

C1 A Feedback Amplifier

Objectives of experiment

1. To build and test a simple three transistor amplifier
2. To compare performance with simulation results
3. To investigate the effect of negative feedback

Preparation

The amplifier that you are required to build is shown in Figure 1. As preparation for the laboratory you are required to simulate the amplifier using PSpice to find the DC or quiescent voltages at each node of the circuit and to measure the gain and bandwidth. To help you with the simulation a pre-prepared set of Orcad files are available on the ELEC2012 Notes web page. These files need to be unzipped into a folder in your work space. The set of files includes 2 library files jnrbipolar.lib and jnrbipolar.olb library which describe to PSpice how to model a BC547. The Appendix to this experiment describes how they should be included in your design. The simulation should run with either the full version of PSpice or the Student version. See the appendix which explains how the library models need to be included.

After measuring the behaviour of the basic amplifier add the feedback resistor as shown in Figure 2. Run the simulation and measure the gain and bandwidth again. Note the peak in the gain at low frequency if C1 is left at 10 μ F. Increase the value of C1 to 47 μ F (as in Figure 2) and observe the change. Any suggestions as to what is happening here?

Read through the rest of these notes and satisfy yourself that you understand what you have to do in the laboratory. To assist this understanding, sketch the block diagram of the circuit you will use to measure the loop gain of the circuit of the feedback amplifier. You should show how the feedback loop is broken and where the signal generator and oscilloscope are connected. Is the use of an attenuator at the input appropriate in this case?

Experiment

The basic amplifier

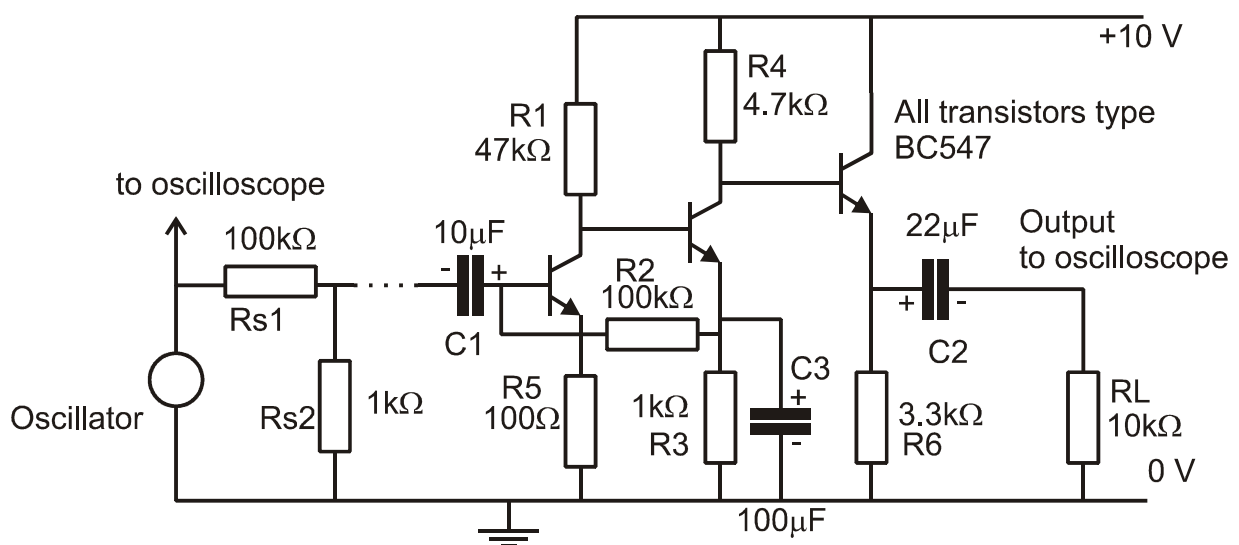


Figure 1

Build the amplifier shown in Figure 1. This amplifier has two common emitter gain stages followed by a common collector stage to provide low output impedance. The first two stages are DC coupled and use feedback to provide stable bias conditions. The emitter decoupling capacitor (C3) means that this feedback only applies at very low frequencies. Within the normal mid-band range of frequencies there is no feedback. This circuit is typical of the type of circuit you will learn to analyse in the Part II *Analogue Electronics* course.

Try to keep the layout tidy, it makes it much easier to find any faults. Metal boxes to provide screening are available, and it is strongly suggested that you put your experimenter board into one of these. The objective is to reduce the interference from the electromagnetic noise which has a high level in the laboratory, from all the computers. The use of screening will make measuring the gain much easier, especially for the amplifier without feedback.

Note that RS1 and RS2 form an attenuator at the input. They are not part of the amplifier but part of the test circuit. This attenuator provides a source impedance to the amplifier of $1\text{ k}\Omega$ at an open circuit voltage $1/101$ times the oscillator voltage. Using the attenuator makes it easier to measure the amplifier gain. It is easier to measure the larger voltage at the input of the attenuator than the small signal at the input to the amplifier itself.

Check your circuit and all connections carefully, and then connect the circuit to a 10V power supply. Do not connect the oscillator yet. Measure the voltages at the collector, base and emitter of each transistor. Are the values reasonable? Do they agree with the results of the simulation? If there are large discrepancies there may be a fault. Small differences may be due to device characteristics.

Connect the oscillator and apply a sinusoidal signal from the oscillator at a frequency of 1kHz and measure the mid-band gain. Before measuring the gain, adjust the amplitude of the input signal so that the output is not distorted, but both the input and output signals are large enough to measure easily. To measure the gain, initially measure the ratio of the output signal to the voltage from the oscillator. Then multiply this by 100 to allow for the attenuator.

Measure the frequency response of the amplifier from 10 Hz to 10MHz. Measure the gain using a logarithmic frequency sequence, ie 10, 20, 50, 100, 200, 500,.....Hz up to the maximum frequency. This characterises the basic amplifier. How does it match with the simulation results?

Applying Negative Feedback

In order to apply negative feedback a $33\text{ k}\Omega$ resistor should be connected between the output and the top of the emitter resistance, R5, as shown in Figure 2. This feedback connection effectively subtracts a fraction of the output voltage from the input voltage. The fraction of the output voltage fed back is determined by the ratio of the feedback resistor ($33\text{ k}\Omega$) to the emitter resistor R5, these resistors act as a potential divider.

Insert the resistor and repeat the measurement of mid-band gain at a frequency of 1kHz.

Measure the frequency response between 10Hz and 10MHz as before. Compare the gain and frequency response with behaviour of the amplifier without feedback.

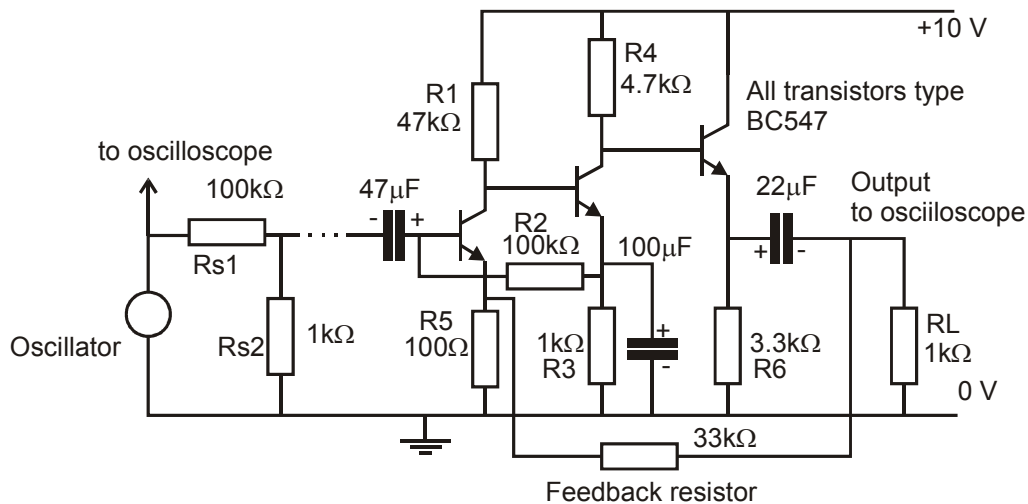


Figure 2

Measuring the loop gain

The feedback signal flows round a complete loop, through the amplifier and back through the feedback resistor. The gain of the feedback amplifier is related to the gain around the loop (T) where A is the open loop gain – the gain of the amplifier without feedback, and A_c is the closed

$$A_c = \frac{A}{1 + T}$$

loop gain – the gain of the amplifier with feedback. To measure the loop gain disconnect the feedback resistor from the output, and feed a signal into it directly from the oscillator (no attenuator). The input attenuator should be left in place to provide the input with the same source impedance as when the signal is being supplied in the usual normal manner. The loop gain is given by the ratio of the output voltage to the oscillator voltage. Measure the loop gain at a frequency of 1kHz.

The loop gain should be equal to the gain of the basic amplifier multiplied by the ratio of the potential divider formed by R_5 (100Ω) and the (33kΩ) feedback resistor. Verify that the measured closed loop gain agrees with that deduced from the equation given above and the loop gain T .

Sensitivity to supply voltage

Measure the open and closed loop gains as before but with the supply voltage reduced to 8 V. Compare these values with the values measured at 10 V.

What conclusions can you draw about the benefits of negative feedback?

Optional extra work

If you have time, use the oscilloscope's spectrum analysis (fast Fourier transform) facility to investigate the input and output spectrum of the amplifier with (Figure 2) and without (Figure 1) feedback. Comment on the distortion of the signal generator and of the amplifier, and of the effect on the latter of applying feedback.

Appendix: Notes on running the PSpice Simulation.

Download from the laboratories web page the “zip” file containing the source files for the simulation (C1_PSpice_Files.zip). The file should be “unzipped” into a suitable directory in your workspace.

There are four files. *Clcircuit.opj* and *Clcircuit.dsn* are the project and design files required by ORCAD Capture. The other two *jnr bipolar.lib* and *jnr bipolar.olb* are library files which provide models for a number of bipolar transistors. (NB these files may be of use to you subsequently to provide transistor models).

The circuit may be analysed using either the full version of ORCAD or using the Student version.

The Procedure is

- Run capture
- Open the project *Clcircuit.opj*. This contains the schematic for the circuit.
- Open the schematic page.
- You may wish to select the libraries to use (this is not essential at this stage as the devices required are in the design cache, but if you wish to make changes you may need access to other libraries). The transistor models are in the library *jnr bipolar.olb* which is located in the directory with the project source files. The schematic library files are installed using the box brought up by the place component button on the toolbar.
- Set up the analysis profile you require. Set the analysis type under the “Analysis” tab. It is probably best to start with AC analysis (it also gives the quiescent voltages and currents), in this case, you will need to select the frequency range and number of points per decade). You will also need to add the PSpice library with the transistor models. This is done by selecting the “libraries” tab and then locating the file *jnr bipolar.lib* using the Browse button. Having found and selected the file open it then add it to the simulation profile using the “Add to Design” button (not the “Add as Global” button). Ignore the warning that it can’t write the .ini file. This includes this library file in this design.
- The design is now ready to simulate.

A few general points

Libraries:- Orcad/PSpice needs separate library files for the schematic capture and the simulation model. The “.olb” library files contain the schematic symbols and part definitions. The full version of ORCAD has many libraries provided; the student version has a restricted set. The installation of these libraries is performed from the dialogue box brought up by the “place part” button on the toolbar.

For simulation a second library is required to give the simulation models for PSpice. These libraries have a file extension “.lib”. For each schematic library supporting PSpice simulation there is a corresponding PSpice library. Installing the schematic library does not necessarily install the PSpice library, unless it is in the default library path. If you add a library in your accessible workspace the PSpice library must be done through the simulation profile. This is done using the “Libraries” tab. Using PSpice over the network the library must be added using the “Add to Design” button.

Multipliers:- The powers of 10 used by PSpice for values are defined by using the suffix letters

Value	letter	name
10^{-12}	p	pico
10^{-9}	n	nano
10^{-6}	u	micro
10^{-3}	m	milli
10^3	k	kilo
10^6	meg	mega (NB m is interpreted as milli)
10^9	G	giga

NB Spice does not differentiate between capital and lower case letters

Ground:- The ground symbol must be called “0” (zero). If it is not available from the library then a ground symbol must be renamed. If a lot of floating node errors are generated by the simulator a common cause is the lack of an ground symbol named 0.