Introduction to the ECE331 lab

Introduction

The objective of the lab is to experimentally verify the lecture material and to get familiar with standard lab instruments such as a power supply, multimeter, signal generator, and oscilloscope throughout the course of designing an audio power amplifier. The lab consists of five sessions as shown below.

Lab 0: Introduction to lab equipment and components

Lab 1: Common-source amplifiers

Lab 2: Current mirrors

Lab 3: Push-pull power amplifier

Lab 4: Operational amplifier

Lab 5: Audio power amplifier with feedback

Labs 1-4 serve as the components for Lab 5 in which you build a linear class-AB audio power amplifier with feedback capable of driving an 8-Ω load, which is the standard impedance of audio speakers, as the final product.

A simple system diagram of the audio power amplifier is shown in Figure 1. The amplifier uses a modified version of the common-source amplifier from Labs 1 and 2 as a voltage amplifier as well as a class-AB biasing stage. An operational amplifier from Lab 4 is used as an error amplifier, which is the heart of a feedback system. Current mirrors from Lab 2 provide bias currents to the common-source amplifier and operational amplifier. The push-pull power amplifier from Lab 3 acts as a buffer to drive an 8-Ω load.

Each block is designed in transistor level, so you are going to design the entire audio power amplifier in transistor level as well without using any black boxes (integrated circuits). This is the lowest level of circuit design.

Labs

Each lab consists of a preparation part and in-lab work shared between 50 minutes Spice simulation challenge and 2 hours circuit implementation. All of which are group work. Form a group of two students before the first lab and stay with the same partner throughout the semester.

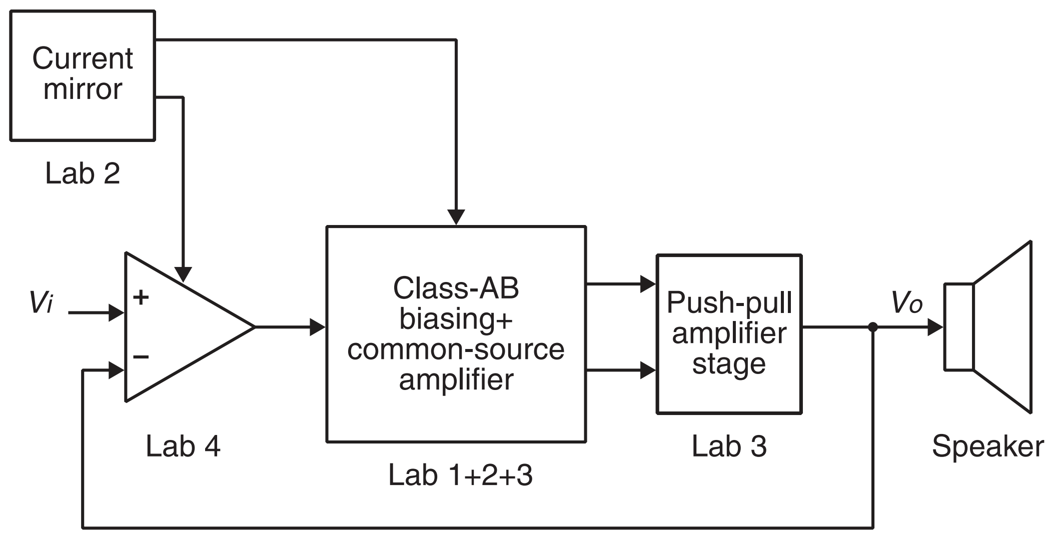

Figure 1: Audio power amplifier system diagram.

Preparation

Preparation involves hand analysis and simulation of the target circuits for theoretical understanding. Preparation has to be completed before each lab session and **must be ready to be checked by your TA at the beginning of the lab session. Each group prepares one set of the preparation material.**

Use the device parameters shown in Table 1. Note that your hand analysis may not match simulations very well because the transistor models used in simulation include many high-order effects not captured by the square law used for hand analysis. However, hand analysis is still important to establish a starting point for circuit design using a simulator.

For circuit simulation, use the model library available on the course website.

Note that just analyzing the circuit and dumping simulation results will not deserve a full mark. The result of the preparation must be neatly organized to prove that you have completed all of the steps shown in the lab manual. Please provide labels, clear indications where necessary and write down your answers to the questions. It is your responsibility to make sure the printed graphs are clearly visible. Staple everything you prepared and put your name on the first page. Results shown on a computer screen or loose sheets of paper will not be considered for grading. Remember that the preparation material can be held by the TA for some time at the beginning of each lab for grading. If you expect to need the material during grading, make a copy of the material.

Lab: Spice Challenge

At the beginning of each lab (except Lab 5) you will be given Spice simulation challenge. This part is mostly similar to the simulations you have performed during the preparation part.

Table 1: Device parameters

Device	Type	V_T (V)	$\mu_{n/p}C_{ox}W/L$ (mA/V^2)	V_A (V)
ALD1101	NMOS	0.71	4.49	80
ALD1102	PMOS	-0.65	-2.10	-19

You will use LTSpice to complete this challenge and have 50 minutes to finish. Completing the preparation part of the lab and general knowledge of the course should be enough to finalize this part. In parts you are stuck, you are free to ask questions to your TA. But we do NOT encourage group to group collaboration in this part.

Lab: Implementation

The circuits analyzed during preparation will be implemented and experimentally verified in the last two hours of each lab session.

The main purpose of each experiment is to verify the results of the preparation step using real components. Circuits are implemented on your breadboard. It is always a good practice to put components close to each other using short wires. Long wires may act as an inductor which might cause various problems such as circuit instability. Place large (roughly 100 μF) and small (about 0.1 μF) capacitors across the power rail. These capacitors maintain the low impedance of the power supply which goes up with frequency without the capacitors due to the inductance of the cable used to connect the power supply unit to the breadboard.

As required by the lab manual, you will need to record and plot various data on your lab note during a lab session. Do it neatly so TAs can read your results. Clearly show important numbers, and label important points on your plots.

Devices

ALD1101 and ALD1102 chips are the NMOS and PMOS transistors, respectively, used in the lab. Each chip has two identical transistors inside. Device parameters of these devices for hand analysis are given in Table 1. Transistor symbols without a device name you see in lab manuals represent **one** of the transistors in one chip. IRF510 and IRF9510 will also be used in Labs 3 and 5 as power transistors. It is strongly recommended to read and bring the datasheets of transistors for your knowledge and convenience. Spice models are available on the course website.

You will be provided with all of the components required to complete the lab. The lab manual has a list of components needed for each lab if you want to refer to.

Table 2: Marking scheme

Lab #	Prep	Spice Challenge	Imp.	Total
0	-	-	-	-
1	4	2	4	10
2	4	2	4	10
3	4	2	4	10
4	4	2	4	10
5	4	-	6	10
Total				50

Circuit simulator

Use LTspice IV for circuit simulation for the preparation part. You will need to use it for the Spice challenge part of the lab as well. It is a circuit simulator available for free from Analog Devices at

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

You can follow the instructions on the course website under LABS tab ”Steps to Using LT-Spice” part. Note that the student version of PSpice is not usable for this course because of its strict limit on the transistor count.

Lab Grading

The grading for the labs will consist of marking

- Lab Preparation (marked as a group)
- Spice Challenge (marked as a group)
- Implementation (marked as a group)

according to the Table 2. Note that there is no Spice Challenge for Lab 5.