Analog Electronic Devices
(ESI038 / SE047)

Lab Experiment 2
BJTs: Designing a Single-Stage Amplifier

Prof. J. E. Rayas-Sánchez
January-May 2007

Objectives

The objectives of this lab experiment are:

a) to design an stabilized single-stage voltage amplifier using a discrete bipolar transistor

b) to simulate some of the most important parameters of a voltage amplifier using a SPICE simulator

c) to measure some of the most important parameters of a voltage amplifier using lab equipment

d) to compare hand calculations, SPICE simulations and lab measurements for a BJT amplifier.

Components and Instrumentation

A general purpose discrete BJT (2N2222A, 2N3904, 2N3906, etc.)

Several resistors, and capacitors

A potentiometer (approximately in the range of 0-10KΩ)

An adjustable DC power supply

A low frequency function generator

A digital multi-meter (DMM) with 2½ or more digits

An oscilloscope with x10 or x1 probes

A circuit simulation software: OrCad, WinSpice, Electronic Work Bench, or something similar.

Theoretical Procedure

1. Taking into account the relevant parameters reported in the manufacturer's data sheets of the selected general purpose discrete BJT (either npn or pnp), design a single-stage voltage amplifier to satisfy the following specifications:

    a. Magnitude of the voltage gain at middle frequencies of at least 80 V/V

    b. Magnitude of the input impedance at middle frequencies of at least 1KΩ

    c. Magnitude of the output impedance at middle frequencies of at least 1KΩ

    d. With a DC operating point stabilized against β variations

   e. With a minimum output voltage swing of $6V_{p\text{-}p}$ (distortion limits greater than 3V)
   f. Loaded with a 10KΩ resistor
   g. Biased with a DC power supply of 12V

2. Once you have calculated all the component values of your amplifier, re-analyze it and fill in the second column of Table I.

Lab and Simulation Procedure

1. Simulate your complete single-stage voltage amplifier using SPICE:
   a. Perform a DC simulation to observe its DC collector current, I_C.
   b. Perform a transient simulation from 0 to 5 ms using an input sinusoidal signal v_s of 10mV of amplitude and 1KHz of frequency. Plot the output voltage v_o at the load resistor of 10KΩ, and calculate the voltage gain $A_v = v_o/v_s$. Increase the amplitude of v_s from 10mV to 100mV and plot again the output voltage v_o to measure the distortion limits.
   c. Perform an AC sweep simulation from 10Hz to 100MHz to observe the steady-state frequency domain response of the amplifier. Use an AC input signal of 1V. Plot the magnitude of the voltage gain $|A_v| = |v_o/v_s|$ versus frequency, and plot also the magnitude of the input impedance $|Z_{in}| = |v_s/i_s|$ versus frequency.
   d. Modify the circuit topology so that you can measure the output impedance with SPICE (delete the original AC input signal and apply a new AC input signal at the amplifier's output, then plot the output impedance $|Z_o| = |v_o/-i_o|$ versus frequency).
   e. Using the results obtained in the previous four steps, fill in the third column of Table I.
2. Implement in the lab your amplifier:
   a. Since accuracy is important, measure with a DMM the resistors you are using (i.e., do not assume that a nominal 1KΩ resistor will have 1000 ohms), as well as the DC voltage your are applying (make sure it is equal to 12V).
   b. Using a DMM, measure the DC collector current I_C.
   c. Using a function generator, apply a small sinusoidal input amplitude (5-15mV) and 1KHz of frequency to your amplifier. Using an oscilloscope, measure the input and output voltages and calculate the voltage gain. Using a potentiometer and the oscilloscope, measure the input resistance.
   d. Increase the amplitude of the function generator from 10mV to 100mV and measure the output voltage with an oscilloscope to calculate the distortion limits.
   e. Measure in the lab your amplifier's output resistance at 1KHz. You have to come up with a simple method to measure it. Describe the method you used for measuring the output resistance.
   f. Using the results obtained in the previous four steps, fill in the fourth column of Table I.

TABLE I

Parameter	Hand calculation	SPICE simulation	Lab measurement
I_C (DC)			
$\|A_v\|$ (at 1KHz)			
$\|Z_{in}\|$ (at 1KHz)			
$\|Z_o\|$ (at 1KHz)			
Positive distortion limit, DL^+			
Negative distortion limit, DL^-			

Report

Write a report including all the theoretical, simulation and lab procedures and measurements. Since it is a design problem, it is very important to justify all the critical decisions during the design process (related to calculations as well as to selection of components). The report must include the schematic of the complete final circuit, indicating the tolerances for the passive components (and their exact values when they were measured). The SPICE simulation results must be clearly distinguished from the lab measurements. All the waveforms measured in the lab must also be reported. Include in the report your conclusions and references.

Deadline and Assessment

The deadline for submitting the report is on Tuesday March 6, 2007. The report can be written either in English or in Spanish.

This lab experiment can be realized in teams of up to 3 students. The evaluation of the report will be as follows:

Quality of the report	30%
Accuracy of the theoretical analysis	30%
Lab measurements and procedures	40%

If the report is written in acceptable English, an extra 10% can be granted.