2. Common-Emitter Amplifier

Chapter 2 Goals

- Become familiar with gain/bandwidth
- Become familiar with a CE amplifier
- Become familiar with LTspice and use it to analyze the CE amplifier circuit
- Breadboard and test a CE amplifier circuit

Note: Carefully handle components as many of them will be used later in the radio. Avoid abusing the leads. Your GTA will indicate the proper way to handle the parts and bend the leads.

2.1 Gain and Bandwidth

An amplifier is characterized by its gain and its bandwidth. Referring to Figure 2.1, the AC voltage gain $A(f)$ is the output voltage amplitude $v_o(f)$ divided by the input voltage amplitude $v_i(f)$, or

$$A(f) = \frac{v_o(f)}{v_i(f)}$$

The "*f*" in parenthesis indicates these values may be frequency dependent. Most amplifiers will exhibit a fairly constant gain over some range of frequency, and this range is referred to as the half-power bandwidth, the 3-dB bandwidth, or simply the bandwidth. The gain over this bandwidth is referred to as the *mid-band gain*. The bandwidth is best observed using a Bode plot, as shown in Figure 2.2. Here, the voltage gain is converted to decibels as

$$A(dB) = 20\log\left(A(f)\right)$$

Frequency is plotted on a log scale.

You may recall that power is related to voltage squared. So the output will drop to half power (1/2) when the voltage gain drops to ($1/\sqrt{2}$), or 0.707, of its maximum value. Note that 20 log(0.707) = -3 dB. To find the 3-dB bandwidth (BW) on the Bode plot, as shown in Figure 2.2, we find the low and high frequency corners, f_L and f_H, where the gain has dropped by 3 dB. Then,

$$BW = f_H - f_L$$

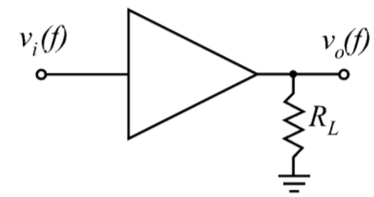


Figure 2.1: simple amplifier

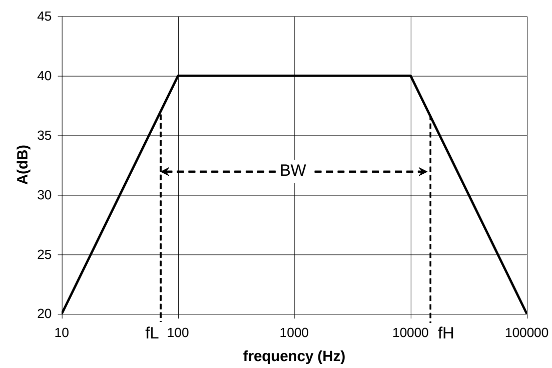


Figure 2.2: Idealized amplifier Bode plot with 3-dB bandwidth indicated.

Most commonly, a signal generator is used to control the amplitude and frequency of $v_i(f)$. $v_o(f)$ is then measured on an oscilloscope. Most generators have some nonlinearity that causes their output voltage to vary with frequency. Thus, if a dual channel oscilloscope is available, it is best to observe the input and the output signals simultaneously so that the $v_i(f)$ amplitude may be adjusted as f is varied.

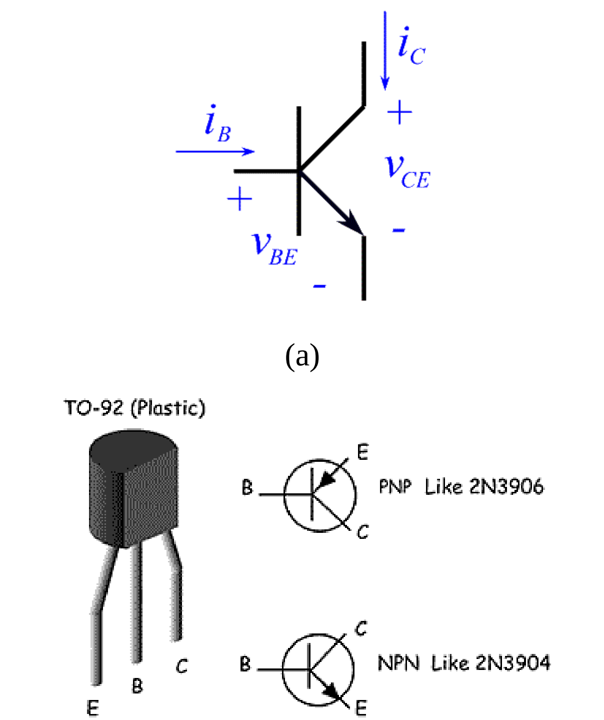
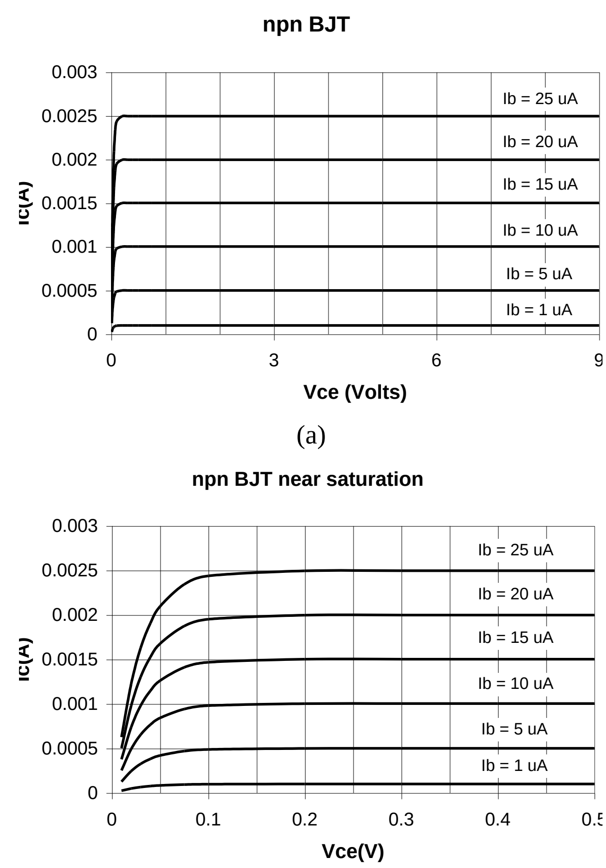


Figure 2.3: (a) npn BJT circuit symbol and (b) pinout http://www.electronicsteacher.com/resources/to-92.gif



(b)

Figure 2.4: (a) Characteristic curves for a typical npn BJT; (b) same curves over a narrow, low range for Vce (near saturation)

2.2 Common Emitter Amplifier

2.2a The BJT

The circuit symbol for an npn BJT (bipolar junction transistor) is shown in Figure 2.3(a), along with the key parameters. A pinout of the BJT is shown in Figure 2.3(b). Consider that v_{BE} represents the total voltage from base to emitter, or $v_{BE} = V_{BE} + v_{be}$, where V_{BE} and v_{be} are the DC and AC portions of the total v_{BE}, respectively. Likewise, $i_C = I_C + i_c$. A typical family of curves for the BJT is shown in Figure 2.4(a), where the collector current i_C is plotted against v_{CE} as a function of the base current i_B. The forward-active mode of operation for an npn BJT occurs when i_C is greater than zero and v_{CE} is greater than a few tenths of a volt. This mode is where we wish to operate the BJT for amplifier applications. In this mode the collector and base currents are related by the forward current gain term β_f, or $i_C = \beta_f\, i_B$. For the 2N3904 npn BJT that we will use for many of our circuits, this gain term varies from part to part over a wide range from 70 to about 300. Henceforth we will drop the f subscript when referring to the current gain.

2.2b The DC biasing circuit

A DC biasing network is required to establish the *Q-point* for the circuit. Q-point (or quiescent point, also called the DC operating point) is optimally located near the center of our forward active region (for instance, referring to Figure 2.4a, at V_{CE} = 4 V and I_C = 1.5 mA). The 4-resistor biasing network, shown in Figure 2.5, is very popular as it stabilizes the Q-point against variations in β (mostly R_e is responsible for this stabilization).

The Q-point (V_{CE}, I_C) for this circuit may be calculated as

$$I_C = \frac{\beta(V_{BB} - V_{BE})}{(\beta+1)R_e + R_{BB}}$$

where

$$V_{BB} = \frac{R_1}{R_1 + R_2} V_{CC},$$

and

$$R_{BB} = R_1 \| R_2$$

Then,

$$V_{CE} = V_{CC} - I_C R_C - \frac{(\beta+1)}{\beta} I_C R_e$$

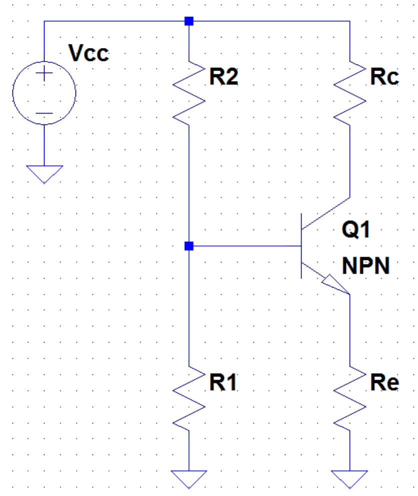


Figure 2.5: 4-resistor bias scheme for a CE amplifier

Exercise 2.1:

Suppose V_{CC} = 9.00 V, R_1 = 6.80 kΩ, R_2 = 30.0 kΩ, R_C = 3.00 kΩ and R_e = 470. Ω. The transistor is assumed to have β = 100., and the on-voltage V_{BE} = 0.700 V. Calculate the Q-point.

Ans: (__________, __________)

Exercise 2.2:

For the circuit values of the previous example suppose β varies from 70 to 300. Calculate the range over which I_C and V_{CE} vary.

Ans: (_________________, _________________)

2.2c Adding the AC components

For use as a small signal amplifier, Figure 2.6 shows how the signal to the CE amplifier is coupled into the base via coupling capacitor C_1, and the output signal is coupled into the load via the coupling capacitor C_2. Notice also the addition of the bypass capacitor C_e. While R_e is needed for Q-point stability, its presence in the AC circuit drastically reduces gain. So we use C_e to bypass, or short out, R_e for the AC circuit.

Consideration must be given to frequency of operation when selecting coupling capacitors. The impedance of a capacitor is given by

$$Z = \frac{-j}{\omega C} = \frac{-j}{2\pi fC}$$

This impedance needs to be low to avoid having a significant drop in signal level across the capacitor.

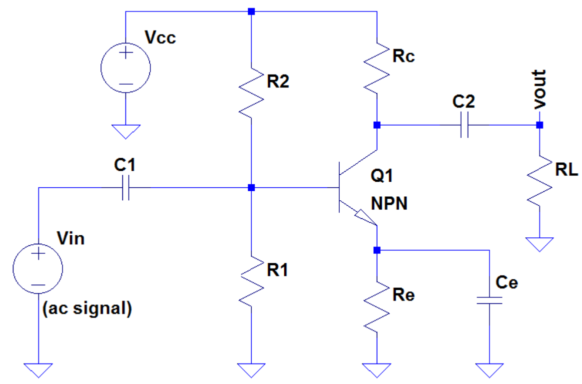


Figure 2.6: Complete common emitter amplifier circuit.

Exercise 2.3:

Calculate the impedance magnitude at 1 kHz for C = 1 pF and 1 μF.

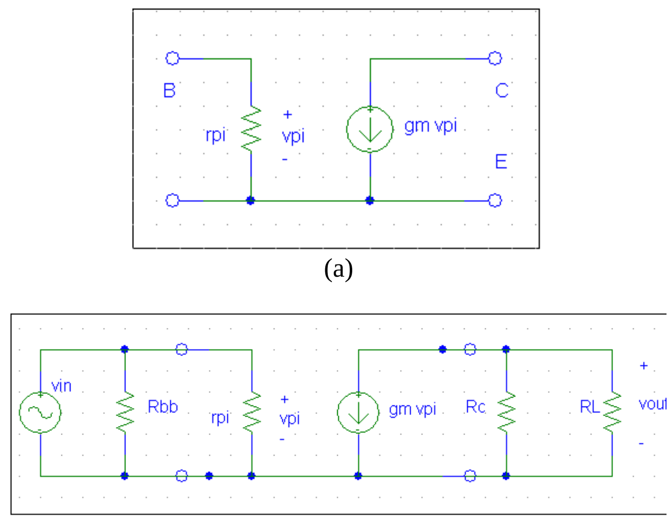
Ans: (___________, __________)

Next we must analyze the *AC equivalent circuit* to determine the gain of the CE amplifier. To create this circuit, we first replace the BJT with an appropriate small signal AC model. One of the best such models is the *hybrid-π* model, a simplified version of which is shown in Figure 2.7(a). Then, the coupling and bypass capacitors of Figure 2.6 are treated as short circuits in the AC circuit. Finally, the DC supply voltage (which can be thought of as a very large capacitor!) is also treated as a short circuit for the AC circuit. These considerations yield the AC circuit shown in Figure 2.7(b). From this circuit, it is fairly easy to determine the mid-band gain of the CE amplifier as approximately

$$A = \frac{v_{out}}{v_{in}} = -g_m\left(R_L \| R_C\right)$$

where transconductance g_m is determined by I_c at the Q-point,

$$g_m = \frac{I_C}{V_T}$$



(b)

Figure 2.7: (a) Small signal AC model for the npn BJT, (b) CE amp model

and V_T is the thermal voltage, a value equal to about 0.026 V at room temperature. The resistance r_π ("rpi") is calculated as

$$r_\pi = \frac{\beta}{g_m}$$

Exercise 2.4:

Assuming β = 100 and R_L = 10 kΩ, calculate the values for g_m, r_π and the mid-band gain for the circuit of Exercise 2.1. For the gain, be sure to denote the units (typically V/V or dB).

Ans: g_m = ___________

r_π = ____________

A = ____________

2.3 LTspice Simulation of the CE Amp

In your next lab section meeting you will assemble the circuit shown in Figure 2.8. This activity will serve as a review of how to breadboard a circuit and reintroduce you to your station's equipment (scope, signal generator, and power supply).

This semester you will be simulating many parts of your AM radio using LTspice, a circuit simulator package similar to Multisim or PSpice. Unlike commercial circuit simulation software where there may be serious operating restrictions on the student version (for instance, in the student version of PSpice you are limited to no more than 10 transistors in a circuit), the full-feature version of LTspice is freely available.

This section will serve as an introductory session on LTspice, in addition to getting you familiar with the common emitter amplifier. You will assemble a brief report to hand in to your GTA prior to the beginning of the hardware portion of this lab. Particular items to turn in are indicated in the steps to follow.

0. Follow instructions provided by the course instructor to install LTspice from: http://www.linear.com/designtools/software/

- Your brief Word document report should have images taken from your LTspice work. Use the "tools" menu to capture circuit schematics and plots to embed into your Word document.
- Also play around with the "tools" menu item "color preferences". For embedding figures into a Word document, the format of Figure 2.12 is **much better** than a figure with a dark background. Do not use a dark background for any work you will turn in.

1. Consider the circuit shown in Figure 2.8. We will assemble this circuit in LTspice.

- Figure 2.9 shows some of the important symbols on the LTspice menu bar.
- Place a resistor by selecting the resistor symbol, place it on the schematic and click, then right click on the name (default "R1") to change the name to, perhaps, "Re". Right click on the value (default "R") to change the value to 150.
- Place an npn transistor by selecting the component symbol, choosing "npn" from the list, and placing this on the schematic. Right click on the part to choose "pick new transistor" and select the 2N3904.

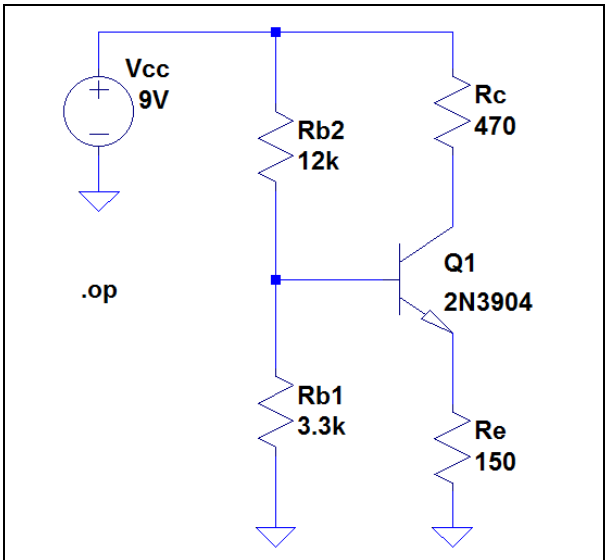


Figure 2.8: CE amp DC circuit in LTspice

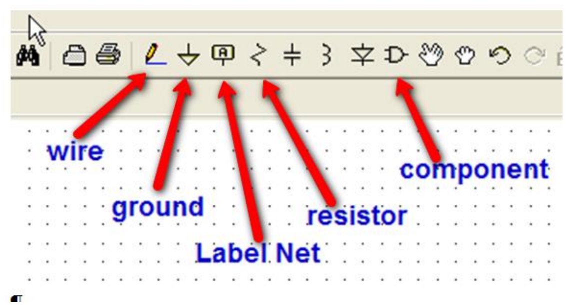


Figure 2.9: close-up of part of the LTspice menu bar.

- Assemble the rest of the circuit as shown and copy the schematic for your report.

2. In the Simulate pulldown menu, select 'Edit Simulation Cmd.' Choose 'DC Op Pnt' and place this (shown as ".op") somewhere on your schematic.

- Simulate by clicking on the running man icon. A list appears showing you the voltages and currents for your circuit. Close this window, and move your cursor around the circuit. Notice the information given at the bottom of the window. It will display voltages, currents, and even power dissipated in the device.
- Record the voltage at the base, collector and emitter of your BJT:

V_B = ________

V_C = ________

V_E = ________

- At this point you should see a V_{BE} (or $V_B - V_E$) of about 0.6 to 0.7V, and a V_{CE} of a few volts.
- Place the cursor on the power supply. Note the direction of the current measurement. If it is measuring the current opposite to the direction of the current flow it may calculate a negative P_{diss}. What is the total power dissipated by your circuit?

P_{diss} = ____________

3. You can modify your circuit to resemble Figure 2.10. Here, the "Label Net" tool is used for applying the DC voltage. This can simplify schematic wiring for larger circuits.

4. Now add the components necessary to form the AC circuit schematic shown in Figure 2.11.

- For the signal source, place a voltage source and then right click on it. Select "Advanced" and then choose a "sine" source. Set amplitude to 20mV (or .020V) and frequency to 1kHz.
- Use the "Label Net" tool to indicate Vin and Vout at places indicated on your circuit in Figure 2.11.
- When simulating the circuits, note the DC voltages on either side of the capacitor (a good idea is to write this on your circuits in your lab manual or prelab to save time in lab). The capacitors used in the lab are electrolytic and need to be placed with the correct polarity to prevent rupturing.

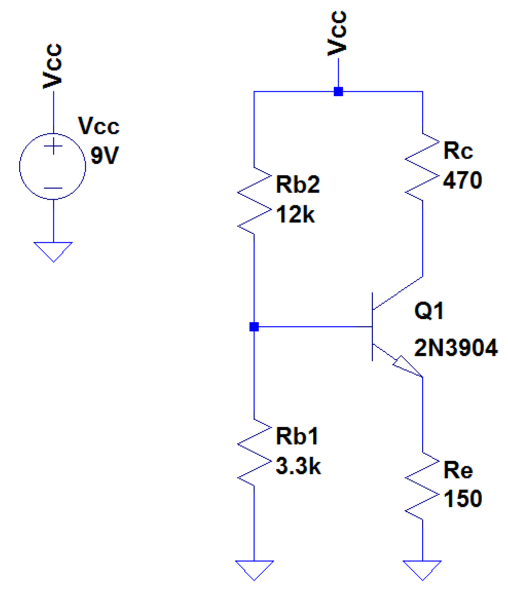


Figure 2.10: alternative configuration of the DC circuit

5. Now set the simulator and observe the transient simulation results.

- In the simulate menu, select "edit simulation CMD" and then choose "transient".
- You can choose stop time at about 4 ms to give several wavelengths of output. Do this and place the transient command on the schematic (indicated on Figure 2.11 as ".tran 4ms"). Press the simulate button (running man).

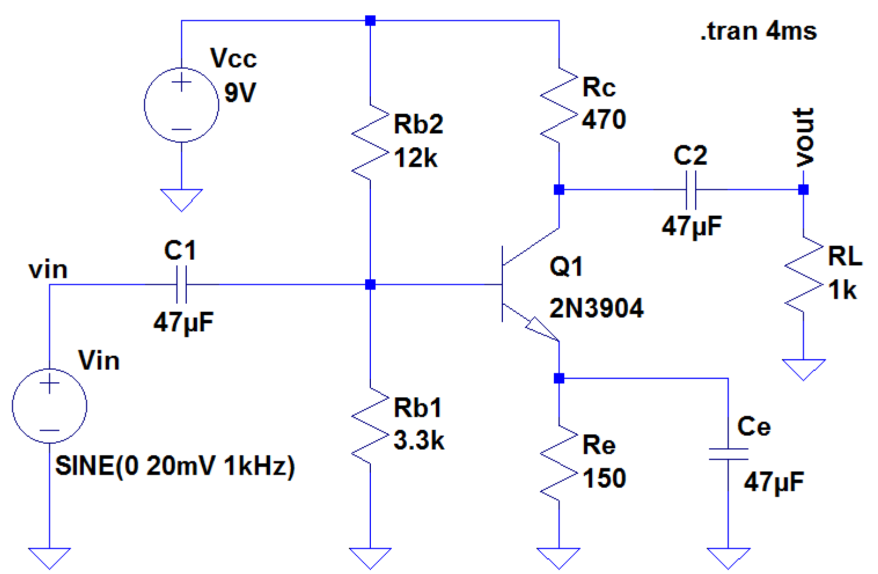


Figure 2.11: CE amp with AC circuit added

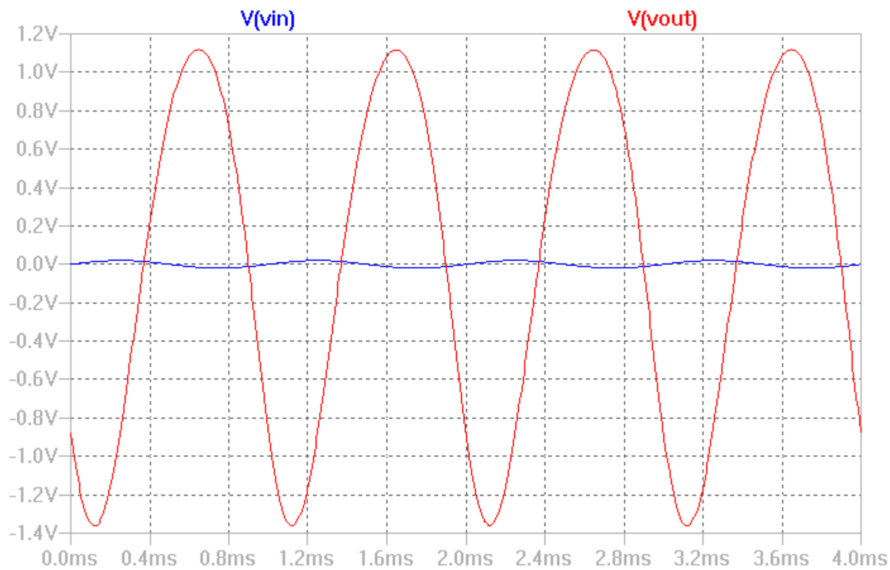


Figure 2.12: Transient analysis plot of the input and output voltage for the CE amp.

- I find the results easiest to see, along with the schematic, if the windows are tiled vertically (window menu, select *tile vertically*).
- When you move the cursor over the schematic, it will turn into a probe icon. Move this icon over vin and click; repeat for vout. You should see a plot similar to Figure 2.12.
- Notice that moving the cursor around the output window, the x-y coordinates (corresponding here to time and voltage) appear at the bottom of the LTspice window. This feature can be quite handy for estimating amplifier gain.
- It is useful to be able to zoom in or out of a particular part of your circuit. Within the output window, right click to select 'zoom area.' Or, you can left click and form a box that will cause a zoom in to occur. Play around with this.
- Notice the first peak appears a bit higher than the ones to follow. This is a result of seeing the amplifier in action before it has had a chance to "warm up". Instead, set the stop time at 8ms and the time to start saving data at 4 ms. Place this new simulation command in the schematic window and run another transient simulation. The result is shown in Figure 2.12.

6. What is the gain of your amplifier? Use the cursor and zoom-in feature to find the amplitude v_{out} and the amplitude v_{in}:

$$Gain = \frac{v_{out}}{v_{in}} = \underline{\qquad\qquad} = \underline{\qquad\qquad}\left(\frac{V}{V}\right)$$

7. Now let's create a Bode plot like the one shown in Figure 2.13. In the previous circuit for the CE Amplifier, right click on the Vs source, and under "small signal AC analysis", enter 10mV. In the 'Edit Simulate CMD', select AC analysis with the following parameters:

Type of sweep:	decade
Number of pts per decade:	101
Start Freq:	10
Stop Freq:	100meg

Place this simulate command in your schematic, replacing the previous one. Now simulate! The generated Bode plot has a voltage in dB, relative to 1 V. Right click on the "V(vout)" at the top of the plot and enter the following equation:

20*log(V(vout)/V(vin))

This will give the amplifier gain in dB.
Display the results in your report.

8. Based on your Bode plot, what is the simulated 3 dB bandwidth ("half-power bandwidth") for your amplifier?

BW = ______________

9. Enhance Your Understanding!
A way to better your understanding of the circuit is to make changes and see what happens.

a. Remove Ce, the 'bypass capacitor', from the circuit of Figure 2.11 and re-simulate. What happens?

b. Return to the default circuit of Figure 2.11. Swap Rb1 and Rb2 and re-simulate (you may also want to look at the DC operating point). What happens?

c. Return to the default circuit. Change the input and output coupling capacitors (C1 and C2) to 1nF ("one nanofarad"). What happens?

d. Return to the default circuit. Remove the transistor. Now re-insert it, using "CTRL-R" to rotate the part and "CTRL-E" to get a mirror image. In this way, flip the transistor (the collector and emitter) in Figure 2.11. Then, "pick new transistor" as 2N3904. What happens when you run the simulation?

10. (Optional-Advanced) Guided Design: Maximize Unclipped Output

a. Change the input amplitude of the circuit in Figure 2.11 from 20 mV to 50 mV. Resimulate and show the transient analysis plot. Repeat with an input amplitude of 100 mV. Reflect on the results.

b. Devise a way to adjust the circuit in Figure 2.11 to provide the "cleanest" (least distorted), largest amplitude sinusoidal output possible for a 100 mV input amplitude using parts available in your kits. The intent of this advanced section is to test your design in lab. Some questions for you to ponder:
   a. How do you measure or quantify a "clean" signal?
   b. How do you characterize the amplifier gain with a distorted signal?
   c. What could be done to "balance" the output, where the peak is above zero by about the same amount that the dip is below zero?

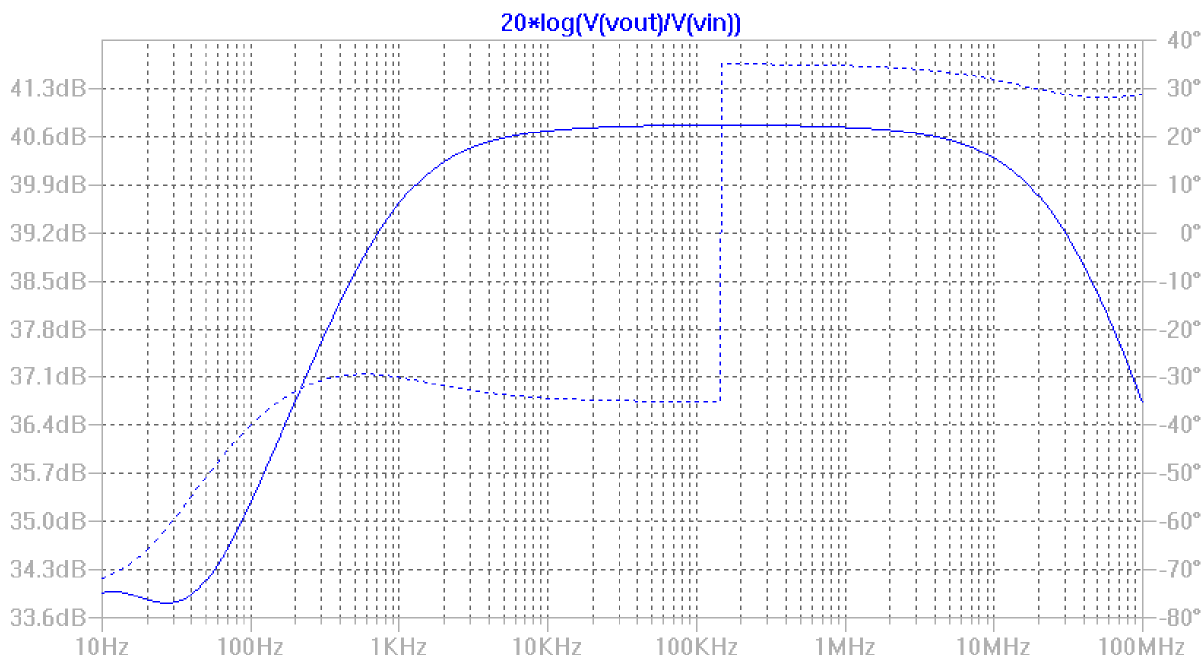


Figure 2.13: Bode plot (magnitude and phase) for the CE amp

2.4 Build and Test a Common Emitter Amplifier

1. Breadboard the DC circuit portion (Figure 2.8) of your CE amp.

- Use the DMM to probe voltage at your transistor's base, emitter and collector. The results will give you a good indication of whether the transistor is operating in the *forward-active mode* ($V_{BE} \approx 0.6$ V– 0.7 V and $V_{CE} > 0.2$ V) and whether the DC circuit is assembled properly.

V_B = ________ $V_{BE} = V_B - V_E$ = _________

V_C = ________ $V_{CE} = V_C - V_E$ = _________

V_E = ________

2. Now add the AC circuit to your breadboard including the 1 kHz AC input signal (see Figure 2.11).

- **Carefully observe the polarity of your electrolytic capacitors!** The most positive voltage goes on the positive terminal of the capacitor. If you are not sure which side will be most positive, LTspice simulation can help you. **Putting the wrong voltage polarity across a capacitor can cause catastrophic failure to that part.** See this video for fun:

https://www.youtube.com/watch?v=77ARIvt9RrE

- Set-up the function generator to output a 1 kHz sine wave with a 10 mV amplitude (20 mV Vpp)
- Use the dual mode capability of your oscilloscope to view both the input and the output signals at the same time. **Don't forget to take pictures and screenshots for your eportfolio!**

3. Determine the amplifier voltage gain (ratio of output voltage amplitude to input voltage amplitude). Record this gain in Table 2.1 for R_L = 1 kΩ.

4. Now let us see how the gain is influenced by the choice in load resistance. Measure the gain for a series of different values of load resistance, starting with high resistance (R_L = 100 kΩ) and working down to low resistance (R_L = 10 Ω or so). *(The resistor color code is given in the appendix).*

- Record your data in Table 2.1. Write in the actual resistance values used if other than the suggested values shown.
- Use the data of Table 2.1 to plot gain vs resistance in Figure 2.14.
- Consider that the audio amplifier will be driving an 8 Ω impedance speaker. How do you think the CE amp will perform driving this particular load?

Table 2.1: Gain compared with load resistance

R_L(Ω)	Voltage gain (0.5 – 200 typical)
10	
100	
1 k	
10 k	
100 k	

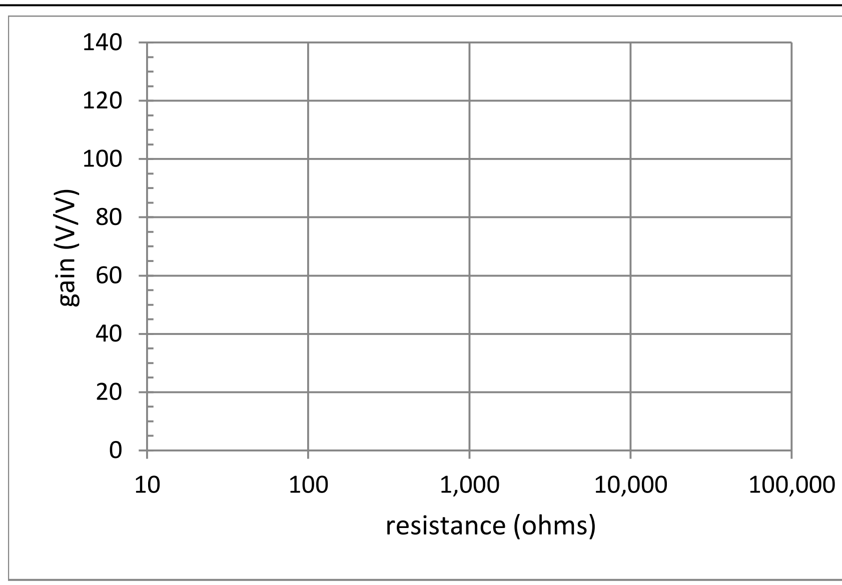


Figure 2.14: Blank Gain(V/V) vs resistance (Ω) for plotting Table 2.1 data

5. Bode Plot!
   Replace the load resistance with its default value of 1 kΩ. Now you will measure the gain at different frequencies. Obtain data and fill in Table 2.2.

- Gain in dB is 20*log (v_o/v_{in})
- As you change frequency, you will need to change the time base on your oscilloscope. Also, the input amplitude may require fine adjustment to maintain a constant value for use in calculating gain. You may alternatively use the spectrum analyzer function to measure these values if you would like.

6. Plot the Gain (dB) against frequency data of Table 2.2 to create a Bode Plot in Figure 2.15.

7. The '3 dB bandwidth', also called the 'half-power bandwidth', is the frequency range shown on your Bode plot over which the gain is within 3 dB of the peak gain. From your plot, estimate the 3 dB bandwidth:

   BW = _______________

Table 2.2: Bode Plot Data for CE Amp

Frequency	Voltage gain (5 – 150 typical)	Gain(dB)
100 Hz		
300 Hz		
1 kHz		
3 kHz		
10 kHz		
30 kHz		
100 kHz		
300 kHz		
1 MHz		
3 MHz		
10 MHz		

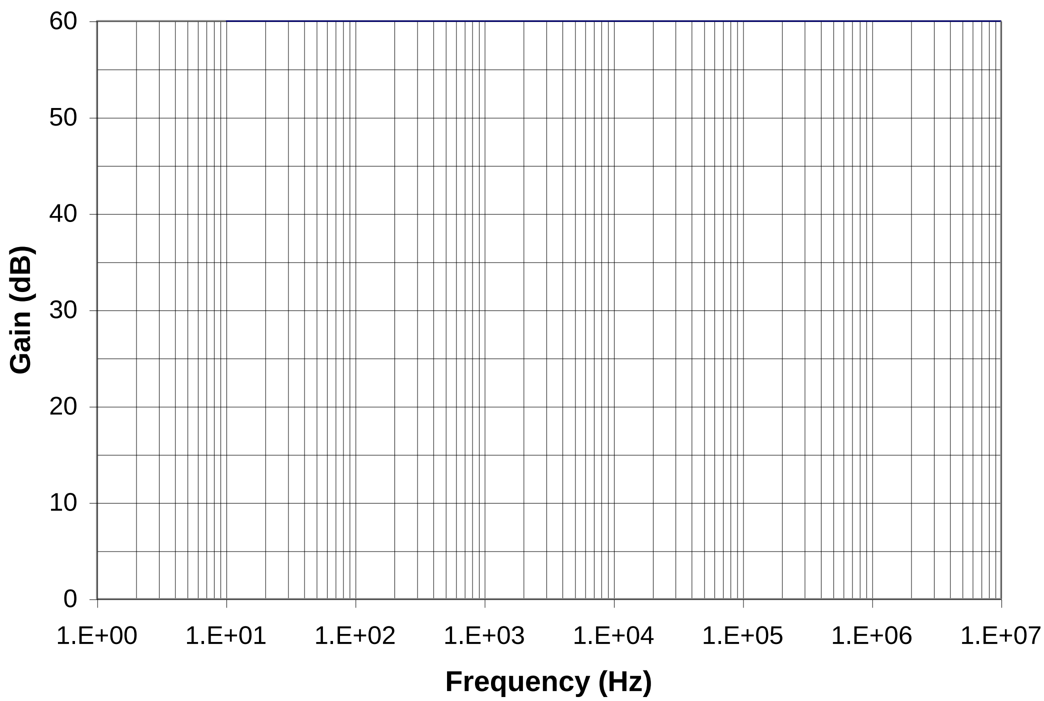


Figure 2.15: Blank Gain(dB) vs frequency (Hz) for plotting Table 2.2 data

8. (Optional-Advanced) Guided Design:
   Maximize Unclipped Output
   a. Build the default circuit with a 100mV amplitude input signal. Get a good picture of the output on your scope.

   b. Build the version (or versions) of your circuit that you designed for reducing the distorted output. Get a good picture of the appropriate outputs.

   c. What conclusions can you draw about your design? What would you do next in your quest to achieve a cleaner output without sacrificing gain?

   d. Make sure you incorporate and reflect on this guided design experiment in your ePortfolio.

(notes)