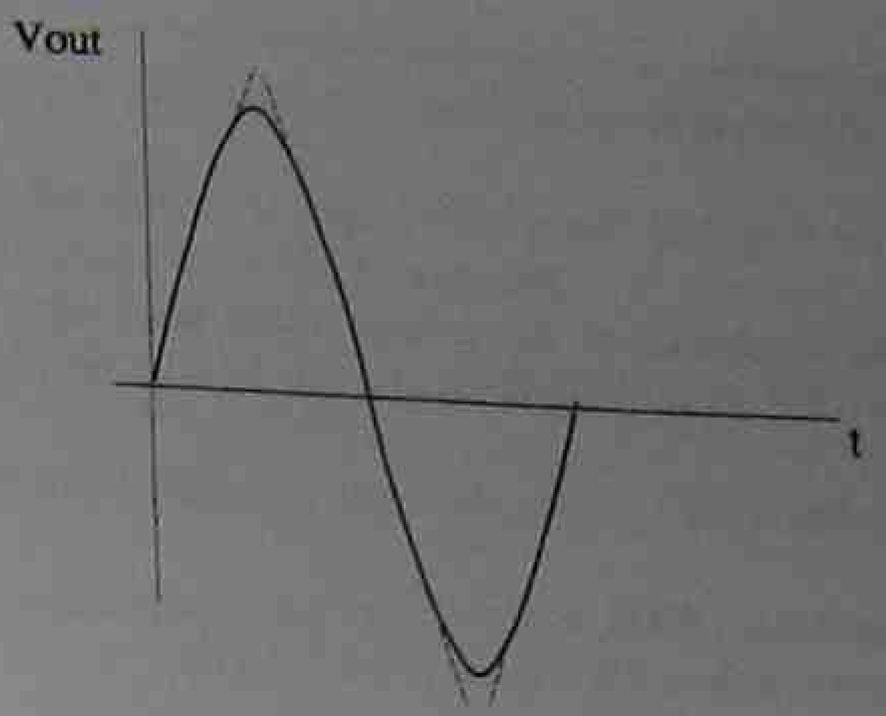


Fig.8. High amplitude sine waves are slew rate distorted

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:

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

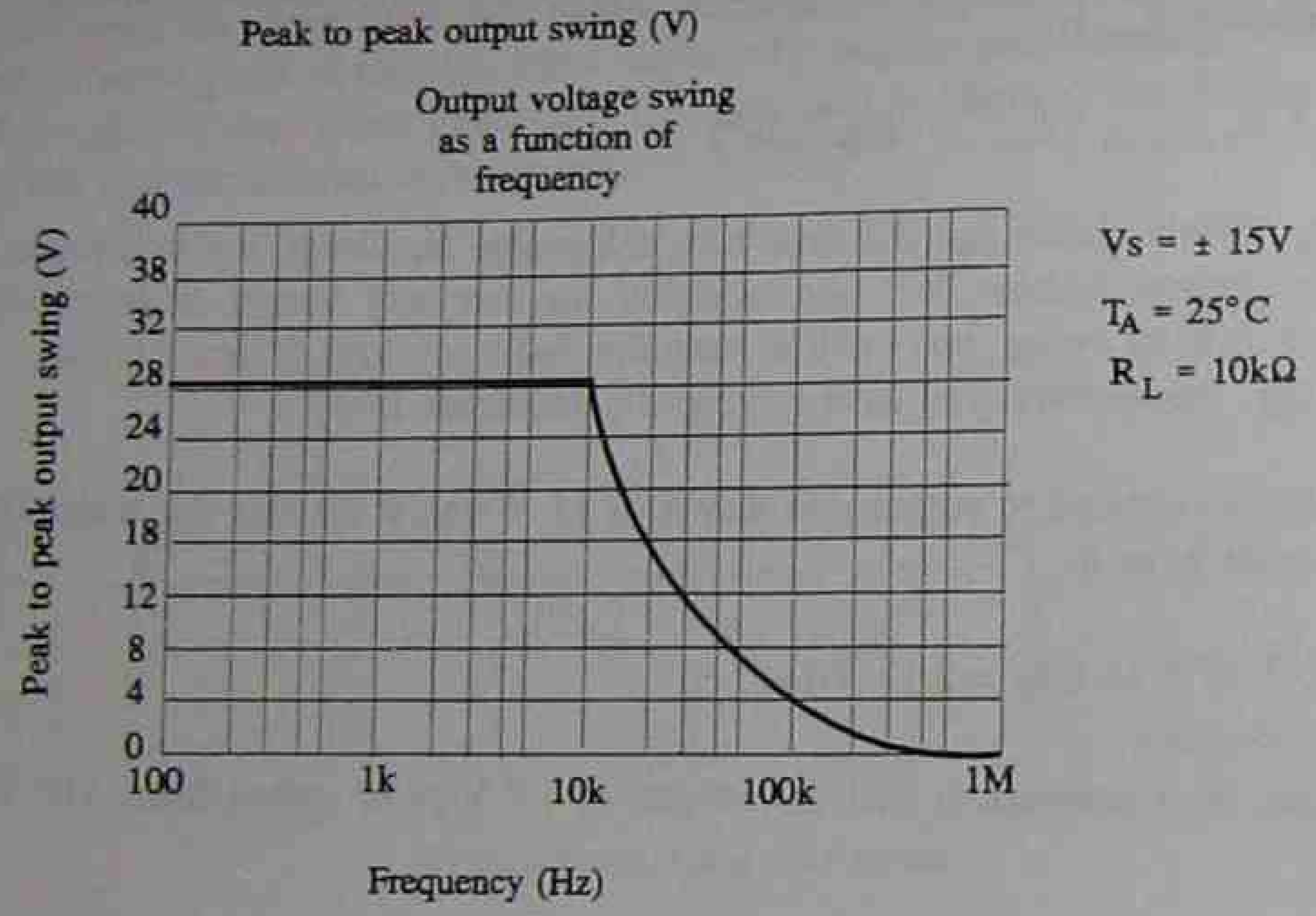


Fig. 9. Output voltage swing graph of the 741 op amp

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)

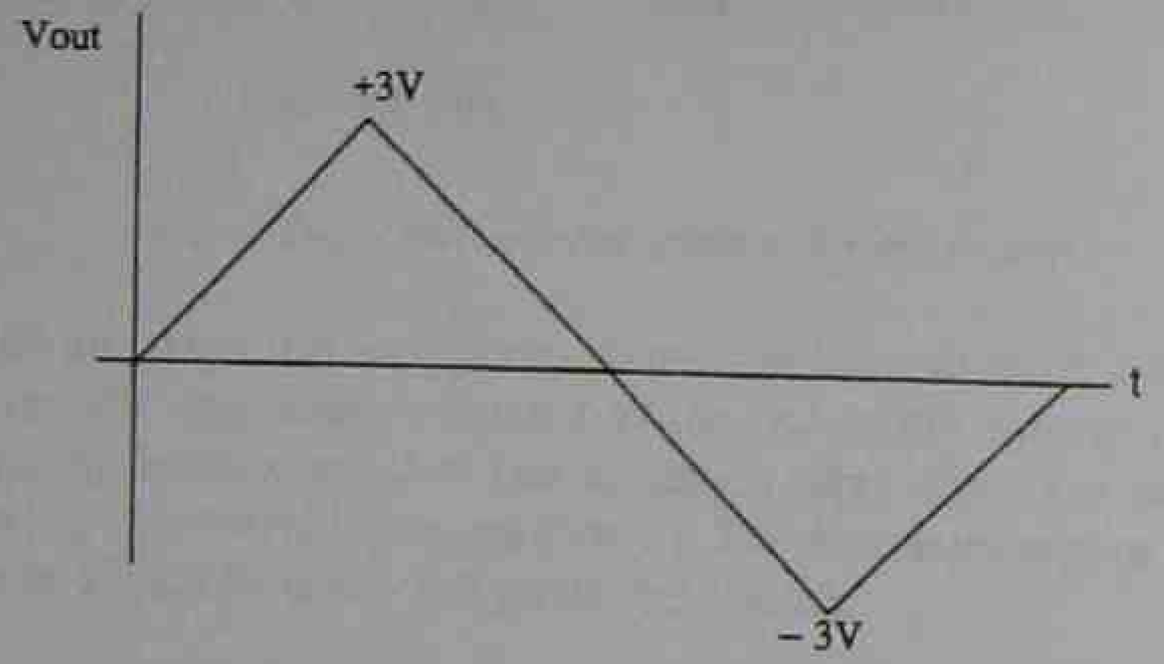
(c) From the graph, at 20 kHz, max. $v_{o\ p\text{-}p}$ = 14V
max v_i rms = 14V / (2 √ 2) / gain = 0.5 V rms
The output is slew rate limited above FPBW
(Note : you could have calculated the answer to part (c) using the slew rate and equation 2. The answer would be about the same, except for the human error in reading graphs.)

Review questions

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

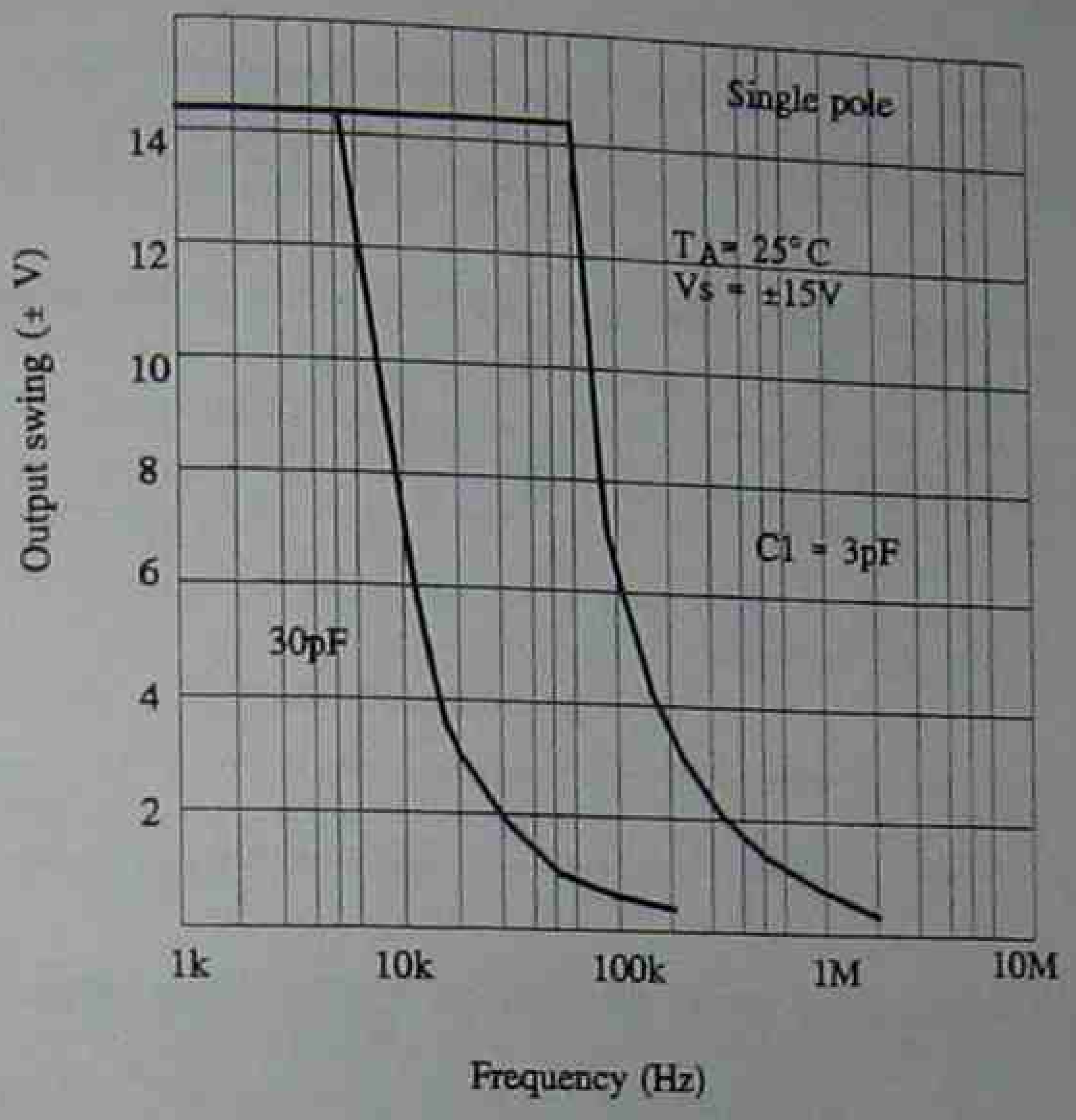
1. An operational amplifier with a slew rate of 0.5 V/μs is connected as a voltage follower and a square wave input at 62.5 kHz, 6V peak to peak is applied. Sketch the output waveform, carefully marking important time and voltage values.

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

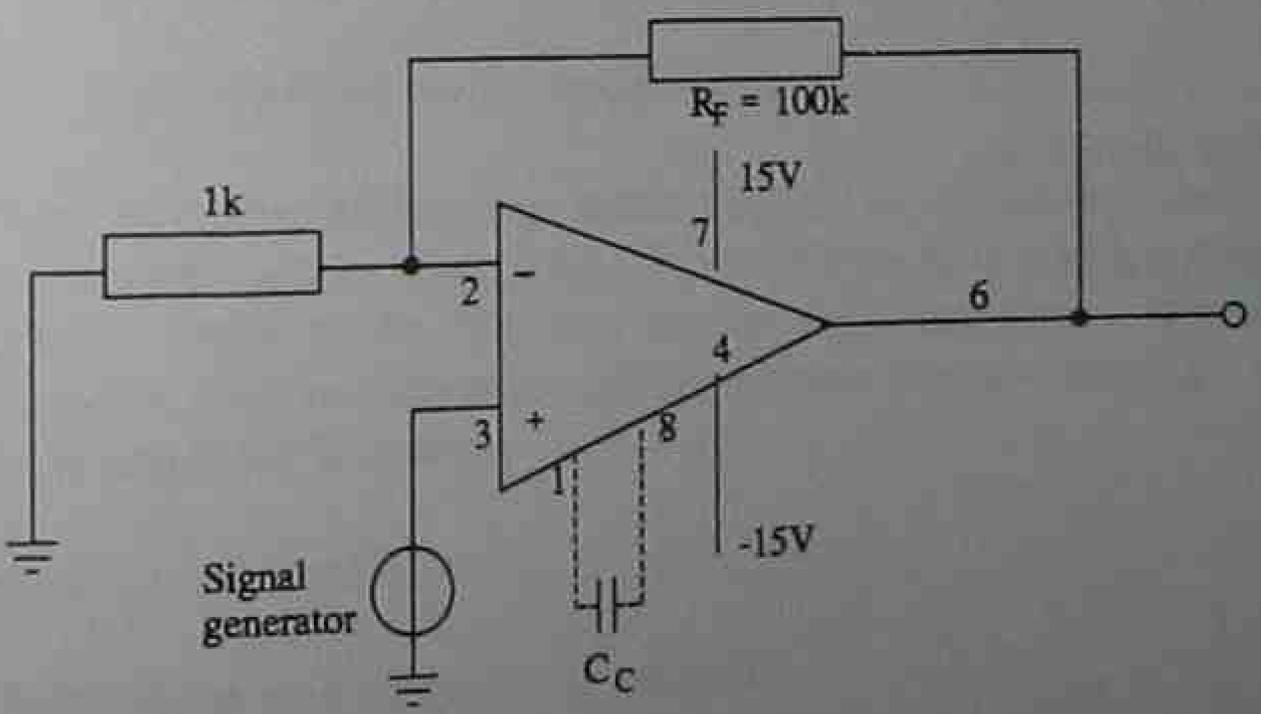
- Either one op amp type LM301 and a 30 pF capacitor OR one op amp type 741. (741 is not suitable for Step 6.)
- General purpose 15 MHz oscilloscope (one with time-base x5 or x10 switch preferable)
- AC signal source (e.g.function generator)
- Dual DC power supply (fixed ±15V supplies can be used)
-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

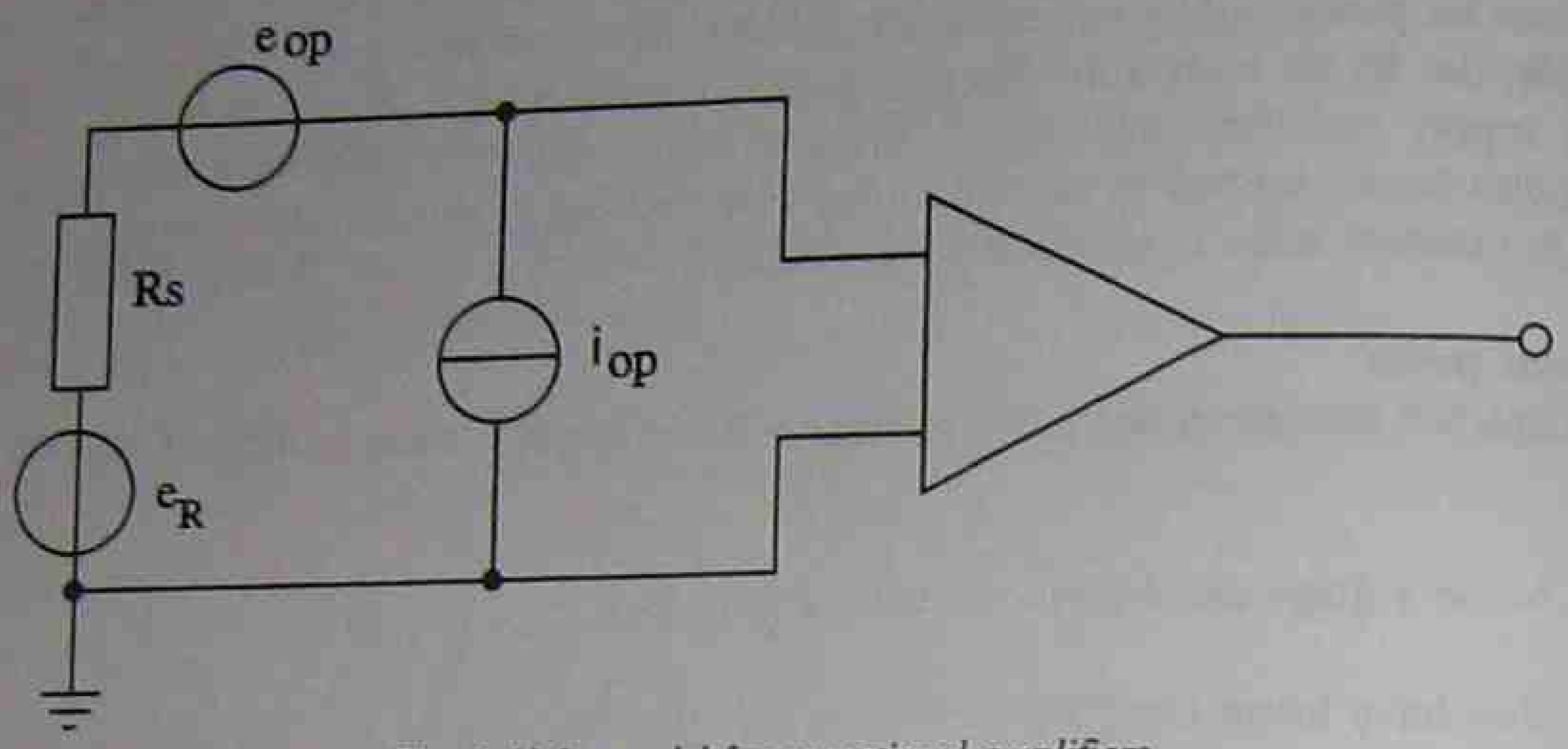


Fig. 1 Noise model for operational amplifiers

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

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

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

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
$= 15\ nV/\sqrt{Hz} * \sqrt{30kHz} = 2.6\ \mu V$ 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 =

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

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.

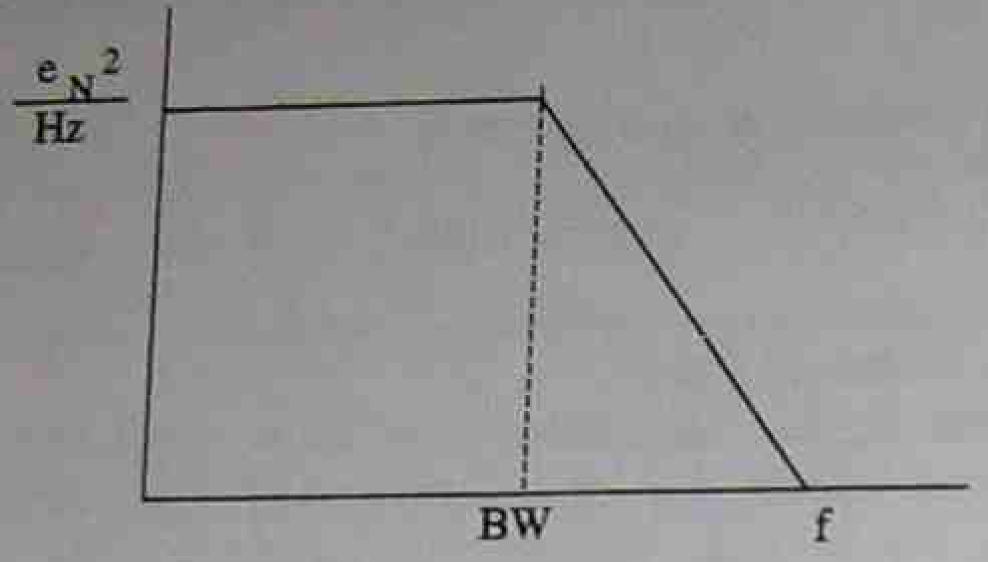


Fig. 2 Practical Bandlimiting Filter response

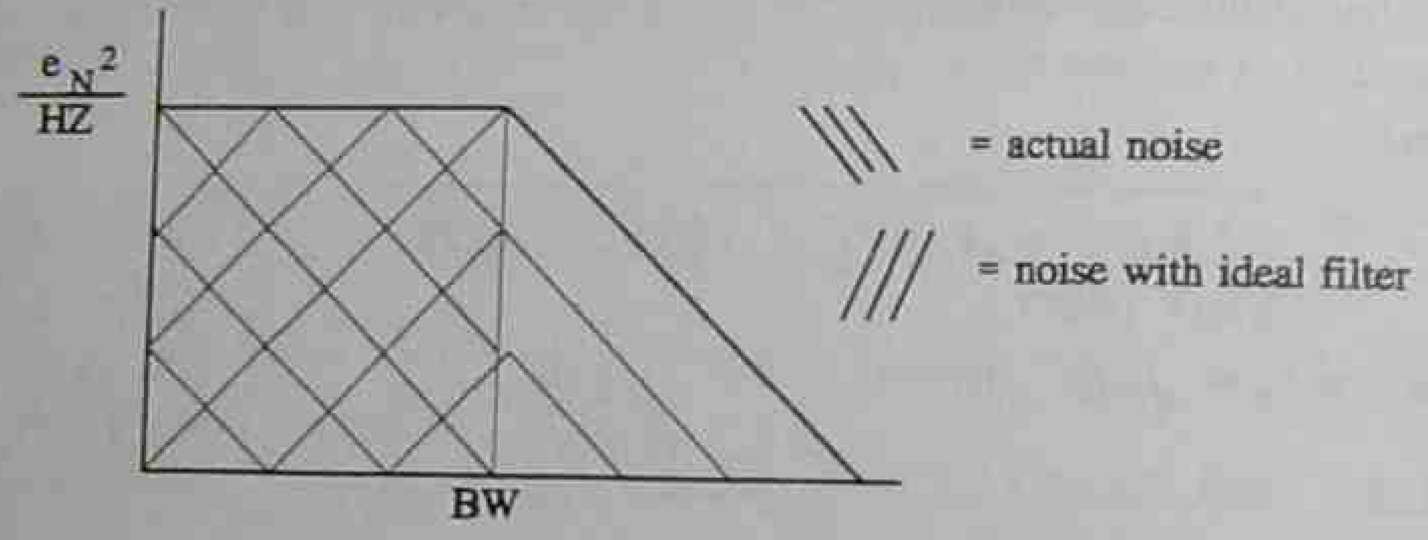


Fig. 3 Comparison of noise allowed in by ideal and practical bandlimiting filters

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).

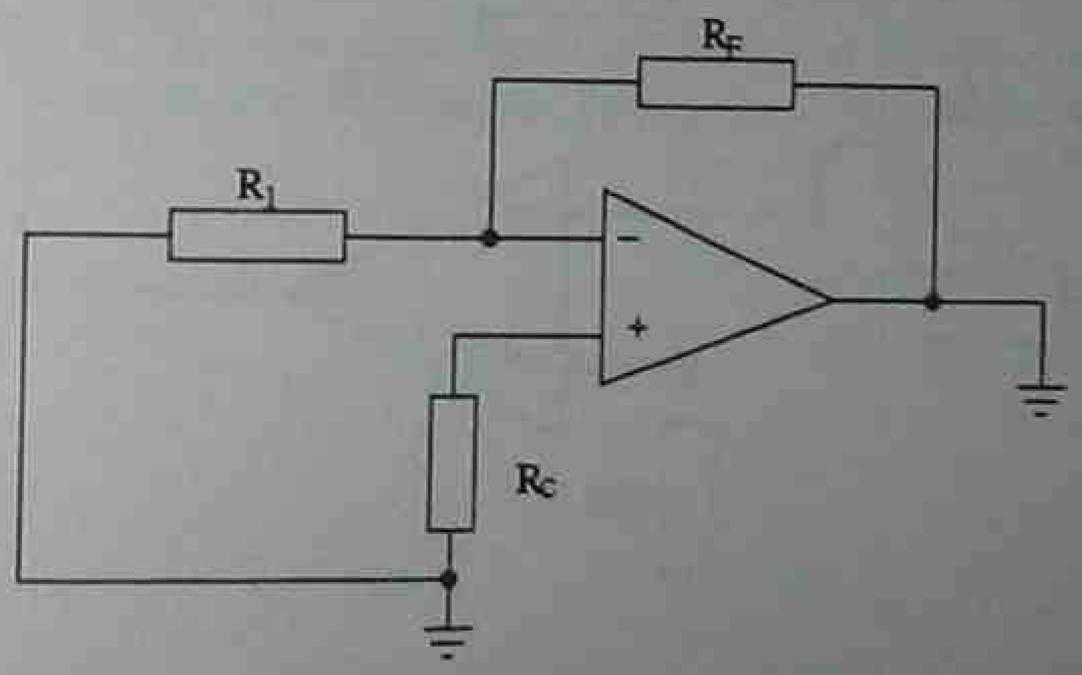


Fig. 4 Negative feedback amplifier

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

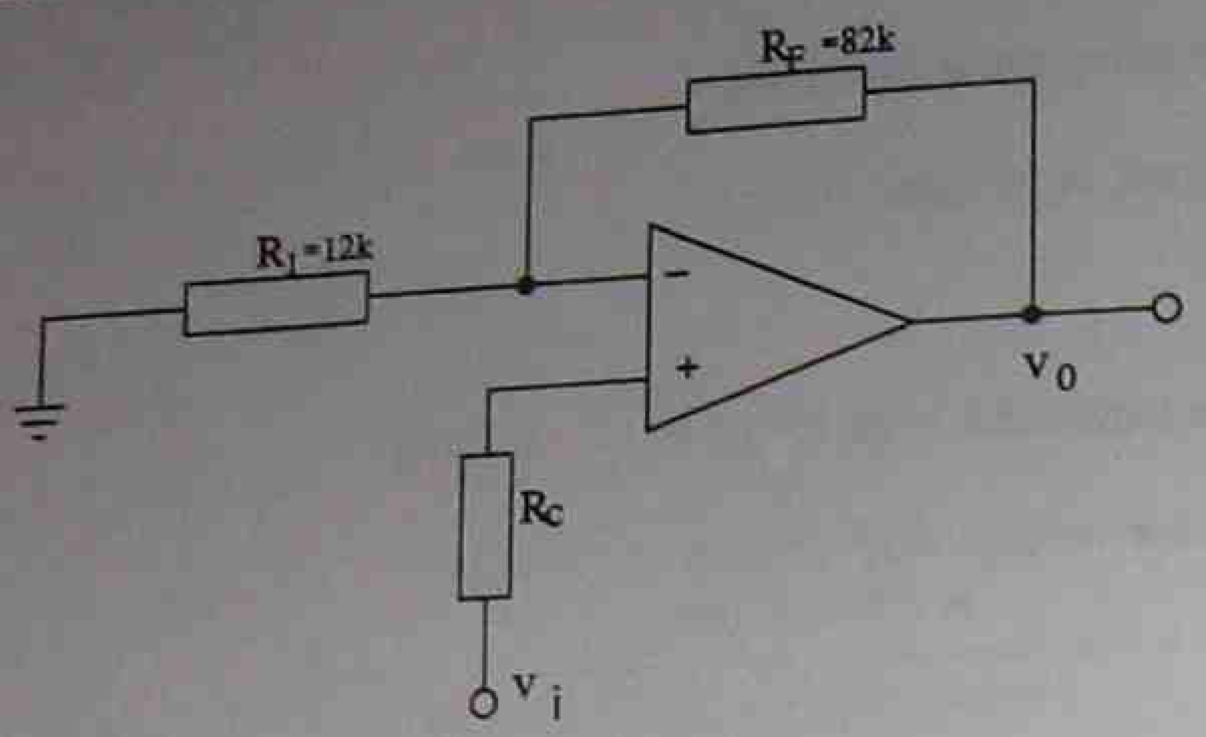
As before, R_c and R_1 are the effective resistances from the +input and -input to ground respectively, after simplifying any series and parallel combinations.(To be exact, the answer in equation 8 must be paralleled with the input resistance of the op amp. Practically, the input resistance is very large 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



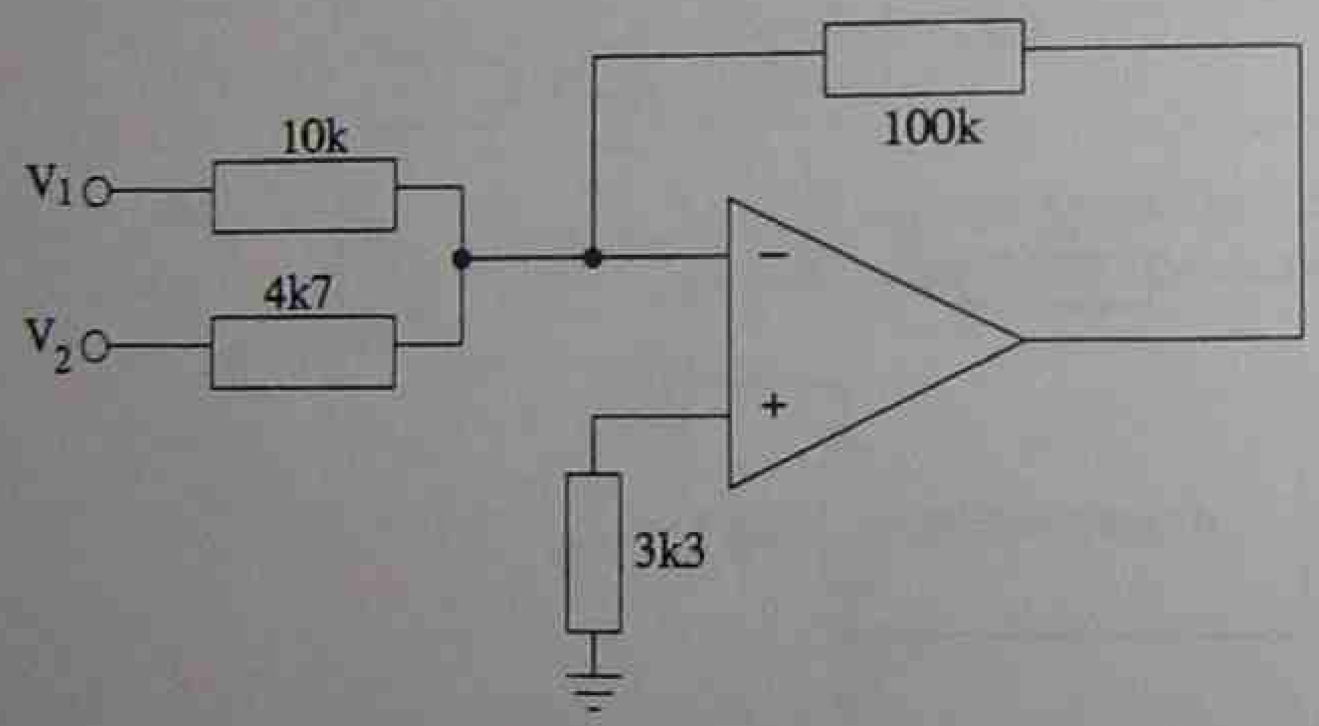
Solutions

(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.

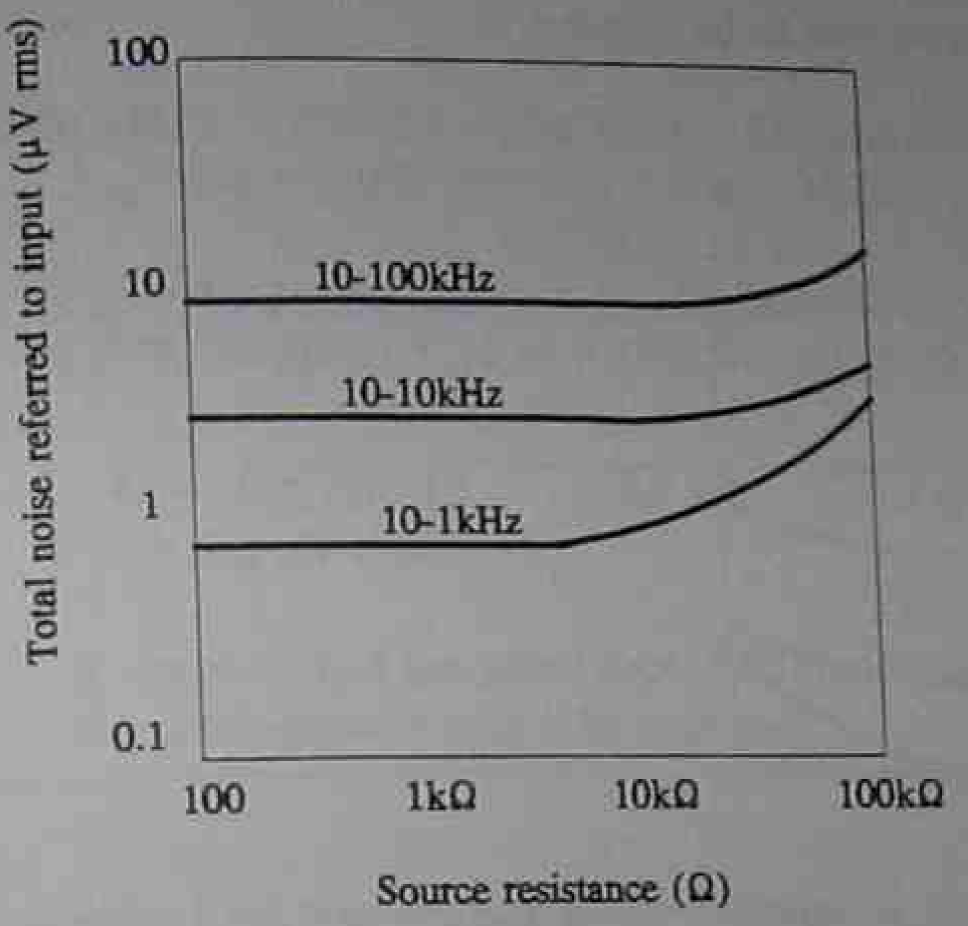


Fig. 5 : Broadband noise graph for 741 op amp

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.

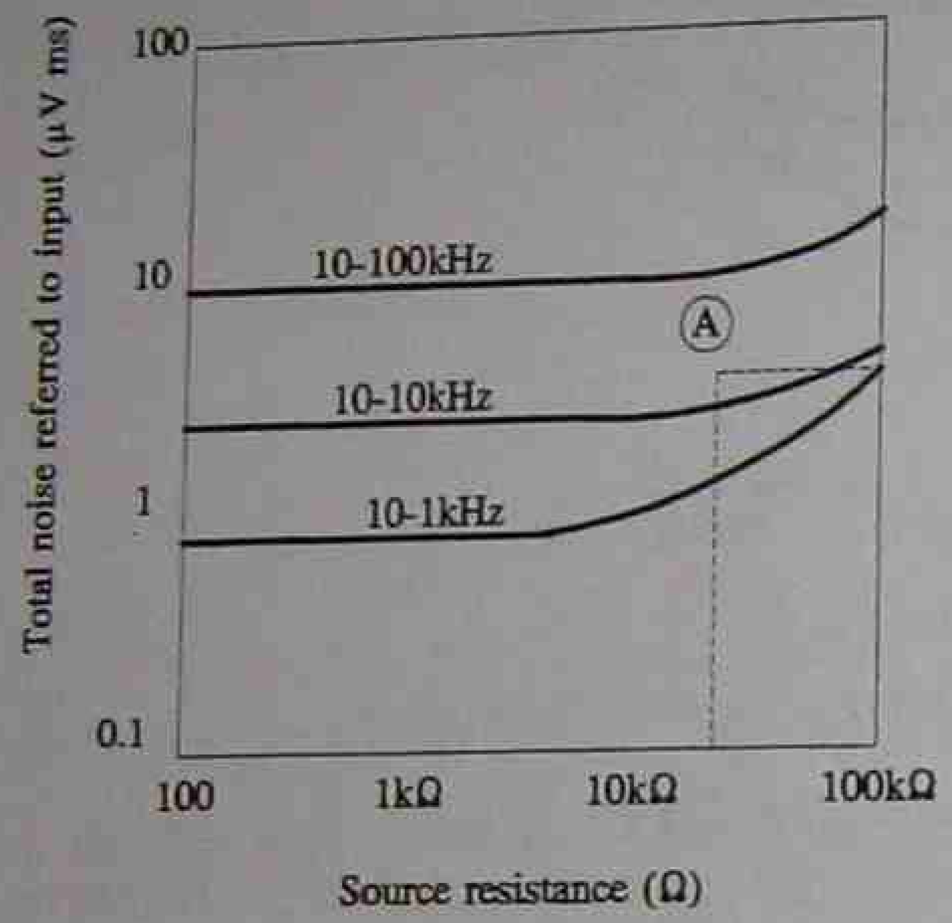
Example 8: Using the broadband noise graph

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

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

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/√Hz (or A/√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.

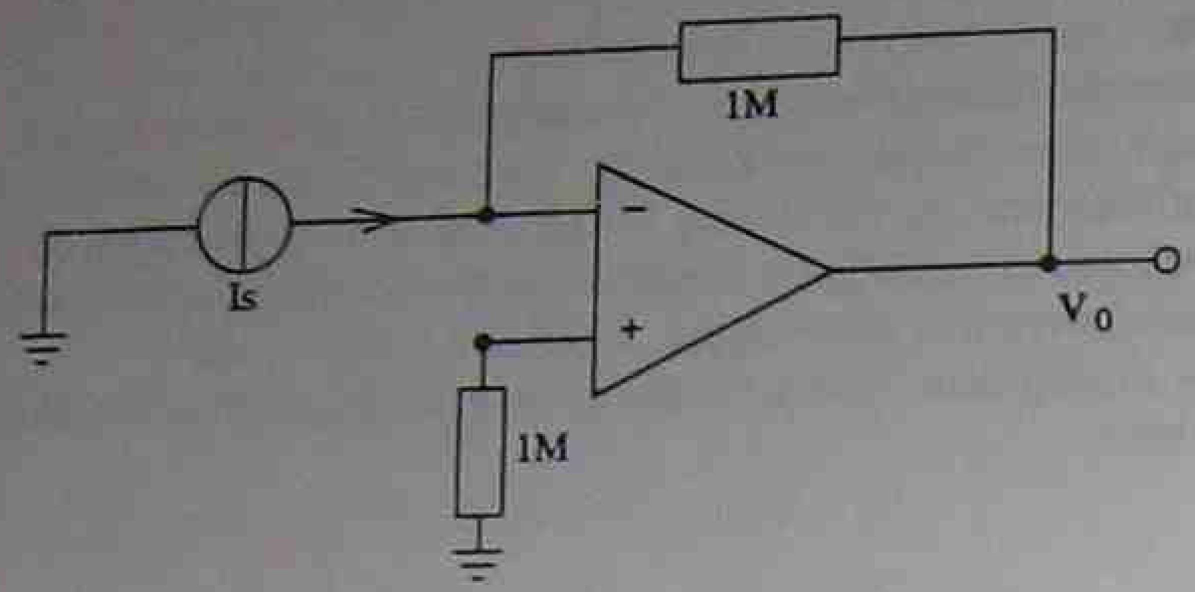
Concluding remarks

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.

Review questions

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.

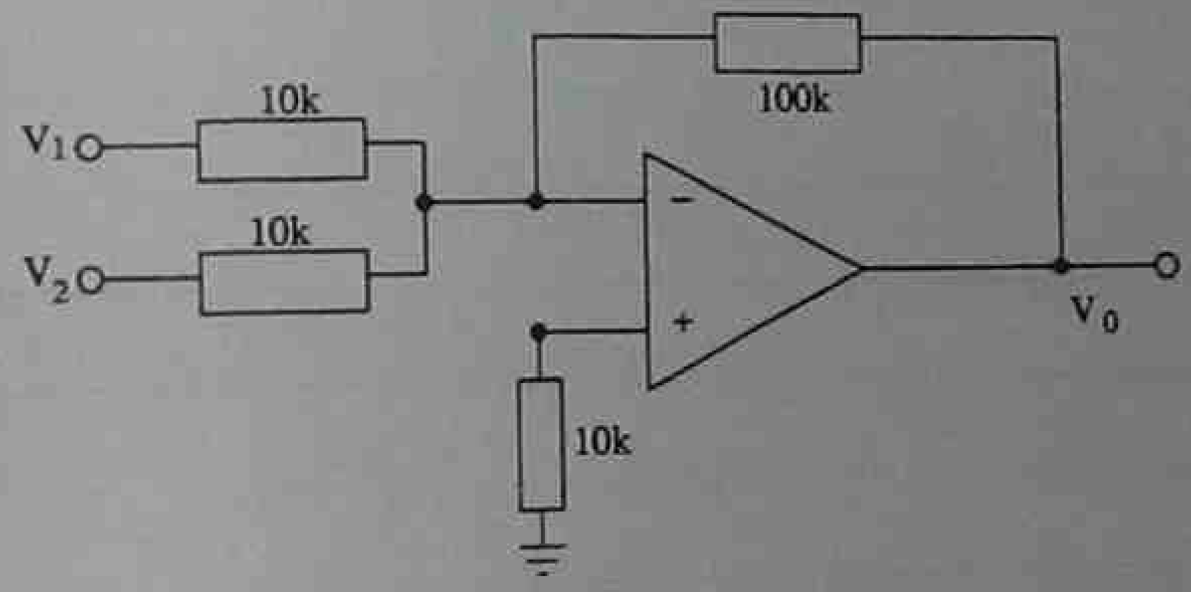
(b) the noise voltage gain.

(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

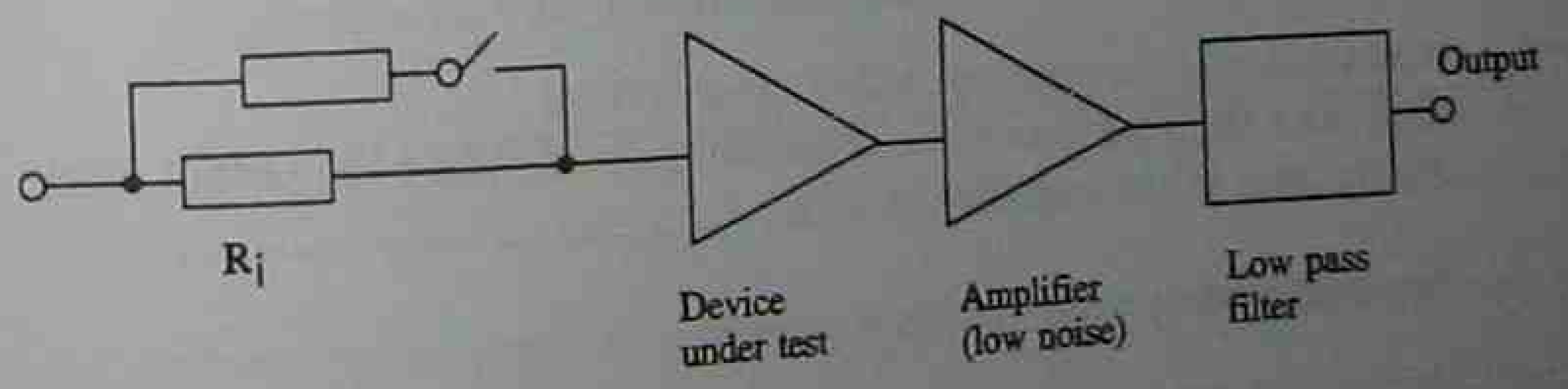
- To measure the input noise voltage of an op amp and observe its variation with source resistance and bandwidth
- To learn to measure noise rms voltage using a RMS voltmeter, a DVM and a CRO

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.

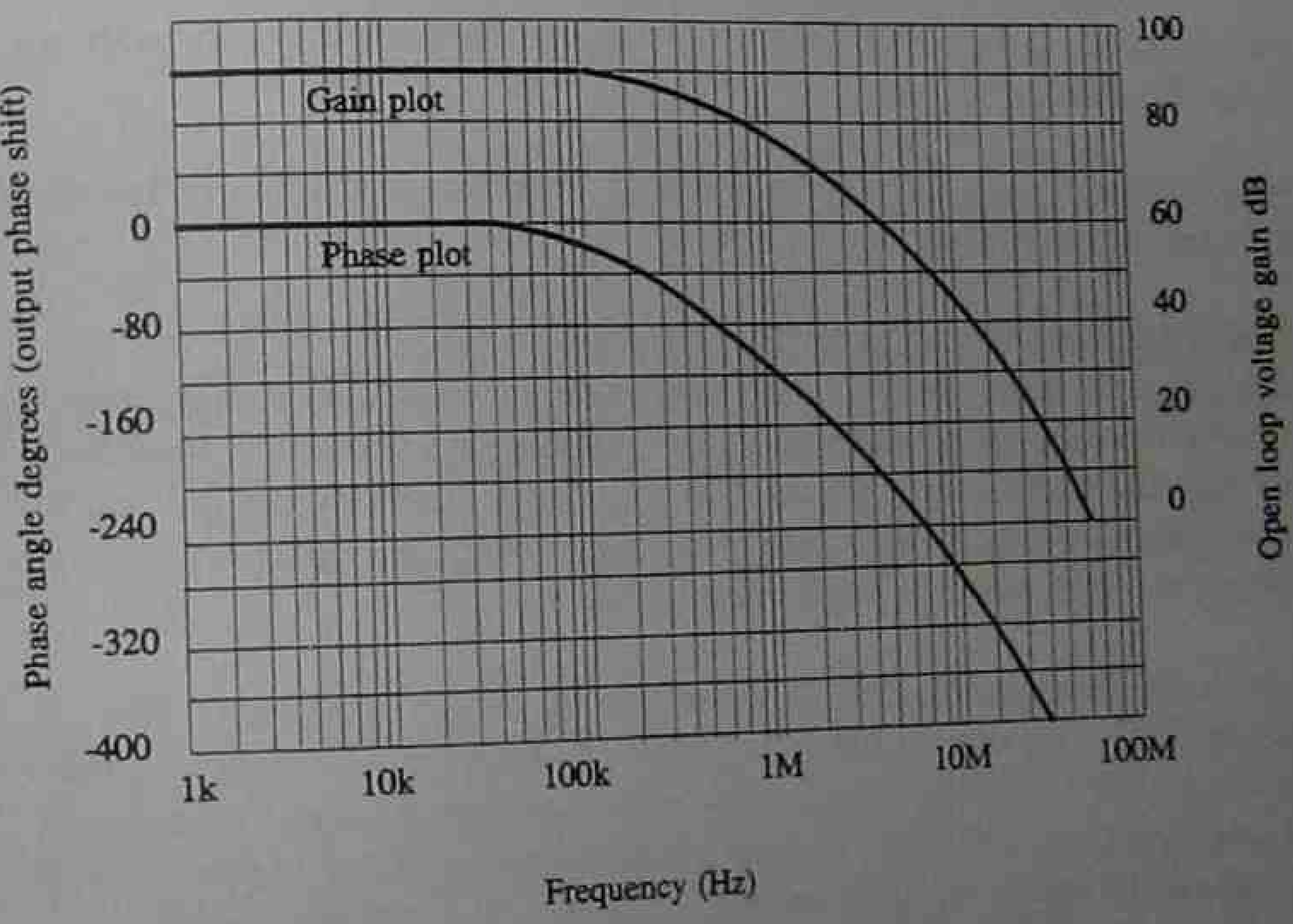


Fig. 1 Typical gain and phase response of an op amp

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.

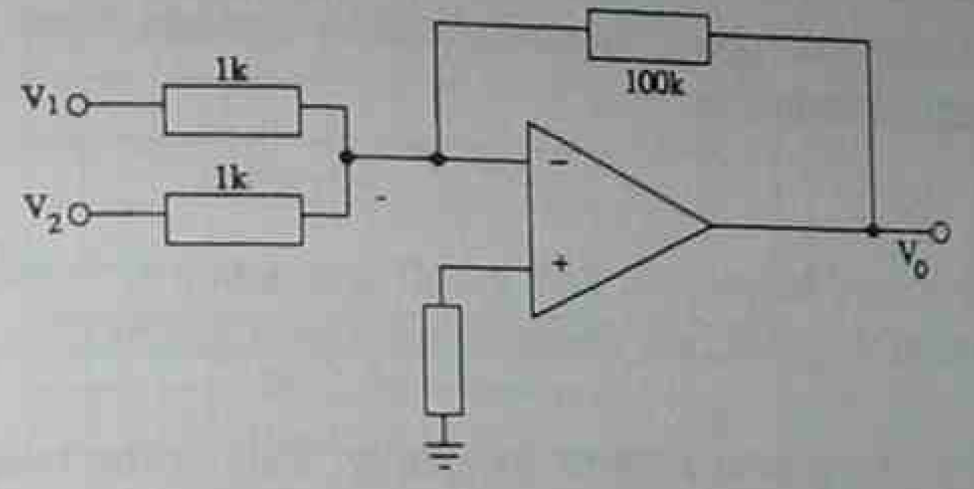
- Calculate $A_{CL} = 1 + R_F/R_1$. This applies to both inverting and non-inverting amplifiers (as with offset voltage and noise calculations). As usual, R_F and R_1 are effective values after simplifying series and parallel connections. Change A_{CL} to dB : A_{CL} (dB) = 20 * log(A_{CL}).

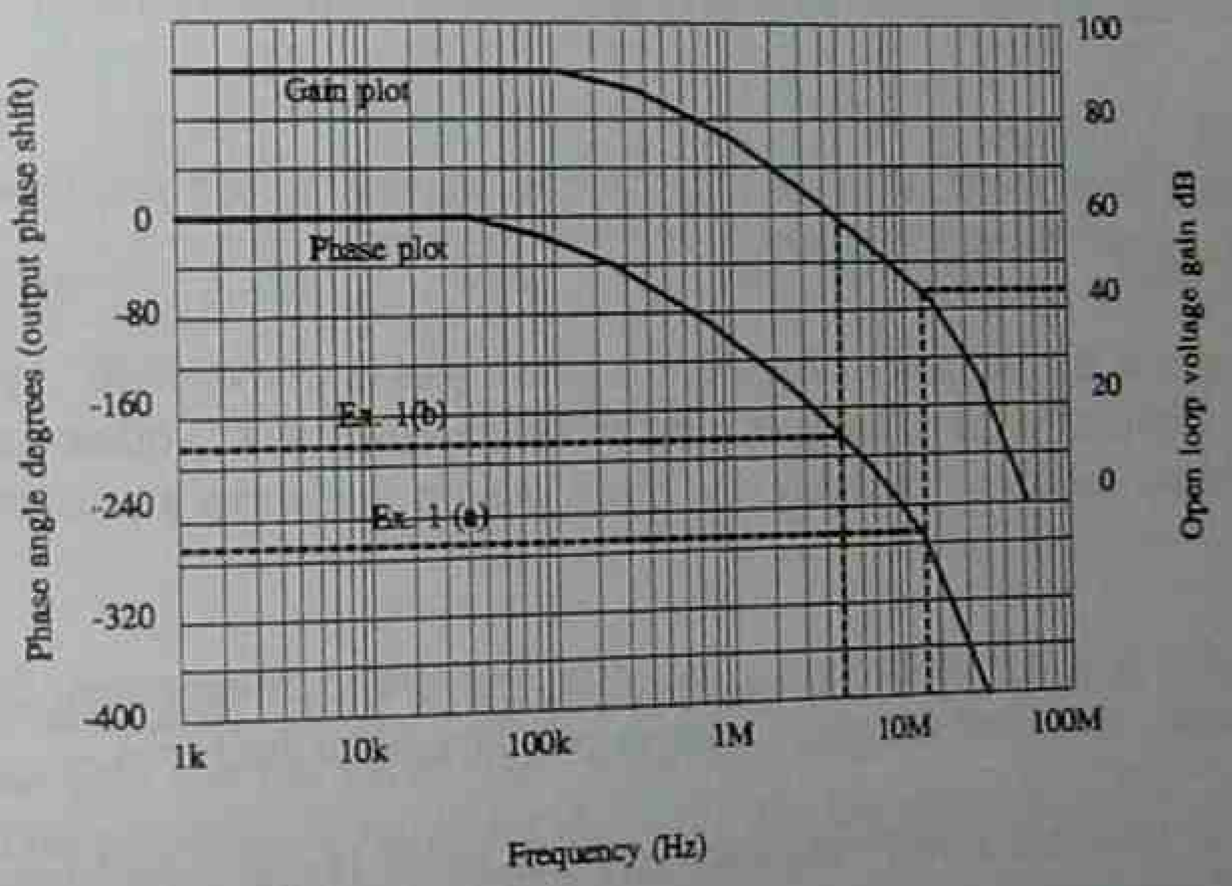
- On the open loop gain plot, draw a horizontal line at height A_{CL} (dB).
- Note the frequency at which the line cuts the open loop plot.
- Read the phase plot value at the relevant frequency.
- If the value is ≤ -180°, the amplifier is unstable. If it is >-180°, the amplifier is stable.

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) $R_1 = 1k \parallel 1k = 500\Omega$
$\therefore A_{CL} = 1 + 100k/500 = 200 = 46$ 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,
$\angle A_{OL} = -180°$ 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}.*

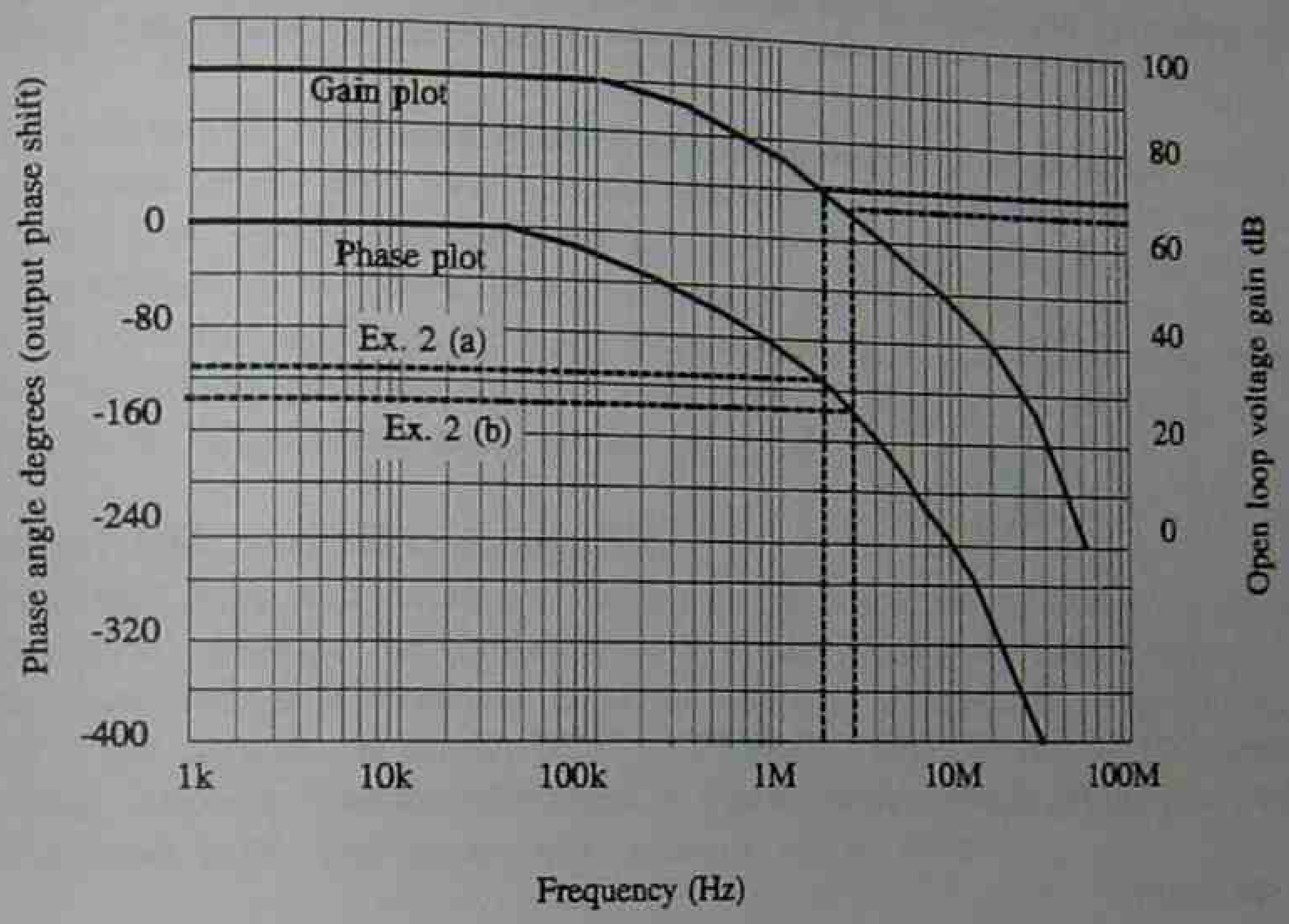
It is important to note that a NFB amplifier gets less stable if the closed loop gain is decreased, or if the open loop gain is increased. Since the smallest A_{CL} possible is 1, the amplifier has the greatest chance to be unstable as a voltage follower. If a NFB amplifier is stable as a voltage follower, 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.

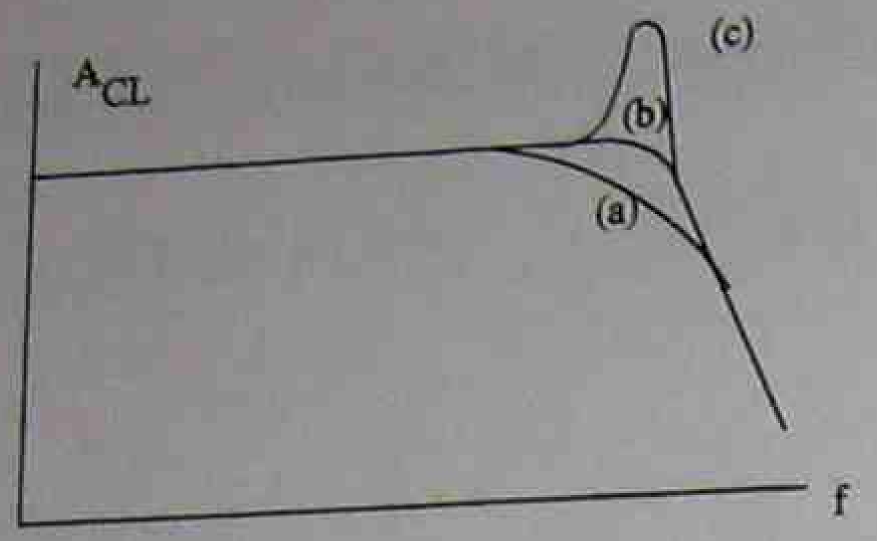


Fig. 2 Effect of phase margin on frequency response

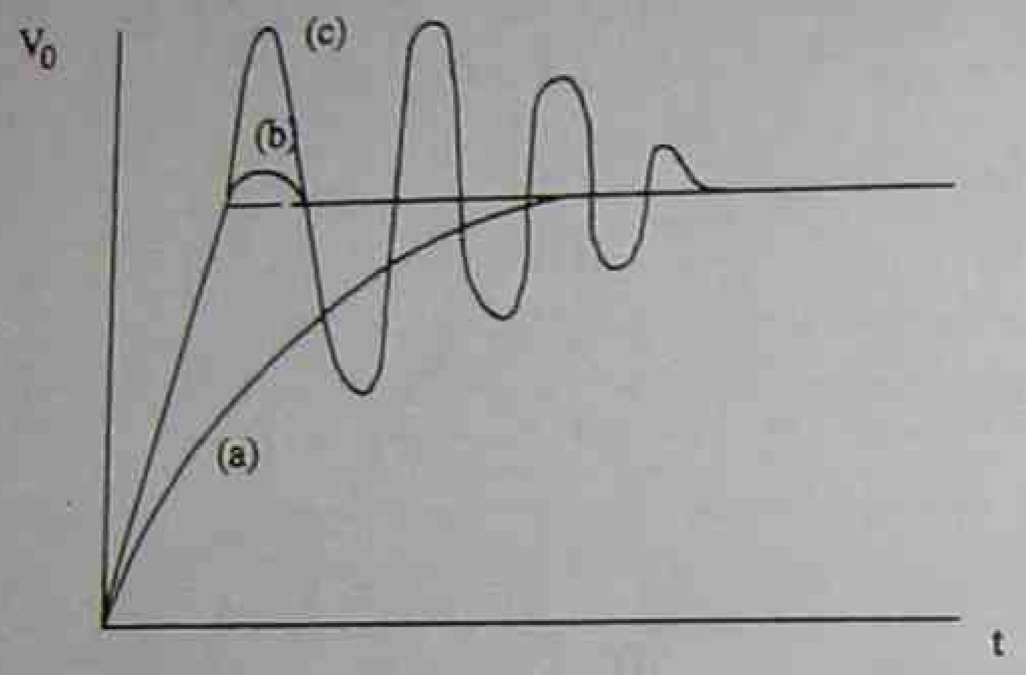


Fig. 3 Effect of phase margin on square wave response

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.

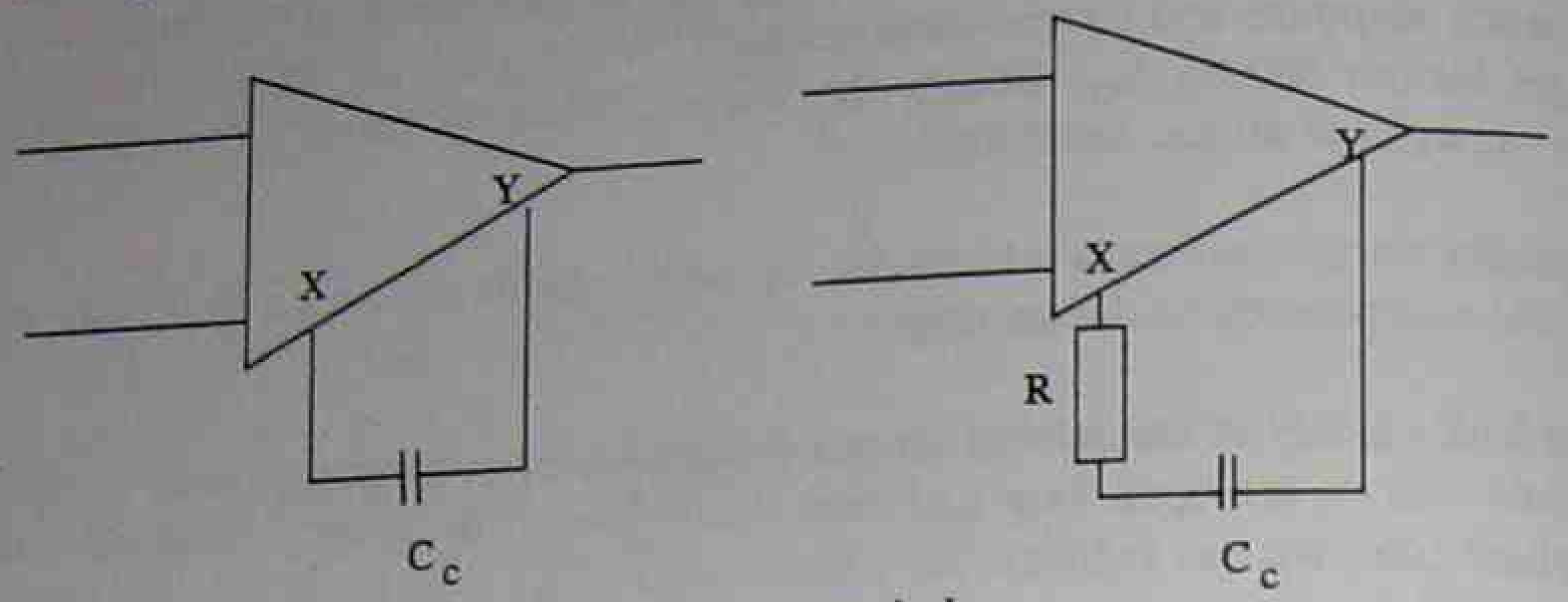
With externally compensated op amps, you have to connect your own filtering components, following the suggestions in the data sheets. For this purpose, the op amp provides compensation terminals in the package.

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 ?

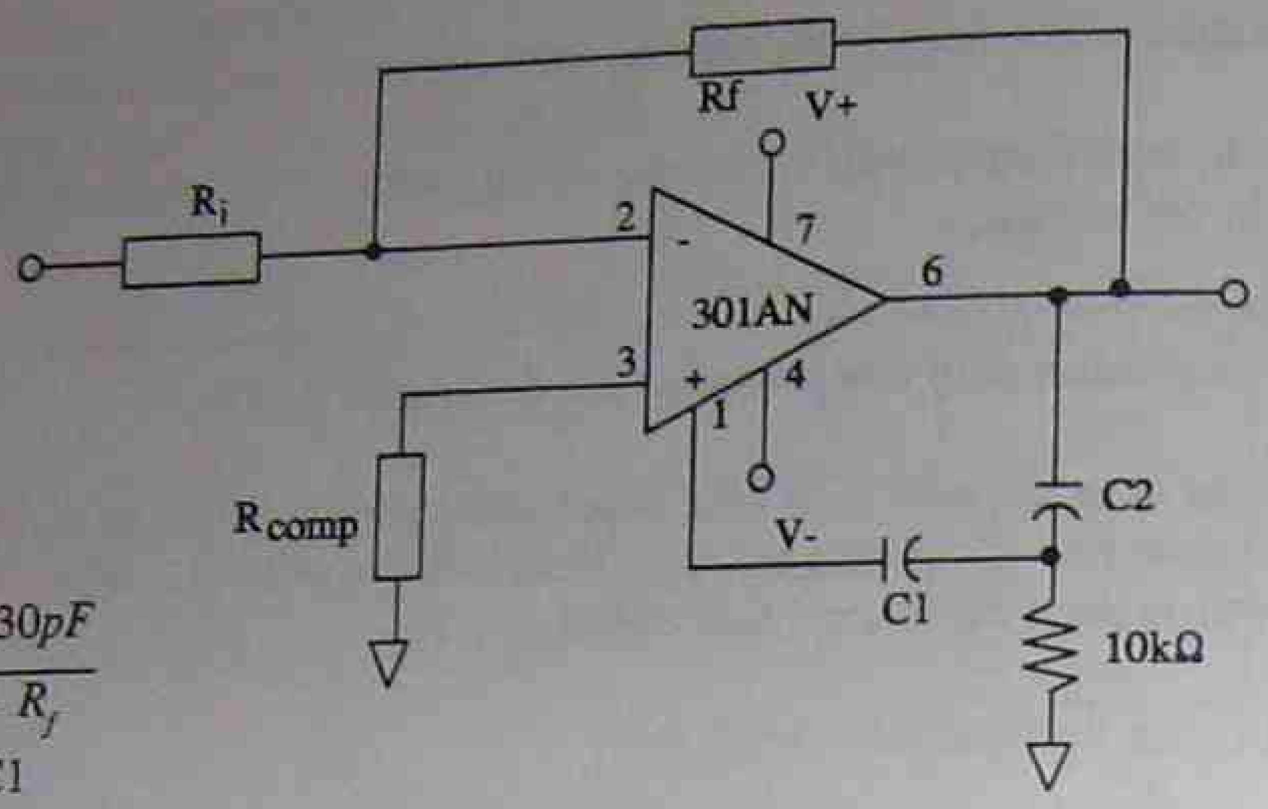
(b) If C_c of 10pF is actually used, what is the bandwidth of the circuit and slew rate ?

Solutions

(a) For $A_{CL} = 1$, $C_c = 30$ pF
$\therefore$ For $A_{CL} = 10$, $C_c \geq 30 \text{ pF} * 1/10 = 3$ 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



$$C1 \geq \frac{R_i \times 30pF}{R_i + R_f}$$

$$C2 = 10 \times C1$$

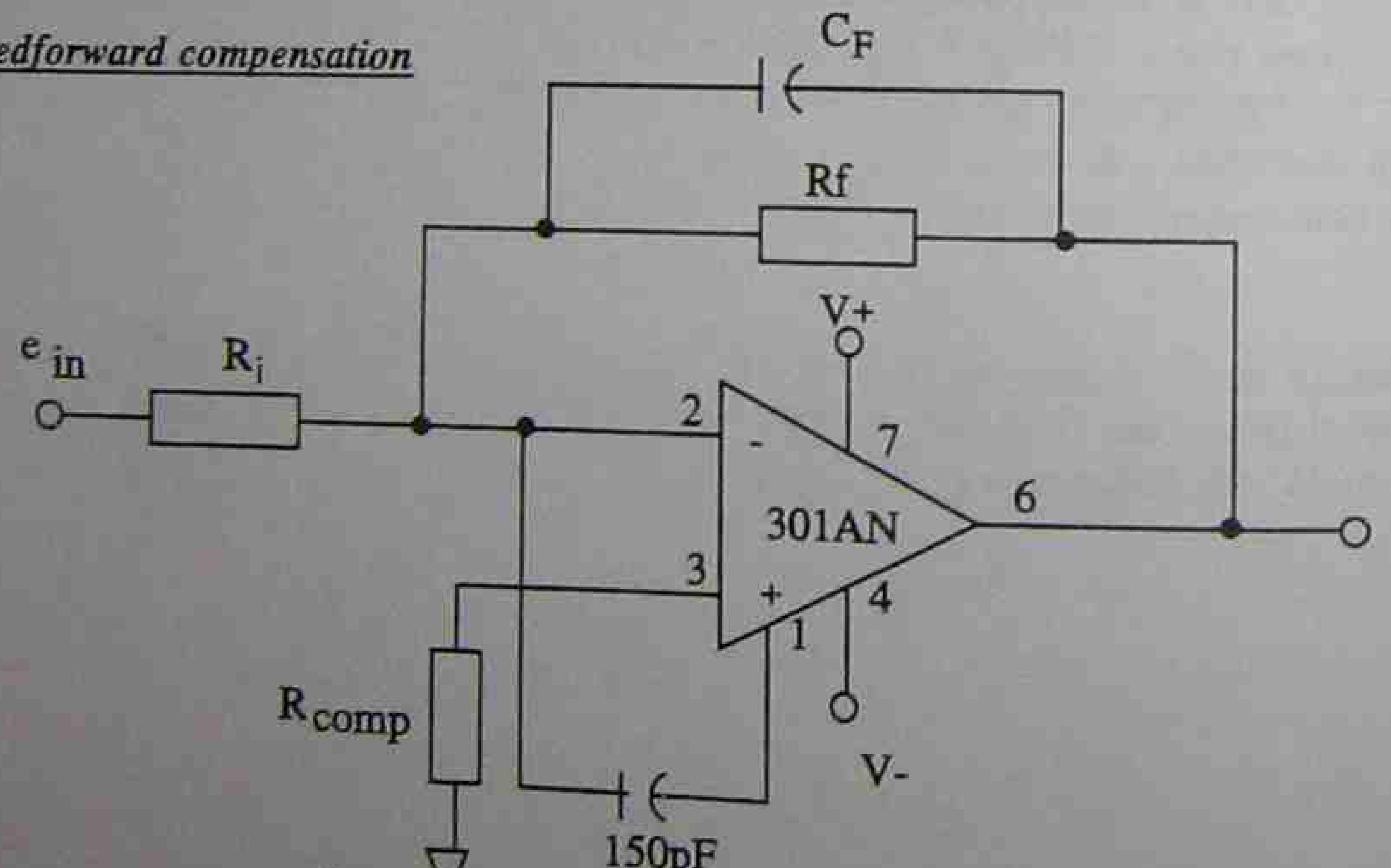
Fig.6 Two-pole compensation.

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.

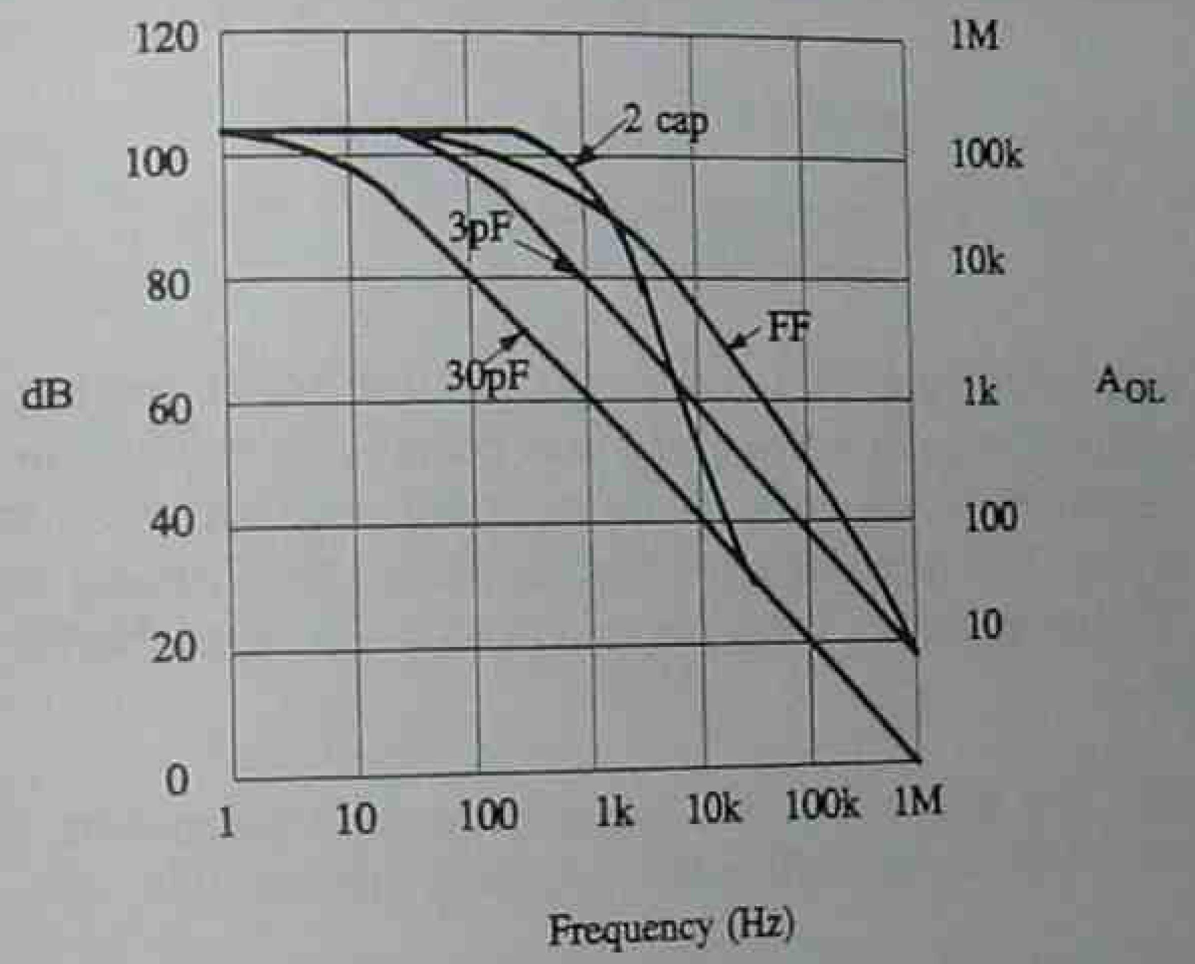


Fig. 8 Effect of external compensation on gain bandwidth.

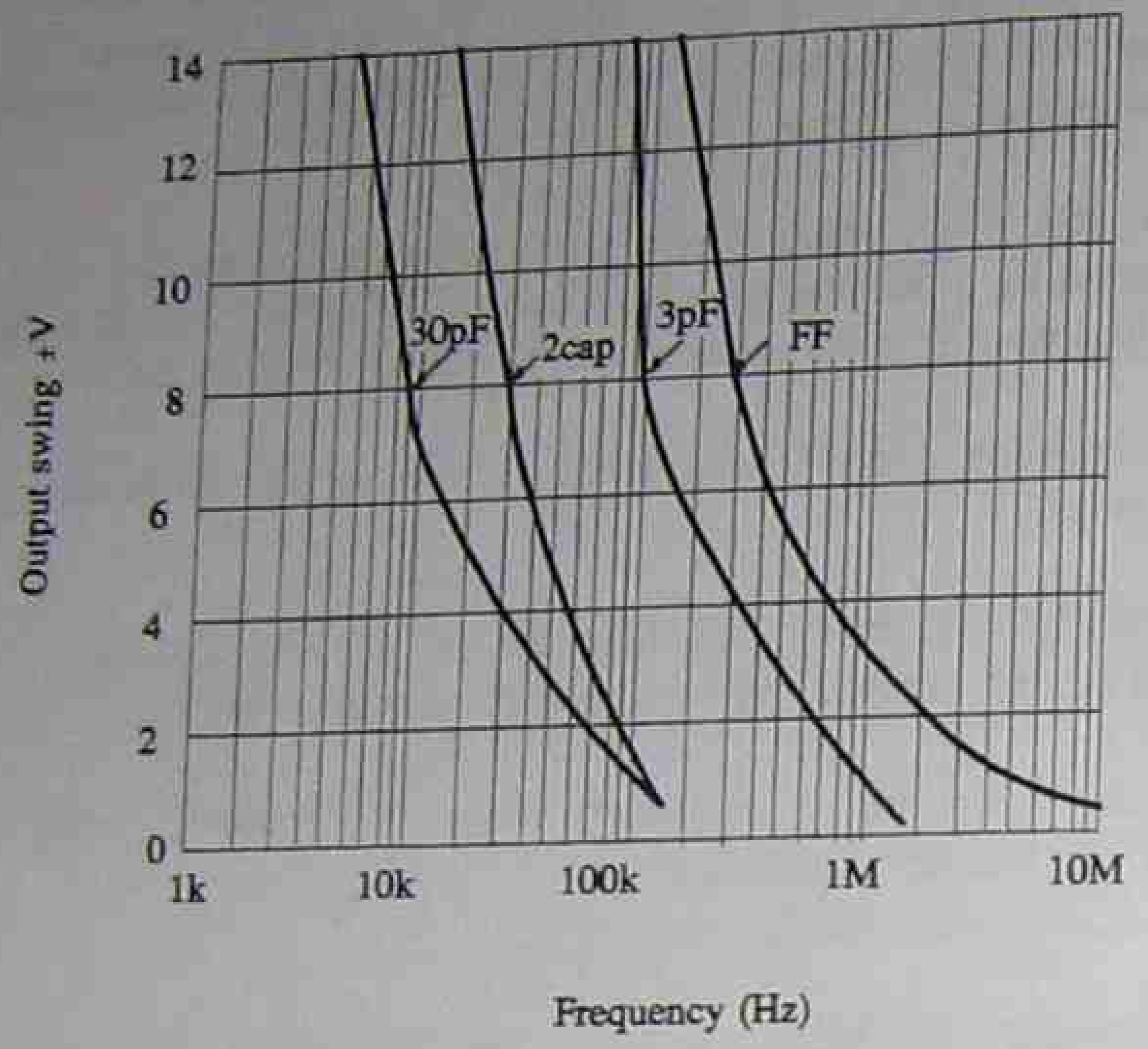


Fig. 9 Effect of external compensation on full power bandwidth

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.

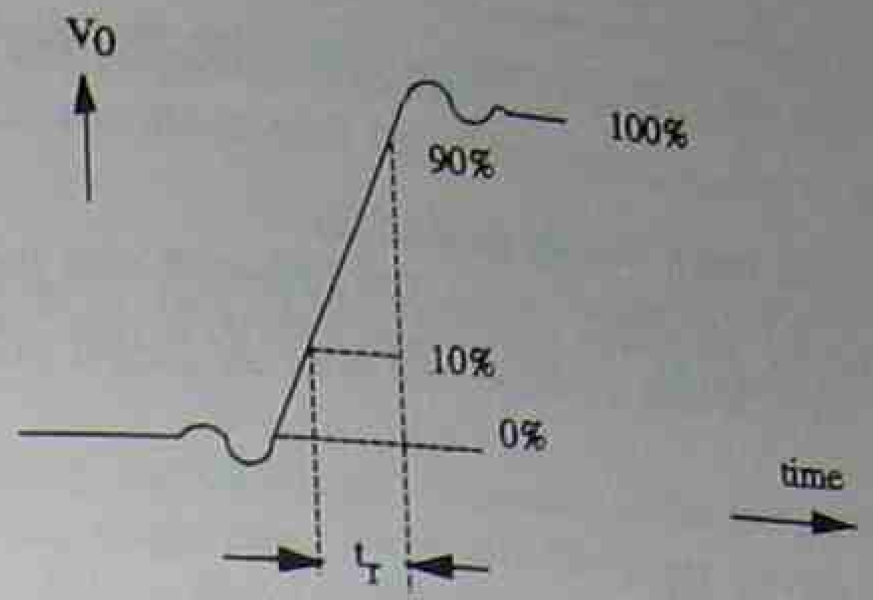


Fig. 10 Bandwidth measurement using square wave testing

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.

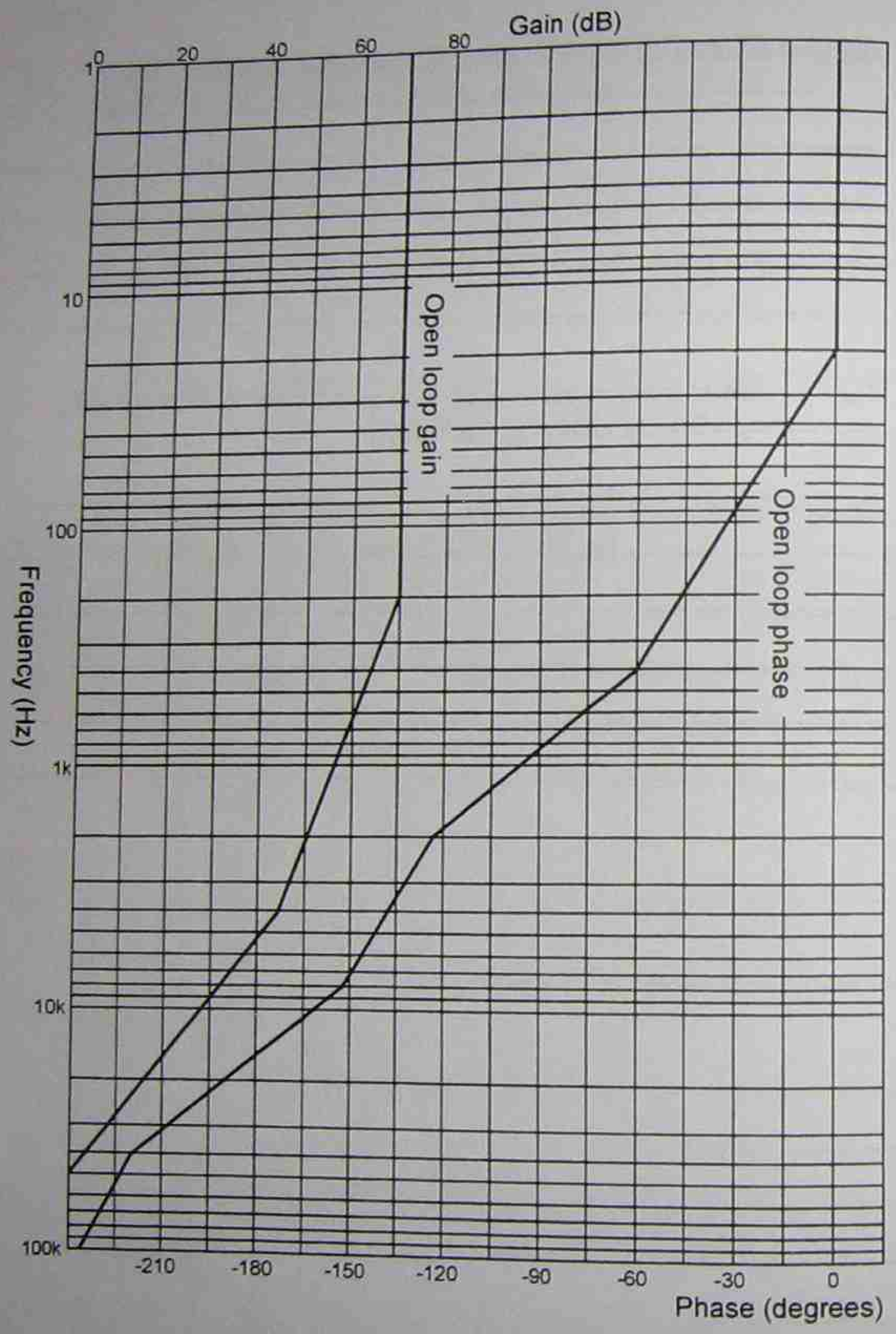
   (a) State whether the amplifier with NFB is stable for a closed loop gain of 0 dB. Outline the reasons for your conclusion.

   (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

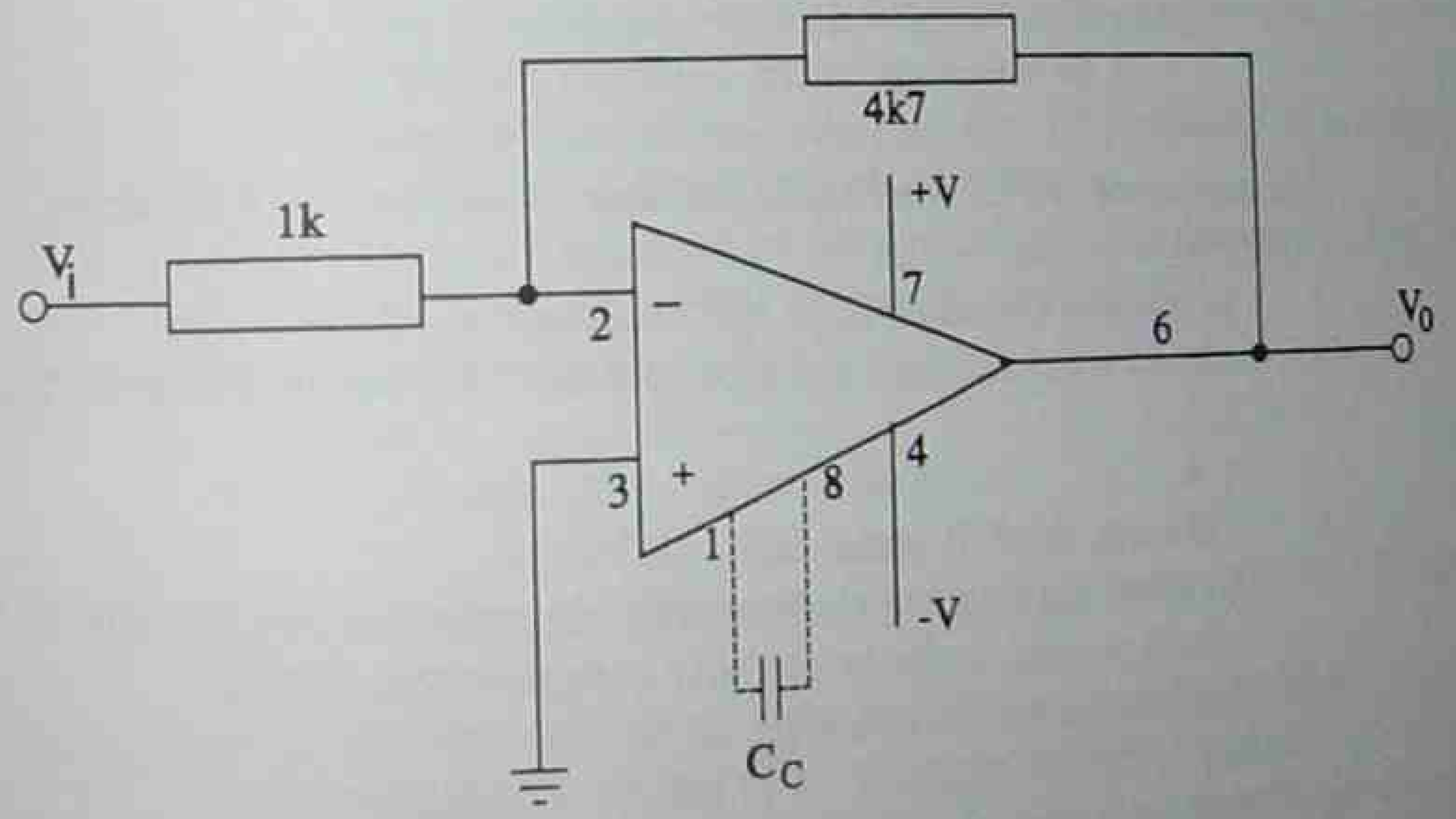
Equipment

- 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 = ΔV/Δt = ...

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 = ΔV/Δt = ...

Step 4 *Measurements of 3dB bandwidth and slew rate for C_c = 100 pF using square wave testing*

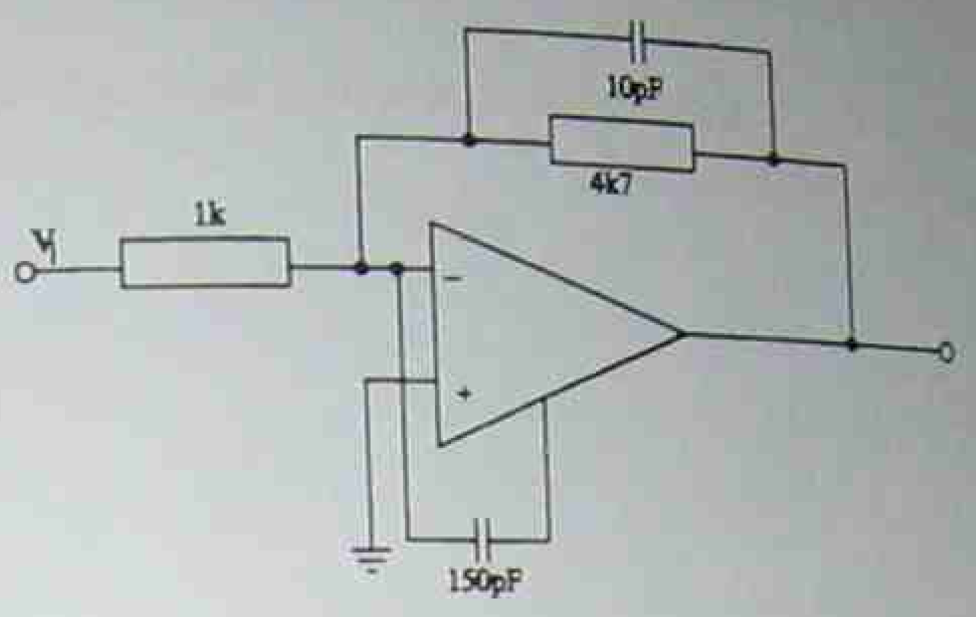
- In circuit A, change C_c to 100pF or closest available value.
- Measure the rise time as in Step 2.
  t_r =
- Calculate the 3dB bandwidth = $0.35/t_r$ = ...

---

and, in particular, whether the circuit is over, under or properly compensated.

- Measure slew rate as in Step 2.
  ΔV = ... Δt = ...
- Calculate slew rate = ΔV/Δt = ...

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 = ΔV/Δt = ...

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 = ΔV/Δt = ...

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.