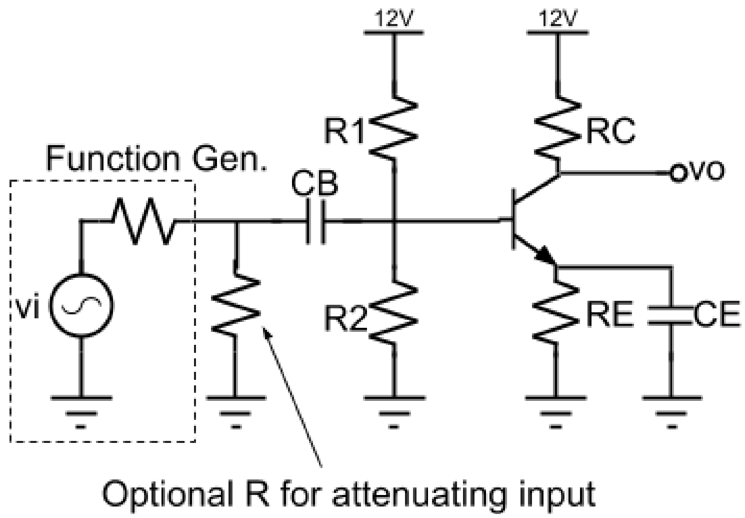

Figure 1: A sample design for a common emitter amplifier.

1 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.

2 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 1. 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 1 is sufficient to meet the specifications.

The design specifications are as follows:

- $r_{in} > 1\text{k}\Omega$
- $r_{out} < 5\text{k}\Omega$
- $a_v = 200 \pm 10\%$
- $V_{SW} > 2\text{V}$ as measured by absence of visible clipping.
- 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

You must simulate and measure three additional quantities: P, the total power consumbed by the amplifier, and f_{high} and f_{low}, the upper and lower corner frequencies of the mid-band. You must make an analytical predication of P in addition to measuring and simulating it, but you don't need to calculate f_{high} and f_{low} because we haven't learned how to do so yet.

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 simulation. Finally, you must build your design and experimentally show that it meets the design specifications.

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 β. You should make your own models with an appropriate beta because you will need them throughout the semester.

Required Data: Table of calculated, simulated and measured r_{in}, r_{out}, a_v, V_{SW} and P. The table should include measured and simulated f_{high} and f_{low}, but no calculations are needed for those quantities. Traces showing how you measured the r_{in}, r_{out}, a_v, V_{SW}, and accompanying descriptions of experimental setups. Explanations of differences between calculated, simulated and measured models. Brief calculations indicating how you reached your component values.