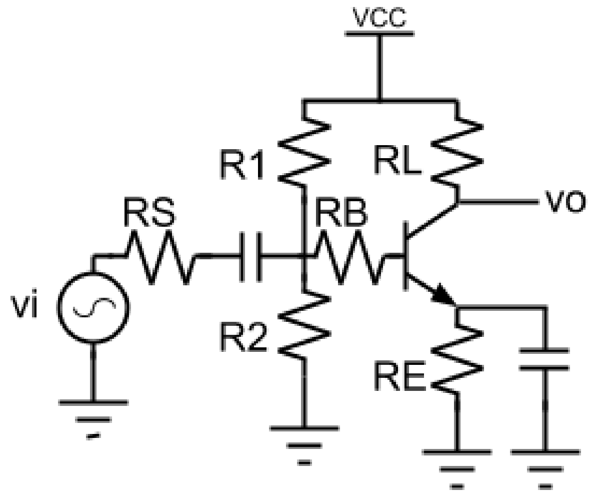


Figure 1: An example design for a common emitter amplifier.

# 1 Warm-Up Problems

1. The above circuit is a common emitter amplifier with a practical biasing scheme. Assume it operates in the mid-band and that $R_s$ is part of the source rather than the amplifier.
   (a) Find $r_{in}$, $r_{out}$, $a_v$ and $V_{SW}$ for this amplifier. Answer in terms of the small signal parameters of the transistor $(g_m, r_\pi, r_o)$ and resistor values.
   (b) Find $V_{OMIN}$ and $V_{OMAX}$. You may assume that a voltage divider equation accurately captures the voltage between $R_1$ and $R_2$. Answer in terms of $V_{CC}, I_C$ and resistor values.
2. What is the current gain, $a_i$, for a common emitter amplifier? Current gain is measured by applying a test current source at the amplifier input and measuring the current through a small signal short at the amplifier output.

# 2 Lab Introduction

In this lab you will build and characterize a common emitter amplifier. This is practice for the design project, where you will use a small number of transistors to design a (possibly multistage) amplifier to a specification. The learning goals are listed below:

- Understand the design process for amplifiers, a process which applies to most analog circuits
- Practice reconciling analysis, simulation and measurement.

# 3 Design a Common Emitter Amplifier

In this lab you will design a common emitter amplifier to meet the set of design specifications below. You may do so by selecting component values for the common emitter amplifier pictured in Figure 2. You may, if you prefer, select a different biasing scheme, make small modifications to this design or select another amplifier, but the schematic in Figure 2 is sufficient to meet the specifications.

You must begin your design by making hand calculations which help you pick your compnent values. After that you should simulate your design to make sure those component values work in

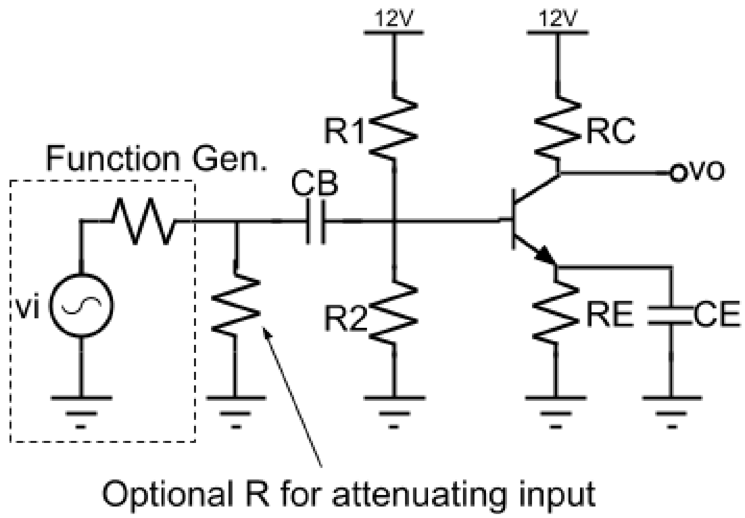


Figure 2: A sample design for a common emitter amplifier.

simulation. Finally, you must build your design and experimentally show that it meets the design specifications. Be sure to calculate, simulate and measure the power consumption of your design.

When simulating, start with a DC simulation to verify your bias point, then an AC simulation to check your small signal model, and finally a transient simulation to see if there are any non-linear effects that you missed. One common mistake that students make is only running transient simulations, whcih are quite confusing without the context of DC and AC simulations. Another common mistake is using the stock BJT models in LTSpice, which have too high a value of $\beta$. You should make your own models with an appropriate beta because you will need them throughout the semester.

- $r_{in} > 1\text{k}\Omega$
- $r_{out} < 5\text{k}\Omega$
- $a_v = 200 \pm 10\%$
- $V_{SW} > 2\text{V}$ as measured by absence of visible clipping.
- $f_{low} < 2\text{kHz}$ (we haven't learned how to calculate this yet, so you can do a rough justification of your capacitor values instead of computing $f_{low}$. However, be sure to simulate and measure $f_{low}$ carefully and be sure simulations and measurements agree.)
- Use only one 12 V power supply.
- Your signal source has a source impedance $> 8\ \Omega$ and may not have any DC offset
- Use one transistor: a 2N3904

**Required Data**: Table of calculated, simulated and measured $r_{in}$, $r_{out}$, $a_v$, $V_{SW}$. Traces showing the $r_{in}$, $r_{out}$, $a_v$, $V_{SW}$ and accompanying descriptions of experimental setups. Explanations of differences between calculated, simulated and measured models.