Lab 2 Day 1: The Amplifier Project

Electronics II

Contents

Purpose 2

Introduction 2

Day 1: Single Transistor Amplifier and 2-Transistor Amplifiers 2
- Part 1: Circuit Construction and DC Measurements . . . 2
  - Experiment . . . 3
  - Report . . . 3
- Part 2: AC Measurements for Single-Transistor Amplifiers . . . 4
  - Pre-lab . . . 4
  - Introduction . . . 4
  - **Simulating the Circuits in Multsim** . . . 5
  - **Performing a Bode Plot of a Common Emitter Amplifier in Hardware** . . . 6
  - Report . . . 7
- Part 3: 2-transistor Amplifiers . . . 8
  - Pre-lab . . . 8
  - Experiment . . . 8
  - Report . . . 8

Schedule for this lab:

Day 1:	Single-Transistor and 2-Transistor Amplifiers
Day 2:	Present your Cascode Amplifier Design to the TA and Test your Circuit
Day 3:	Continuation of Week 2. Your amplifier should be working by the start of this period so that you can demonstrate it to the TAs

The report should discuss all of the results including Day 1. You must discuss your design and show the complete process and results.

Purpose

The purpose of this laboratory is to investigate the use of BJTs as amplifier circuit elements. First, the three basic configurations (CE, CC and CB) are observed. Then, by proper combinations and permutations, 2-transistor amplifier configurations can be studied for improved gain-bandwidth performance. Finally, a specific configuration is required to be designed to meet or exceed a prescribed set of specifications.

Introduction

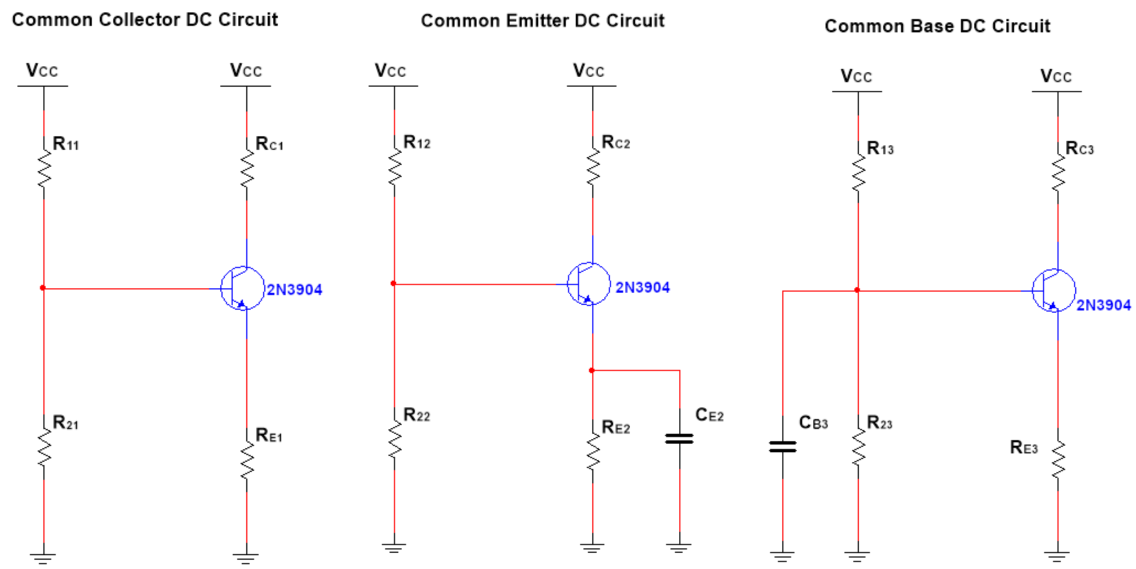
Before starting this project, read the relevant class notes, and the sections of this laboratory outline dealing with design approaches.

Day 1: Single Transistor Amplifier and 2-Transistor Amplifiers

You will be constructing and measuring 3 different single amplifier configurations and 3 different two-transistor configurations. Your results should help you identify the strengths and weaknesses of each one. Note that while it is mandatory to do the prelab for day 1 prior to coming in to the lab, you are strongly suggested to do the prelab for the day 2 as well, as it is long and intensive.

Part 1: Circuit Construction and DC Measurements

Construct the circuit shown in Figure 1. This circuit contains 3 separate amplifiers.



(a) Common Collector (b) Common Emitter (c) Common Base

Figure 1: DC Biased Single Transistor Amplifiers

The component values are as follows (i.e. $R_{1.1} = R_{1.2} = R_{1.3} = 100\ k\Omega$):

Component (CC)	Component (CE)	Component (CB)	Component value
$R_{1.1}$	$R_{1.2}$	$R_{1.3}$	100 kΩ
$R_{2.1}$	$R_{2.2}$	$R_{2.3}$	39 kΩ
R_{C1}	R_{C2}	R_{C3}	5.6 kΩ
R_{E1}	R_{E2}	R_{E3}	3.3 kΩ
	C_{E2}		100 μF
		C_{B3}	1 μF

Note: R_{C1} is to be present when verifying the DC operating point. However, when performing your AC measurements, it is to be shorted out.

Experiment

- See Nagui's website for the circuits for Lab 2. The URL is shown here:

  `http://www.doe.carleton.ca/~nagui/Elec3509/Lab2/`

  You must enter the values for the resistors and capacitors from the part 1 table.

- The DC network is already built, along with the AC nodes connected from Figure 2 and 3. You must disconnect the wires connecting the input and output test circuits, and perform an Interactive simulation.

  Go to Simulation → Analysis and Simulation, and from the window that appears, find Interactive from the menu options, and choose to Run.

- For every transistor, take DC measurements of all relevant voltages (V_B, V_E, V_C and V_{CC}) using voltage probes. These probes can be found through Place → Probe, and you are able to choose a voltage probe, or differential probe.

  You should use the voltage probe on nodes along with the resistor values to determine the terminal currents from the following relationships:

$$I_C = \frac{V_{CC} - V_C}{R_C} \tag{0.0.1}$$

$$I_E = \frac{V_E}{R_E} \tag{0.0.2}$$

$$I_B = I_{R1} - I_{R2} = \frac{V_{CC} - V_B}{R_1} - \frac{V_B}{R_2} \tag{0.0.3}$$

  Note that I_B is calculated by subtracting two large values that are very similar, and so the error will be large if you are not accurate with your measurements. Check if $I_E \approx I_B + I_C$ and determine the DC value of β by calculating $\frac{I_C}{I_B}$

- Define the "AC ground" node of the three transistors as follows:
  - The first transistor is to be connected as a common-collector amplifier. Thus, the collector should be by-passed to ground either by a capacitor, or, by changing R_{C1} to short-circuit the collector to V_{CC}.
  - The second transistor is to be connected as a common-emitter amplifier by connecting a large value (100 μF) capacitor C_{E2} from E2 to ground.
  - The third transistor is to be connected as a common-base amplifier by connecting a capacitor C_{B3} ($C_{B3} \geq 1$ μF) from B3 to ground.

    Make sure these conditions are met and built into the CC, CB, and CE Amplifiers.

Report

Briefly summarize the results you have obtained here. Answer the following questions:

- Why can R_{C1} be made 0 when testing the amplifier? Why do you need to have it when checking the DC operating point?
- Could we instead use a capacitor like we do for the CE and CB amplifiers?
- Why do we need R_{E2}? Why can't we just short this node to ground?

Part 2: AC Measurements for Single-Transistor Amplifiers

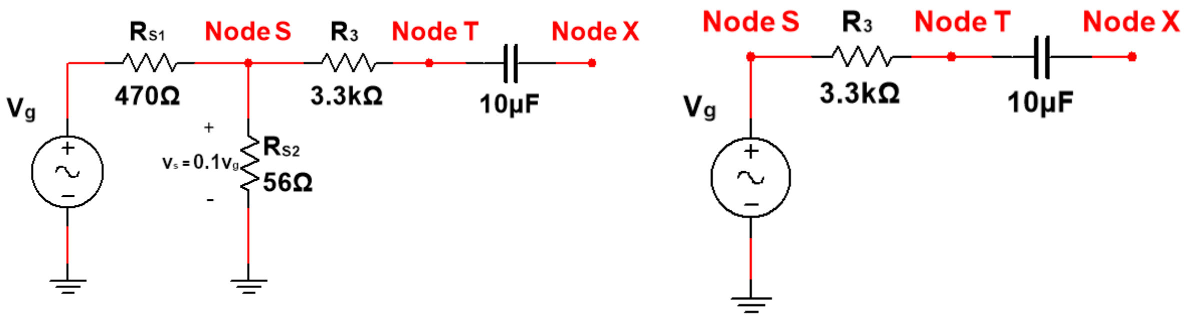
Pre-lab

Using the formulas in the notes, **identify the expected gain, input and output impedance, and high/low frequency cut-off points of each amplifier** (this is a lot of work so plan accordingly). For component parameters, use the values you measured in lab 1, or make reasonable assumptions.

For each amplifier, a 3.3 kΩ resistor is placed at the series of the input, and the source impedance of the generator is 50 Ω. The load impedances and the coupling capacitors can be seen in Figure 2 and Figure 3.

Introduction

- You must reconnect the network seen in Figure 2 to the common emitter amplifier. You should understand the use of the voltage divider if you've completed the prelab. For the Common Emitter, the input voltage is Node S. The Low Gain Circuit (b) is used for the Common Collector and Common Base.



(a) High Gain Input Stage Circuit

(b) Low Gain Input Stage Circuit

Figure 2: Different Input Stage Circuits

- Notice the load circuit in Figure 3. This circuit is built as a high pass AC measuring circuit. Note the jumper between node Z and the load resistor. What happens to the load when the jumper is removed? The purpose of this jumper is for finding the output impedance, as the output load will change dramatically when the load is set, or removed.

- Typically, when measuring the output or input nodes of Figures 3 and 2 with an oscilloscope, you would have to remove the DC offset caused by the current leak from the capacitors and other sources. You would be able to remove the DC offset by setting the scope to AC coupling, but for the purposes of simulation, these effects are inconsiderable, as the capacitors are near ideal capacitors.

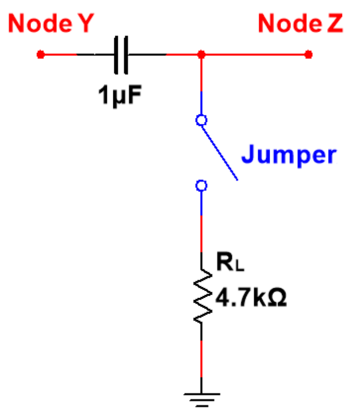


Figure 3: Schematic of Load Test Stage

Simulating the Circuits in Multsim

* **Note** that students who wish to also perform this experiment in hardware must also perform the simulation of the circuit.

- **CE Amplifier**

  Find, download, and open ELEC3509_Lab2_CE_Amplifier.ms14 from Nagui's Website as seen in the experiment section in part 1. You must enter the values for the resistors and capacitors from part 1.

  – Node X of Circuit (a) of Figure 2 must be connected to the base of the Common Emitter Amplifier. Connect the load circuit's node Y of Figure 3 to the collector of the Common Emitter Amplifier.

  – You are able to find a two channel oscilloscope by going to Simulate → Instruments → Oscilloscope, and add it to the design window. The A and B channels can be used to measure the differential voltage. Your Ext Trig should be used to sync your oscilloscope to the input Vs, and so it's important that you use the trigger signal to measure across Vs. In the oscilloscope window, you should set the option from None to Normal in the Trigger panel.

  – Attach oscilloscope probes to monitor v_Z (at node Z). Since the jumper is connected, v_Z is v_O. Set the signal frequency of Vs to 1 kHz and adjust the output level (amplitude) of the voltage source until v_o shows an output peak-to-peak swing of 2 volts, use your gain calculation to get as close to your 2 volts as possible.

  – You are able to measure from node S to node X by using one of your channels, and attaching the positive wire to node S, and the negative wire to node X, and running an interactive simulation from Simulate → Analysis and Simulation. You can pause the interactive simulation at any time, and read the values from the oscilloscope using the cursors for the peak to peak voltage.

    Also measure the node S signal to ground, and note the peak to peak voltage.

  – Measure v_o' by disconnecting the jumper, connecting one of the oscilloscope channels, and measuring the voltage peak to peak with the oscilloscope cursors.

  – Determine R_{in}, R_{out} and the mid gain A_{mid} using the following relationships.

$$R_{in} = R_3 + R_{inA} = R_3 + \frac{v_x}{i_{R_3}} \tag{0.0.4}$$

$$R_{out} = \frac{v'_o - v_o}{i_{R_L}} \tag{0.0.5}$$

$$A_{mid} = \frac{v_o}{v_s} \tag{0.0.6}$$

  – Next, we wish to find the **high frequency cutoff** point of this circuit by finding where the gain drops by 3 dB. Short the jumper again for a load R_L and observe v_o and v_s with the two oscilloscope channels. Increase the frequency of the voltage source until v_o drops to 1.4 volts peak-to-peak. Ensure that the value of v_s remains constant. If it decreases or increases with frequency, adjust the signal generator to maintain the same value. This frequency is the upper cutoff (or 3 dB) frequency, f_H, of the amplifier. Next, decrease the frequency of the signal source until, again, v_o drops to 1.4 volts peak-to-peak (again keeping vs constant) to obtain the **lower cutoff frequency**, f_L, of the amplifier.

- **CC Amplifier**

  Find, download, and open ELEC3509_Lab2_CC_Amplifier.ms14 from Nagui's Website as seen in the experiment section in part 1.

  – Add the oscilloscope from the Simulate → Instruments menu, and use the trigger signal on the voltage source. You should ground the negative voltage probes for the oscilloscope since you are not required to measure across any resistors for this amplifier.

  – Connect the Low Gain Input Circuit (b) of Figure 2 node X to the base of the Common Collector Amplifier. Connect node Y of the circuit of Figure 3 to the emitter of the CC Amplifier. Set the input voltage so that the output node Z signal has a 2 volt peak to peak amplitude. 2.

  – Repeat the previous steps to get R_{in}, R_{out}, A_{mid}, f_H and f_L of the CC amplifier.

- **CB Amplifier**

  Find, download, and open ELEC3509_Lab2_CB_Amplifier.ms14 from Nagui's Website as seen in the experiment section in part 1.

  - You must attach the two channel oscilloscope from Simulate → Analysis and Simulation, and trigger the oscilloscope with the input Vs.
  - Connect the Low Gain Input Circuit (b) node X from Figure 2 to the emitter of the Common Base Amplifier. Also connect the load circuit node Y to the collector of the CB Amplifier. Set the voltage source so that the output node Z has a 2 volt peak to peak signal. 2.
  - Repeat the previous steps to get R_{in}, R_{out}, A_{mid}, f_H and f_L of the CB amplifier.

Performing a Bode Plot of a Common Emitter Amplifier in Hardware

* **Note** that students who have signed up for hardware should perform this experiment through hardware, as it uses the real life oscilloscope tool.

- The CE Amplifier is already connected to some of the VLSI computers, where if you log into the VLSI desktop computers, you can access the scopes. If you are a little lost on how to access the scopes in this manor, refer yourself to Lab 0 of Nagui's website:

  `http://www.doe.carleton.ca/~nagui/Elec3509/Lab0/`

  You must access your assigned desktop, and start the remote scope view from a browser.

- Once you have access to the Remote Front Panel, you are able to start the process for the Bode plotting tool. In the Analysis panel, choose the bottom right button, which has two sets of three black squares. Clicking on this will bring up the App Selection menu on the display. Choose the Bode Plot.

Figure 4: App Selection Option

- The Bode plot should be without data points, and look like that in Figure 5. You must make sure the settings are properly configured as to sweep over a large range, and to use the smallest signal. You should set your Start frequency to 10 Hz, and your Stop frequency to 10 MHz, with 20 Pts/decade.

Figure 5: Bode Plot Window from the Oscilloscope

In the Amplitude Profile (Ampl. Profile), change all Amplitudes to 20 mV, and the Frequency of Index 1 to 10 Hz.

Run the sweep, and notice how the Gain and Phase change for this Common Emitter Amplifier. If one of the Gain, Phase, or Amplitude are not in the window, you can drag the Orange, Blue, or Green arrows to the left of the Bode display to change the vertical offset.

- You are able to save this data to a USB connected to the scope using the Save button. You should note down the name of the file, and which station you are working at.

Figure 6: Save Button

- When you are finished analysing the data, hit the Reset button, and log out.
- You should send an email to Nagui at nagui@doe.carleton.ca when you want to receive the data. Attach the Lab number, your lab section, your file name, and your station number along with your own name.

  If you experience any difficulties throughout the lab, contact your TAs, and then possibly the hardware lab coordinator at (613)-520-2600 ext. 8166, and give them the coordinator your station number and issue.

Report

Show a complete table summarizing the values of calculated and measured of R_{in}, R_{out}, A_{mid}, f_H and f_L for all 3 amplifiers. Comment on any differences you see between measured and calculated values. Explain any differences you see. Try to be as specific as possible.

In a normal circuit, the AC measurements could be prone to many sources of error. Try identifying some of these possible sources of error that would be encountered in a real circuit; how could we avoid these sources of error?

Comment on the differences between all 3 amplifiers. Mathematically explain any differences you see: for instance if one amplifier has a higher gain, explain why that is so. Which ones are better for which tasks and why?

Part 3: 2-transistor Amplifiers

In addition to the input signal coupling network and the output loading network, you will need an AC coupling capacitor to connect from the first transistor to the second one. Since the large value capacitor is usually of the polarized type (electrolytic), make sure that you observe the polarity of the capacitor when connecting it to the circuit.

Pre-lab

- For each of 2-transistor amplifiers you will test in the section below, qualitatively estimate what you expect their properties to be (eg. R_{in}, R_{out}, A_{mid}, f_H and f_L). Feel free to do a full analysis here if you like, as you will need to do that for the report anyway.

Experiment

To change the design window size: You should change the widths of the design windows in order to accommodate larger circuits. You can do this by going to the Window menu → Workspace → Custom Size, and change the width of the window.

- **CE-CB Amplifier**

  You must cascade the Common Emitter with the Common Base, which means both the CE and CB must be on the same page. You can highlight the components and copy them over to another design page easily. The input stage and output stage still use the circuits from Figures 2 (a) and 3, where the input circuit uses circuit (a) from Figure 2.

  The output for the CE (the collector) is connected to the input of the CB (the emitter) through a 22 μF capacitor, while the rest of the circuit remains the same.

  Include the oscilloscope from Simulate → Instrument → Oscilloscope, and trigger the oscilloscope using the input voltage source.

  Repeat all the AC measurements to find the amplifiers parameters (R_{in}, R_{out}, A_{mid}, f_H, and f_L), these steps are similar to those in part 2, and the calculations are also similar.

  What do you notice about the cascading amplifiers parameters?

- **CC-CB Amplifier**

  Cascade the Common Collector and Common Base amplifiers, such that the input of the CC uses the Low Gain Input Stage Circuit (b) from Figure 2, the output of the CC amplifier is coupled with a 22 μF capacitor to the CB input, and the output of the CB amplifier is connected to node Y of Figure 3.

  You must include the oscilloscope for measurements, and perform the AC measurements for the amplifier parameters (R_{in}, R_{out}, A_{mid}, f_H, and f_L).

- **CC-CE Amplifier**

  Cascade the Common Collector and Common Emitter amplifiers. The input of the CC comes from the first voltage divider input circuit and connects to the base, the output of the CC comes from the emitter, and is coupled with the base of the CE using a 22 μF. The output of the CE is connected to node Y of the circuit in Figure 2.

  You should include the oscilloscope for measurements, and perform the AC measurements for the amplifier parameters (R_{in}, R_{out}, A_{mid}, f_H, and f_L).

Report

Include a table summarizing the measured values of R_{in}, A_{mid}, f_H and f_L. **Calculate expected values** for these and compare them. Like with the 1 transistor amplifiers, explain any differences between the two set of values.

Comment on the differences between all three 2-transistor amplifiers. Like you did for the previous section, explain any differences in performance. Which ones are better for which tasks and why?

In weeks 2 & 3 you will be designing a cascade amplifier, a variation on the CE-CB amplifier. Identify what advantages this has over any of the single transistor amplifiers you tested. What are the advantages and disadvantages of using the cascade topology over any of the 2-transistor topologies you just tested?