Pearson BTEC Levels 4 Higher Nationals in Engineering (RQF)

Unit 19: Electrical and Electronic Principles

Unit Workbook 4

in a series of 4 for this unit

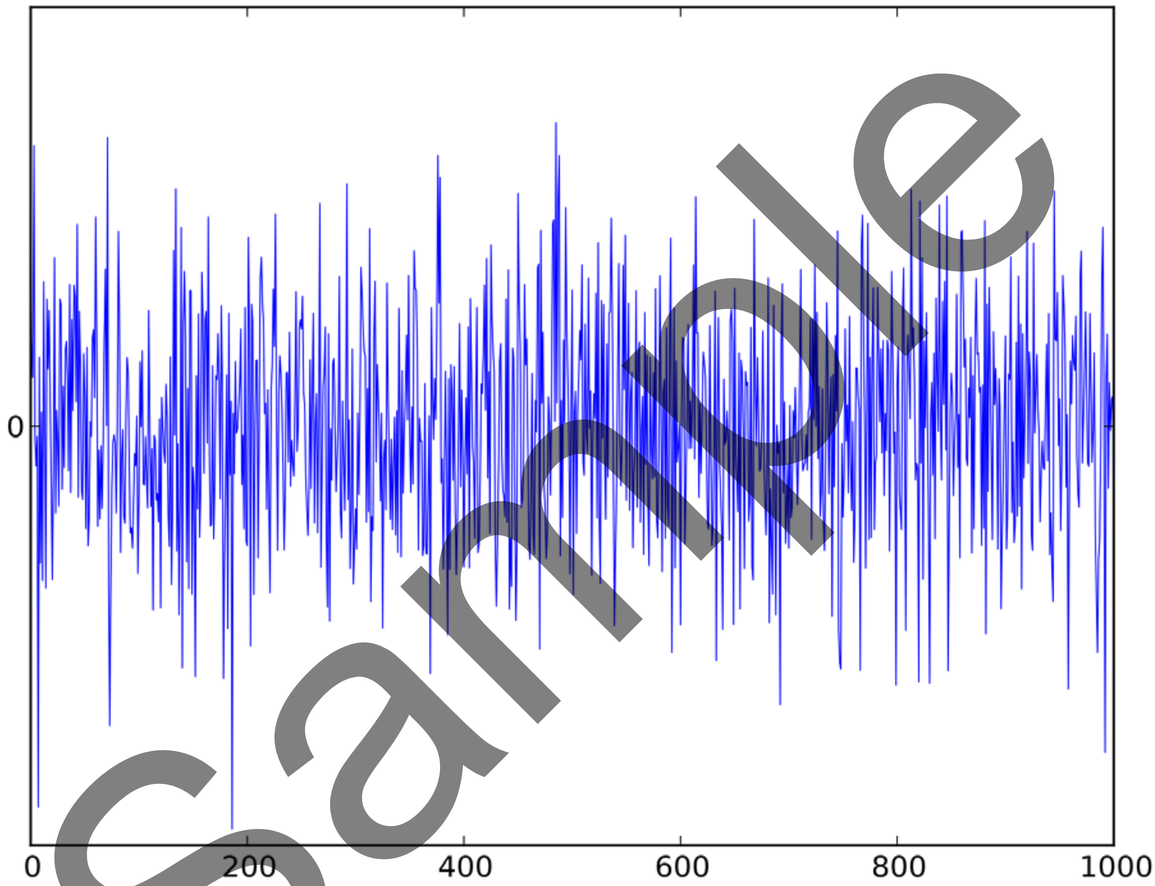
Learning Outcome 4

Digital & Analogue Electronics

Analogue concepts:

Analogue quantities

An analogue signal is one which has a continuous track through time. It can vary continuously both in amplitude and frequency. An audio signal is a very good example of an analogue signal…



Amplifier Characteristics

Ideal Characteristics

Amplifiers are used in the vast majority of electronic and fibre-optic systems. There are many types of amplifier, as you shall see later, but there are some ideal characteristics which are common to all varieties;

Gain

This is the amount that the amplifier boosts the input signal. The input signal is typically a voltage or a current, and the output is a larger (or smaller) version of this signal. The ideal case is that the gain can be any value we wish, without side effects.

Bandwidth

When an amplifier is used across the frequency spectrum its design, and parasitic capacitance, will dictate how well it produces the desired gain at all frequencies. When the gain drops to provide half the ideal power then this marks a half-power point, commonly referred to as a '-3 dB' point. There will be such a point at a lower and upper frequency. The difference between the upper and lower -3 dB points is referred to as the 'bandwidth' of the amplifier.

For many amplifiers the ideal bandwidth will be infinite i.e. it performs equally well at all frequencies. Some amplifiers may be designed to work only within at a select band of frequencies, so infinite bandwidth will not be ideal in those cases.

Input Impedance

Very often the input impedance is desired to be high, so as not to load preceding circuitry. Some amplifiers would ideally have infinite input impedance, whilst others may need a finite input impedance to match the output impedance of a preceding amplifier or device.

Output Impedance

The output impedance of an amplifier is typically desired to be low, or zero. This will allow the amplifier to drive high currents to a connected load, such as a loudspeaker or other type of transducer.

Noise

Noise in an amplifier may originate from the amplifier itself (its components or design), the power supply, nearby circuitry, long conductors acting as antennae, or even external sources such as radio/CB broadcasts.

The ideal amplifier will not generate noise and will be immune from noise originating from outside sources.

Thermal Drift

When components age their temperature-specific characteristics change. This can result in diminished performance and drift from normal operation in an amplifier. The ideal amplifier will not be susceptible to thermal drift of any kind.

Common Notation

A table of common notation used in amplifiers is shown below. This workbook will explain all new notation as it arises in various sections.

Notation	Description
A_V	Voltage Gain
A_i	Current Gain
A_p	Power Gain
Z_{in}	Input Impedance
Z_{out}	Output Impedance
Q Point	The quiescent operating point
CE	Common Emitter (bipolar amp.)
CB	Common Base (bipolar amp.)
CS	Common Source (FET amp.)
h_x or h_{xx}	h-parameters used for amplifier analysis

DC and AC Behaviour

Bipolar transistor amplifiers (using npn or pnp transistors) tend to be biased with DC voltages in order to provide a steady operating point (Q-point) which is in the linear region of the transistor's characteristic performance. When an AC input is applied to such an amplifier then the DC conditions will preset the amount of AC gain produced. Bipolar transistors are current-controlled devices but the DC bias voltages set the conditions for these currents. Bipolar transistors will have a specific output current (collector to emitter) for any given input (base) current.

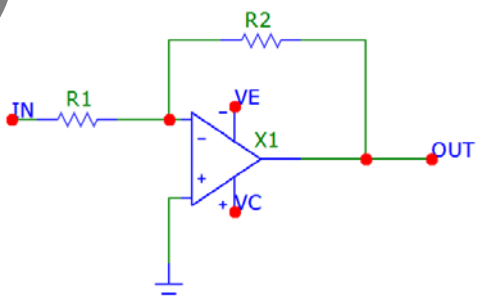
For field-effect transistors the situation is somewhat different. These devices are voltage-controlled and the current through the output terminals (source-drain) will be influenced by the voltage on the controlling input terminal (gate).

For any type of amplifier the DC conditions set the limiting parameters for the AC signal swing. Many modern systems use operational amplifiers to provide DC gain or AC gain.

OpAmp Basic Circuits

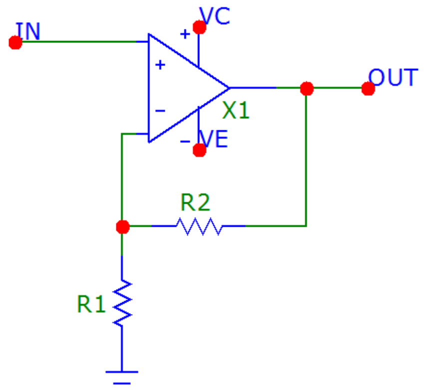
The two basic operational amplifier (OpAmp) circuits are the inverting and non-inverting types.

Inverting OpAmp



The circuit above is an example of an *inverting amplifier*. It takes this name because the input signal is presented to the inverting terminal of the OpAmp (via R1). When we analyse OpAmps we come to *assume that the voltage between the two input terminals of the OpAmp is zero*. This being so, we see that the RHS of R1 is *virtually* connected to ground. In that case it is clear that the input impedance of this inverting amplifier is R1 itself. The output signal is an inverted version of the input signal.

Non-Inverting OpAmp



Clearly the input signal is connected to the non-inverting '+' terminal of the OpAmp. In this configuration the ideal OpAmp has infinite input impedance. The output signal is a non-inverted version of the input signal.

Limitations

All amplifiers have their limitations. The DC supply voltage is a definite upper limit on the amount of AC voltage at the output. When power, and an input signal, is applied to an amplifier then the transistors will be conducting. No transistor can conduct without developing a voltage across any of its terminals. An AC amplifier which is supplied by a +12 V DC source will never be able to supply 12 V AC at the output because the output transistor(s) will consume a small proportion of that supply voltage.

If the gain of the amplifier is set too high (by the DC conditions) and a large input signal is applied then the amplifier will 'clip' the signal - it will look distorted.

All amplifiers will have power limitations. These limits are imposed by the selection of components at design time. It is no good selecting a 3 Watt transistor if it is desired that it should dissipate 10 Watts of power. Such a transistor may well burn out.

Common Applications

Amplifiers can be used in many applications. They can be constructed with bipolar transistors, field-effect transistors, thermionic valves, operational amplifiers or optical components. There are far too many applications to list here. However, the list below indicates some notable uses for amplifiers...

- Audio amplifier
- Power amplifier
- DC amplifier
- Current amplifier
- Instrumentation amplifier
- Buffer
- Filter

- Video amplifier
- Communications amplifier
- RF amplifier
- Line driver

Internal Circuitry of 741

The internal circuitry of a 741 OpAmp is copyrighted material. However, a good description of this chip may be found here.

Analysis of Operation and Performance

Use of Quantitative Methods

In order to analyse the performance of single-stage transistor amplifiers it is usual practice to enlist the aid of a set of tools known as h-parameters. These are presented in the next section.

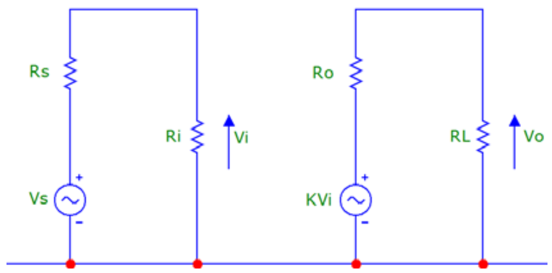
For operational amplifiers we do not normally use h-parameters, preference being given to direct circuit analysis of voltages and currents.

Equivalent Circuits

General Amplifier

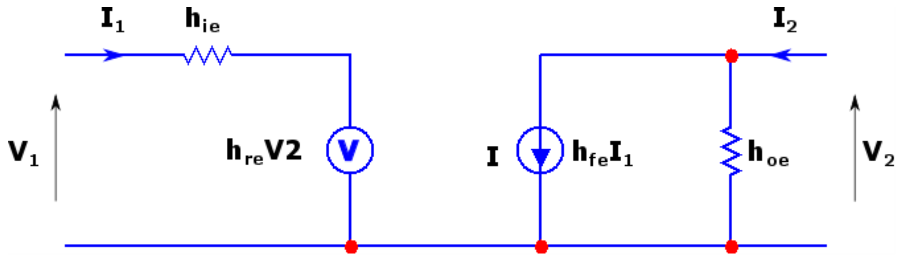
The following circuit is a general model for any amplifier. The portion on the left contains the input signal (Vs) and its internal resistance Rs. This then connects to the actual input resistance of the amplifier (Ri). A voltage of Vi is developed across the internal resistance.

To the right of the diagram is the output stage of the equivalent circuit. The voltage gain of the amplifier is denoted by K here. The voltage gain times the voltage developed across the input resistance will give the source voltage (KVs) in the output section. This drive current through the output resistance of the amplifier (Ro) and also through the load resistor (RL). A voltage of Vo will be developed across the load resistor.



Bipolar Junction Transistor Amplifier (BJT Amplifier)

To model CE bipolar junction transistor amplifiers we employ a hybrid-pi model. The parameters used within this model are commonly referred to as 'h-parameters'. This general model is given below.



Each of the terms in this hybrid-pi model are explained as follows...

Term used	Explanation
V_1	Input voltage
I_1	Input current
V_2	Output voltage
I_2	Output current
h_{ie}	h-parameter for **input resistance**. Given by V_1/I_1 when $V_2 = 0$ (i.e. **when the output is shorted**)
h_{re}	h-parameter for the **reverse voltage gain** V_1/V_2 **with the input open-circuited** (i.e. when $I_1 = 0$)
h_{fe}	h-parameter for the **forward current gain** I_2/I_1 with the **output short-circuited** (i.e. when $V_2 = 0$)
h_{oe}	h-parameter for the **output conductance** I_2/V_2 when the **input is open-circuited** (i.e. when $I_1 = 0$)

From this hybrid-pi model we may deduce the following equations...

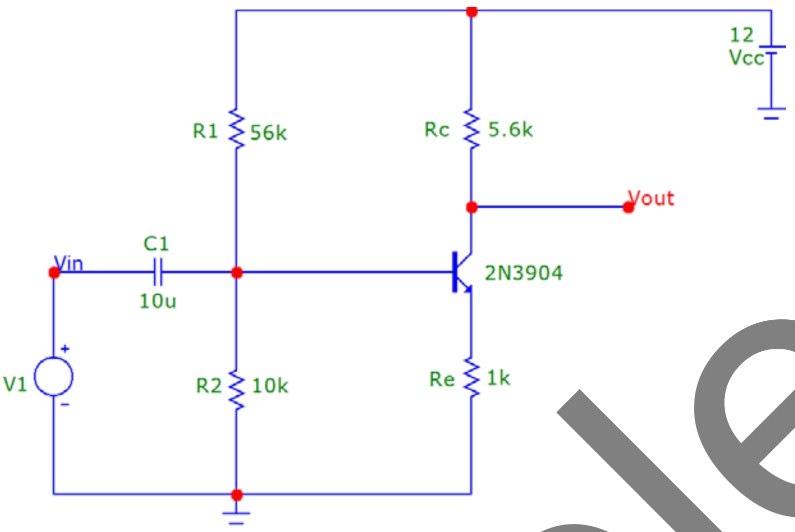
$$\boldsymbol{V_1 = h_{ie}I_1 + h_{re}V_2}$$

$$\boldsymbol{I_2 = h_{fe}I_1 + h_{oe}V_2}$$

We shall next build and analyse a CE amplifier. The two equations above will be used to confirm its operation.

Computer Modelling

The circuit diagram below shows a 2N3904 general purpose NPN transistor connected in common emitter (CE) mode.



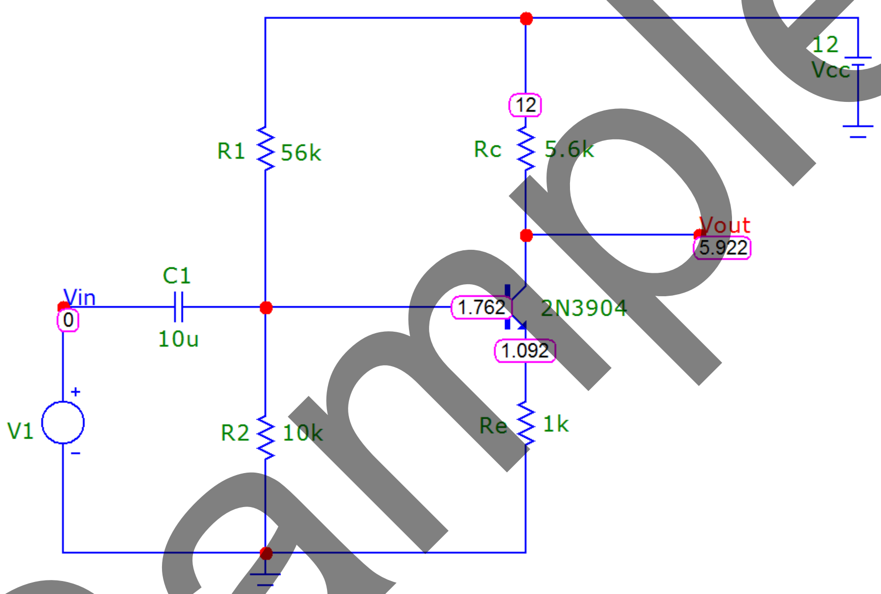
This circuit was constructed using the MicroCap circuit simulator and is available to download on Moodle. The design and operation of the circuit is as follows...

- The transistor needs to be DC biased so that the collector voltage (Vout) sits at half of the supply voltage. This will allow for an equal swing of the amplified signal at the output. Since the supply voltage is 12 V then we would like Vout = 6 V DC.
- We would like a collector quiescent current of 1 mA. Since we have 6 V across the Rc then its value will be 6/1mA = 6 kΩ. We must choose a resistor value which is commercially available, so select 5.6 kΩ for Rc, as shown.
- A rule-of-thumb for the design of CE transistor amplifiers is to let the DC voltage across the emitter resistor be about 8% to 10% of the supply voltage. If we choose 1 V to develop across Re, and we assume that the current through Re is roughly the same value as the collector current (1 mA), then the value of Re is 1V/1mA = 1 kΩ, as shown.
- The next step is to consult the datasheet for our 2N3904 transistor and refer to figure 11 which indicates a current gain of around 150 when the collector current is 1 mA. This indicates that we will have a steady-state (quiescent) base current of 1mA/150 = 6.7 µA.
- We need to provide base current to the transistor. This is usually done by constructing a potential divider of resistors (R1 and R2). Another rule-of-thumb is to let the current through R2 be around 25 times the base current i.e. 25 X 6.7 µA = 167.5 µA. Since the voltage across R2 is equal to the voltage across Re (1 V) plus the base-emitter voltage (normally around 0.7 V for silicon) then we have a total of 1 + 0.7 = 1.7 V across R2. The value of R2 is then 1.7/167.5µA = 10.15 kΩ. Our nearest preferred (commercially available) resistor value is therefore 10 kΩ, as shown.
- The next resistor to determine is R1. Since we have a 12 V supply and 1.7 V of this appears across R2 then we must have 12 – 1.7 = 10.3 V across R1. The resistor R1 provided current to the base of the transistor AND R2, so the current flowing through R1 must be the sum of the base current and the current through R2 i.e. 167.5µA + 6.7µA = 174.2µA. The value of R1 must therefore be 10.3V/174.2µA = 59.1 kΩ. Our best preferred value is 56 kΩ, as shown.

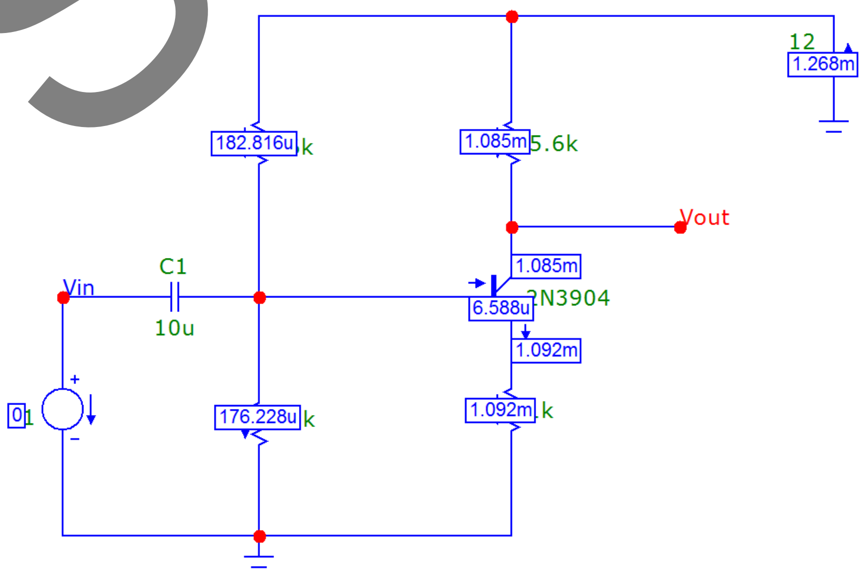
- When a sinusoidal signal (V1) is presented to the amplifier then we do not wish this to upset the careful DC conditions which we have set up. For this reason we need a capacitor (C1) to block the DC from the AC in that region of the circuit. We shall be injecting a 5 kHz signal into this amplifier, so choose a capacitor with a low reactance at 5 kHz. Capacitors of 10 μF are commonly available, and if we chose this value then the reactance at 5 kHz would be $1/2\pi fC = 3.18\ \Omega$. This is a very low reactance and will have a negligible effect on the operation of the circuit. We therefore include a 10 μF DC blocking capacitor, as shown.

Transistor amplifier design is not an exact science. When transistors are manufactured their parameters can vary quite significantly from one to the other, even in the same production batch. The procedure and rules-of-thumb presented above are quite a usual tactic to produce functional CE amplifiers.

Within the MicroCap simulator, by clicking on 'Analysis' then 'Dynamic DC' the DC voltages around the circuit are shown, as follows...



These are in fairly close agreement with our calculations. Changing the settings to current-only measurement gives the following...



edexcel
Approved Centre

These current readings are also in pretty close agreement with our calculations.

Curves representing the four h-parameters for the 2N3904 transistor are given in figures 11, 12, 13 and 14 of the datasheet. The table below gives interpolated values for a collector current of around 1 mA...

Datasheet term	Value, given $I_C = 1\ mA$
h_{fe}	150
h_{oe}	9×10^{-6} [mhos]
h_{ie}	3.5 kΩ
h_{re}	1.3×10^{-4}

We may now use our model equations for this design...

$$V_1 = h_{ie}I_1 + h_{re}V_2 \qquad [1]$$

$$I_2 = h_{fe}I_1 + h_{oe}V_2 \qquad [2]$$

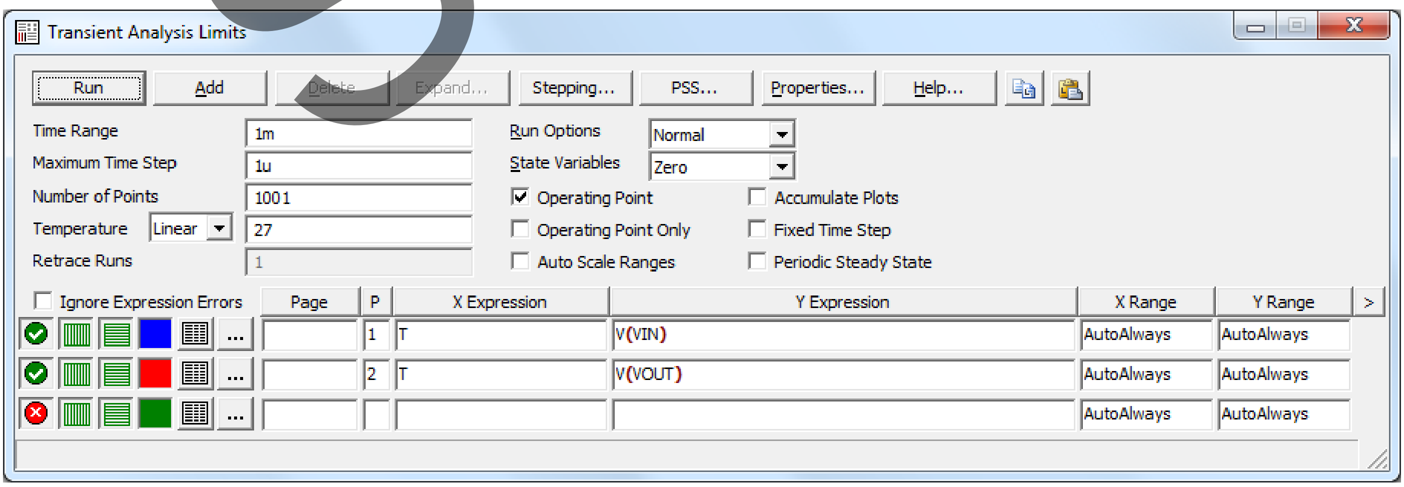
Since we know that I_2 is the collector current then this is 1 mA. Also, V_2 represents the quiescent output voltage, which is 6 V. Let's now transpose equation [2] to find the value for the hybrid-pi model value for I_1, which is the base current...

$$I_1 = \frac{I_2 - h_{oe}V_2}{h_{fe}} = \frac{0.001 - (9 \times 10^{-6})(6)}{150} = 6.3\ \mu A$$

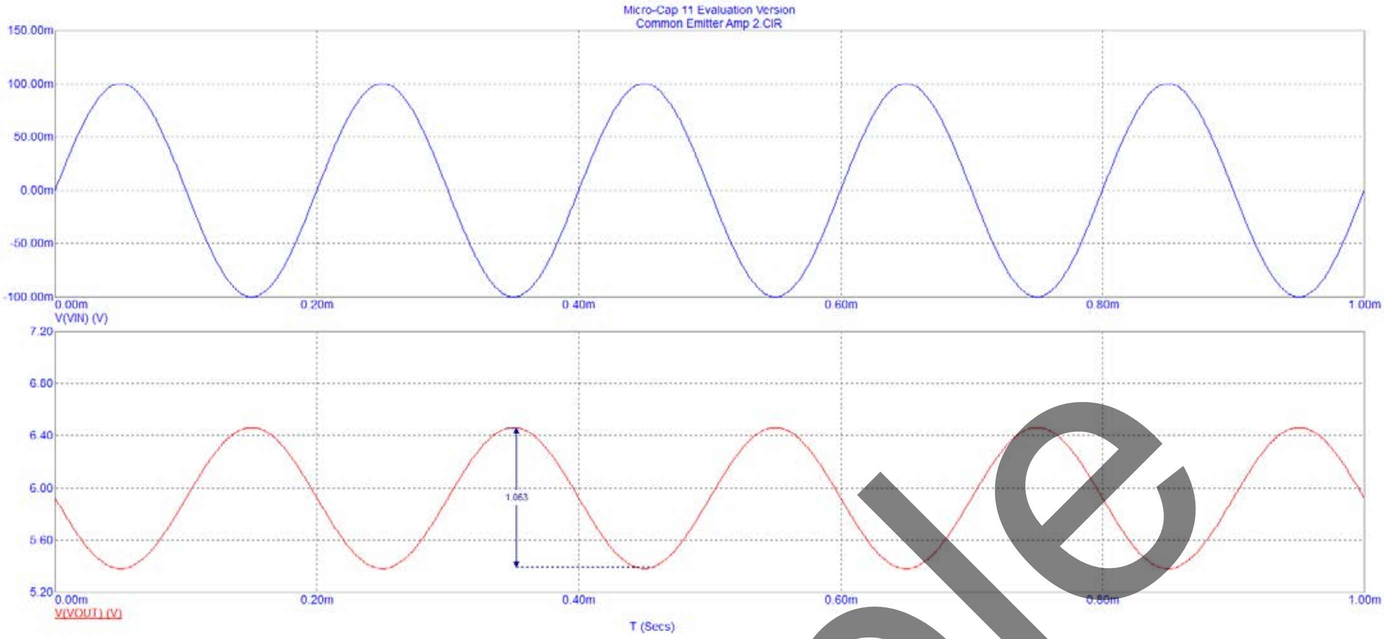
That's quite close to what the simulator gives (6.588 μA).

Voltage Gain

The CE amplifier under consideration has a sinewave source signal (Vin) of 0.1 V peak and a frequency of 5 kHz. To simulate the performance of this circuit in terms of voltage gain it is necessary to perform a 'Transient Analysis' in MicroCap. This analysis has time along the horizontal axis and the magnitude of our selected parameters on the vertical axis. To do this click on 'Analysis' then 'Transient'. A dialogue box appears and should be filled in as follows...



Click ‘Run’ and the following results show...



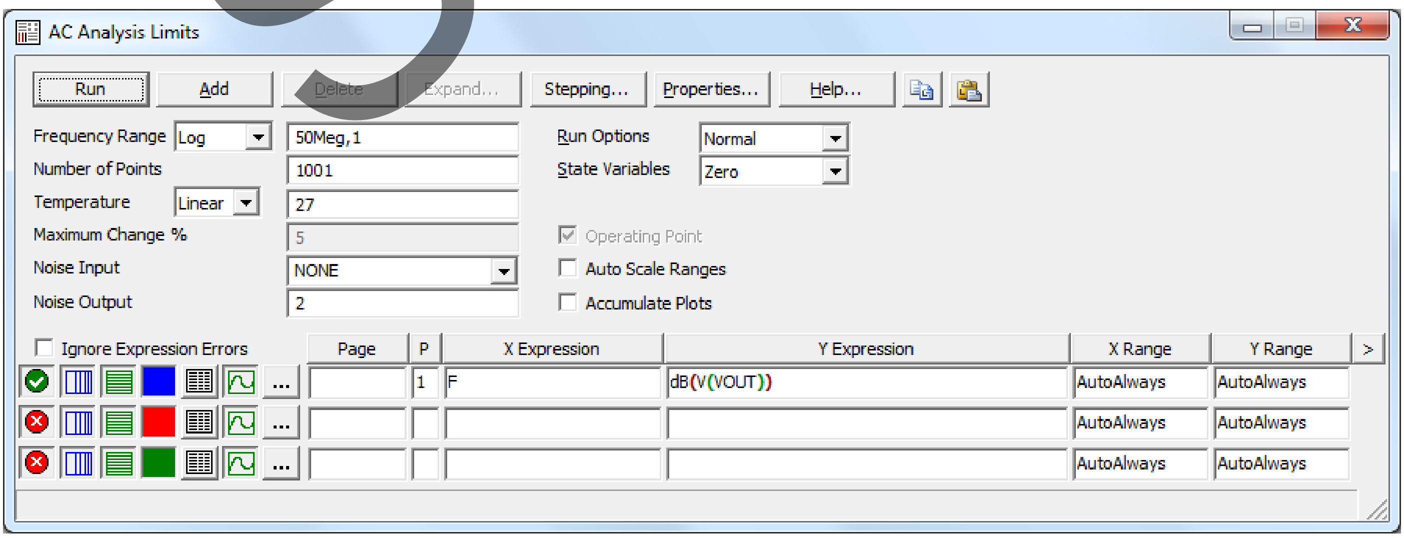
The input voltage here is shown in blue. This is 200 mV peak-to-peak (which is the same as 100 mV peak). The red plot is the output. Its marker indicates a peak-to-peak output voltage of around 1.0 V. We may therefore determine the voltage gain (Av) for this amplifier as...

$$A_V = \frac{1.0}{0.2} = 5$$

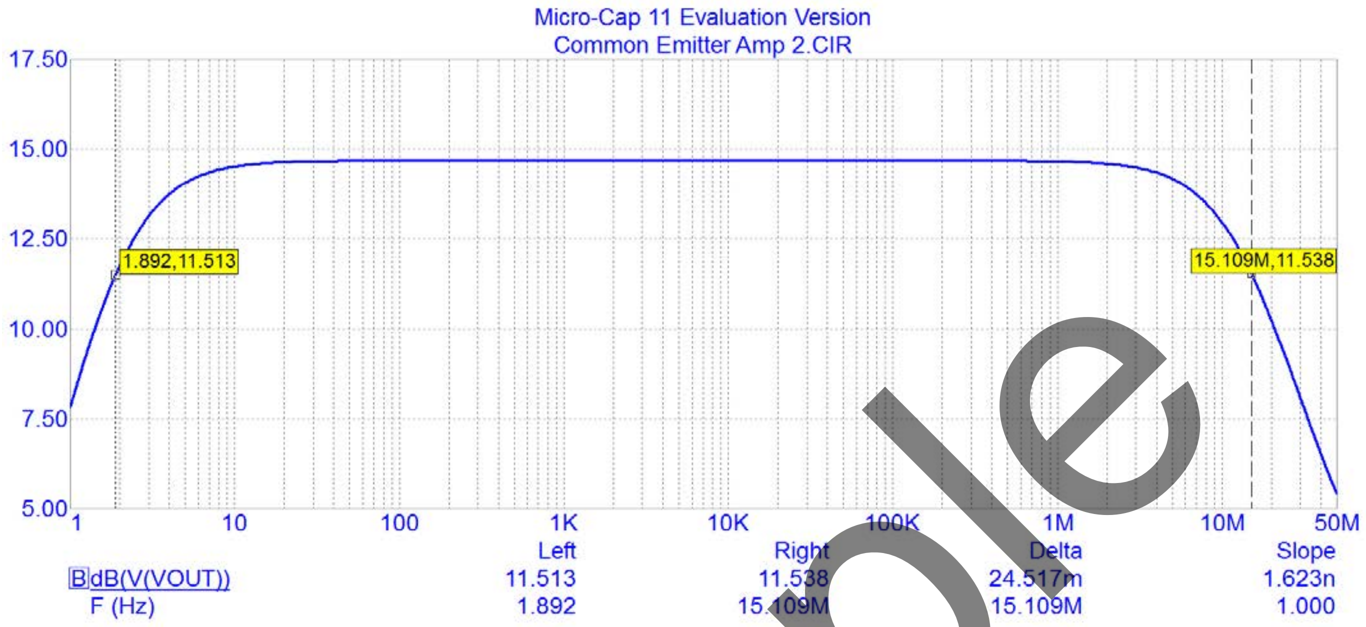
A rule-of-thumb for such amplifiers is that the voltage gain is roughly determined by the ratio of the collector resistor and the emitter resistor i.e. 5.6k/1k = 5.6 (reasonable agreement with simulation).

Frequency Response and Bandwidth

To perform a simulated AC analysis for our CE amplifier press ‘Analysis’ then ‘Ac...’. A setup dialogue box appears. This should be set as follows...



The frequency range is set from 1 Hz to 50 MHz, as shown. We are asking for the output voltage to be shown in decibels, which will allow us to find the -3 dB points and thus find the bandwidth for the amplifier. Press 'Run' to see the response...



The flat portion of the response is the normal working frequency range for this amplifier. As the frequency drops below around 10 Hz then the output voltage diminishes. The same is true for when the frequency approaches around 4 MHz. The flat portion measures around 14.6 dB so if we come 3 dB down on either side of this we see the points marked in yellow. The difference in frequency between these two points is termed the 'bandwidth' of the amplifier. The data area in blue at the bottom of the response shows that the bandwidth is 15.109 MHz.

Output Power

The output power for our CE amplifier is developed across the load resistor. The quiescent DC conditions on the load resistor produce a current of 1 mA and a voltage of 6 V. The power developed in the resistor is then the product of these two terms i.e. 1mA X 6V = 6 mW.

As an AC signal is injected into the amplifier then the output will swing to its greatest extents as the input swings to its peak values (positive and negative peaks). The peak power for the amplifier will therefore be much greater than 6 mW.

Distortion

Distortion can occur when the frequency of the input signal is too high. It can also occur when the magnitude of the input signal is too large. The Transient Analysis below shows the distortion which results when our amplifier has an input signal of 2 V peak (i.e. 4 V peak-to-peak)...