EE320L Electronics I

Laboratory

Laboratory Exercise #8

Amplifiers Using BJTs

By

Angsuman Roy

Department of Electrical and Computer Engineering

University of Nevada, Las Vegas

Objective:

The purpose of this lab is to understand how to design BJT amplifiers in the common-emitter, common-base and common-collector configurations. These single stage amplifiers will be designed and analyzed from an applications perspective rather than a detailed theoretical perspective.

Equipment Used:

Dual Output Power Supply

Oscilloscope

Breadboard

Jumper Wires

Resistors, Capacitors

NPN and PNP transistors (2N3904/2N3906 or similar)

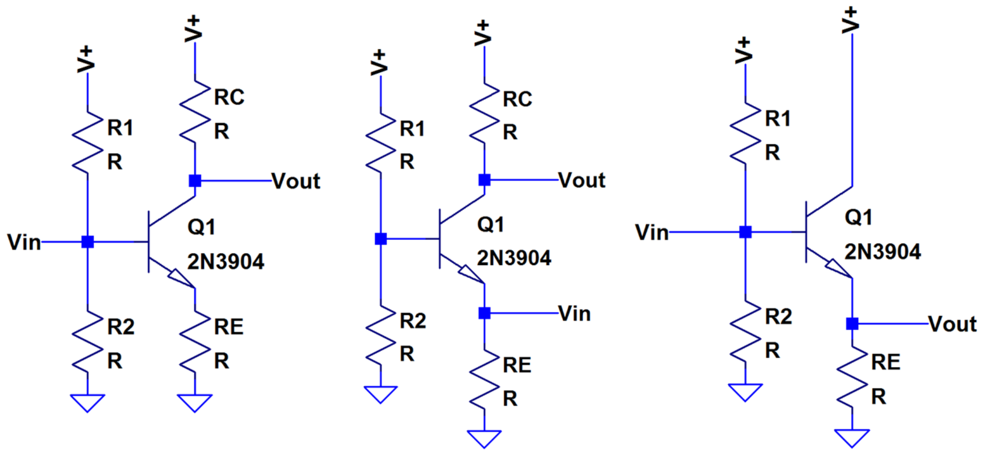
10x Scope Probes

Background:

A long time ago, before the invention of and proliferation of easy to use op-amps, electrical engineers had to build amplifiers out of discrete transistors, resistors and capacitors. While this is seldom done anymore except for specialized applications involving low-noise or RF, it is important to be able to design single-stage amplifiers to do get a good understanding of circuit design fundamentals. Furthermore, a good foundation with discrete amplifiers is essential if one wishes to pursue a career as an IC design engineer or an RF engineer.

In this lab the scope will be limited to just bipolar-junction transistors for the sake of time. Although the vast majority of integrated circuits are designed in CMOS, most applications requiring a discrete amplifier use BJTs. This is due to the fact that most discrete MOSFETs are optimized for switching applications (although they can still be used as linear amplifiers). BJTs on the other hand are more popular as amplifiers. Instead of discussing the physics and operation of BJTs, this lab will pose three circuit problems and then attempt to answer them with BJTs.

The three types of BJT voltage amplifiers are the common-emitter, common-base, and common-collector amplifiers. The word common means that both the input and output share that particular node. The common-emitter and common-base amplifiers have voltage gain. The input signal voltage is multiplied by the gain of the amplifier at the output. The common-collector amplifier does not have voltage gain. Rather, the output *follows* the input which gives rise to the more popular name, the emitter-follower. The emitter-follower is used to drive a load that could otherwise not be driven by the signal source. The three topologies are shown below in Fig. 1.



Common
Emitter
Amplifier

Common
Base
Amplifier

Common
Collector
Amplifier
(Emitter
Follower)

Figure 1. Three main types of BJT voltage amplifiers.

Solved Example Problem #1: Common-Emitter Amplifier

It is a good idea to explore more detailed properties of these amplifiers through the use of examples. Suppose that a basic audio amplifier is needed with a gain of 10. Suppose that the output impedance of the source is 16 ohms. This is a fairly typical value for the headphone jacks of mobile phones. We would like the input impedance of our amplifier to be at least ten times (or more) higher than the output impedance of our source. The output of this amplifier will be connected to a 10k load and the power supply voltage will be 12V. The amplifier will be AC-coupled for both ease of use and to isolate the mobile device from the DC power supply. Since this is an audio-amplifier the bandwidth should be flat from 20 Hz-20 kHz. There should be no problem reaching the 20 kHz upper limit with standard small-signal BJTs. However the 20 Hz lower limit poses a problem due to the high-pass filter formed by the coupling capacitor and the input impedance of the amplifier. This is more apparent from looking at the AC-coupled common-emitter amplifier shown in Fig. 2.

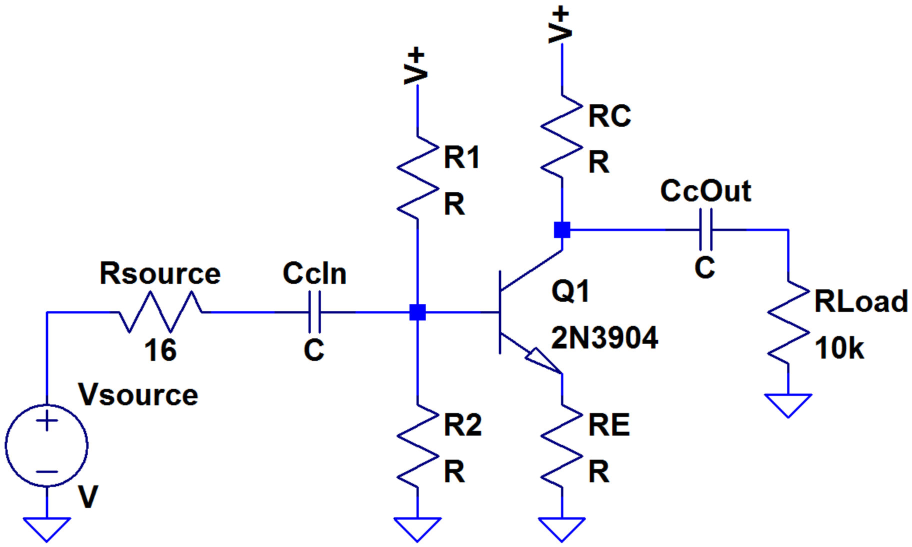


Figure 2. AC-coupled common emitter amplifier.

The first task is to pick our collector resistor. Assuming that the output resistance of the transistor is very high, the collector resistor is responsible for setting the output resistance. Using the rule of thumb that our output resistance should be at least 10 times smaller than our load resistor, results in a collector resistor value of 1k. Although smaller values can be used, the current flow through the amplifier will become excessive. Next, the collector current needs to be set. To maximize signal swing, the collector should be biased at half of the supply voltage which is 6V in this case. This allows the output to swing symmetrically from 6V up to 12V and from 6V down to 0V. The current that results in 6V dropped across the collector resistor is (6V/1k=6 mA). The next steps involve setting 6 mA to flow in the collector. The impedance seen at the emitter is equal to the inverse of the transconductance, gm.

$$g_m = \frac{I_{Collector}}{V_{thermal}} = \frac{6mA}{26mV(at\ room\ temp)} = 0.23\ S \to 4.3\ \Omega\ seen\ at\ the\ emitter$$

The voltage gain of the amplifier is given by the resistance in the collector divided by the resistance in the emitter. The resistance in the emitter is the sum of the emitter resistor and the emitter resistance which is the inverse of the transconductance. In many cases the emitter resistance is small enough that it can be neglected.

$$A_V = -\frac{Resistance\ in\ Collector}{Resistance\ in\ Emitter} = -\frac{Rc||Ro}{Re + \frac{1}{gm}} \approx -\frac{Rc}{Re}$$

To get a gain of -10, the emitter resistor should be 100 ohms. Since this is much larger than the 4.3 ohm emitter resistance, we can neglect that resistance. However for accuracy, this term can be included although temperature effects are probably too great for it to matter. The common-emitter amplifier is an inverting amplifier. The output is inverted by 180 degrees with respect to the input.

The next step is to set up the biasing circuit. At our desired collector current of 6 mA, the emitter voltage is 6 mA * 100 **Ω**=0.6V. We are ignoring the base current flowing into the emitter in this example since it is 1/100th of the collector current. The base voltage needs to be a diode drop higher than the emitter voltage. A good rule of thumb voltage is 0.7V. This results in a base voltage of 0.6V+0.7V=1.3V. The voltage divider formed by R1 and R2 is used to set the voltage at the base to 1.3V. Recall that a voltage divider's output will change when it is significantly loaded down. The problem with a BJT is that the base draws current. The amount of current the base draws depends on β. The relationship between collector current and base current is,

$$I_b = \frac{I_C}{\beta}$$

The problem is that the beta value can vary significantly and should not be relied upon for good design. Rather, a minimum expected beta value is selected for worst case design. The datasheet for the 2N3904 indicates a minimum beta value of 100 for a collector current of 10 mA. We will select this as our minimum beta value. Since beta changes with collector current, it is a good idea to check the datasheet and adjust the minimum beta value accordingly when operating at very low or very high currents. For our 6 mA collector, the base current will be 60 uA. The voltage divider formed by R1 and R2 needs to be designed such that the voltage remains at 1.3V even with 60 uA being drawn from it. If the values of R1 and R2 are too high, then the base current will cause the voltage to drop significantly below 1.3V. On the other end, if R1 and R2 are very small values, the voltage will remain at 1.3V but the divider will waste a lot of current. For this reason a good rule of thumb is that the divider should allow for 10 times the base current to flow through it, or 600 uA in this case. One reason for using 10 times the base current instead of a larger number such as 25 or 50 times the base current is to offer a form of bias stabilization. Suppose that the collector current and consequently the base current doubled due to temperature effects, the transistor could destroy itself because the resistive divider is able to provide the necessary base current for this situation. By using 10 times the base current for the divider current, a sudden large current draw into the base would drop the voltage at the divider which would reduce the collector current until it stabilized back to the set value. The emitter resistor also helps regulate the collector current. A sudden increase in collector current would increase the voltage drop across the emitter resistor and decrease the Vbe which would then reduce the collector current back to the set value. Using this knowledge, we can set R1 and R2. For a 12V supply, R1+R2 needs to equal 20k to set 600 uA through the divider (12V/600uA=20k). R2 should be set to 2.2k to set 1.3V at the base (1.3V/600 uA=2.2k). Finally R1 is 20k-2.2k=17.8k.

The final task is to set the coupling capacitors to ensure that our bandwidth requirements are met. The input and output coupling capacitors form a high-pass filter. Let's set both corner frequencies to 10 Hz to preserve the full audio bandwidth. The input corner frequency is set by the input coupling capacitor and the parallel combination of R1, R2 and the "reflected emitter resistance".

$$f_{Cin} = \frac{1}{2\pi C_{in} * \left(\left(R1||R2||Re_{ref}\right) + R_{source}\right)} = 10Hz$$

$$= \frac{1}{2\pi C_{in} * \left(\left(17.8k||2.2k||10.5k\right) + 16\right)} \rightarrow C_{in} = 9.6uF$$

The reflected emitter resistance is the input impedance of the base. This term should not be neglected because it is fairly close to the values of our biasing resistors. Any resistances in the emitter of the BJT are reflected to the base by the beta value. That is,

$$Re_{ref} = (\beta + 1) * \left(Re + \frac{1}{gm}\right) = 101 * (100 + 4.3) = 10.5k$$

The input impedance of our amplifier is the parallel combination of all the resistors connected at the input,

$$R_{in} = R_1||R_2||Re_{ref} = 1.66k$$

This exceeds our requirement that the input impedance is at least 10 times the 16 ohm source impedance of our mobile phone headphone jack. The output corner frequency is given by,

$$f_{Cout} = \frac{1}{2\pi C_{out} * (R_C||R_L)} = 10Hz = \frac{1}{2\pi C_{out} * (1k||10k)} \rightarrow C_{out} = 17.5uF$$

The final schematic is shown below in Fig. 3. The frequency response is shown in Fig. 4. The corner frequency is around 8.6 Hz which is fairly close to our 10 Hz goal. The output of the circuit with a 1 kHz 100 mV input is shown in Fig. 4. It is apparent that the gain appears to be only 8.5 instead of 10. This is mainly due to the setting the collector resistor to be 1/10th of the load resistance. Increasing the load resistor to 100k causes the gain to be 9.5, confirming this suspicion. The remaining lost gain can be attributed to neglecting the inherent output resistance of the transistor. If a higher final gain is desired, the analysis can be repeated with higher collector currents and a smaller collector resistor value. However, it is important to understand that it is difficult to set a precise gain with single stage transistor amplifiers, especially in the lab with 5% tolerance components.

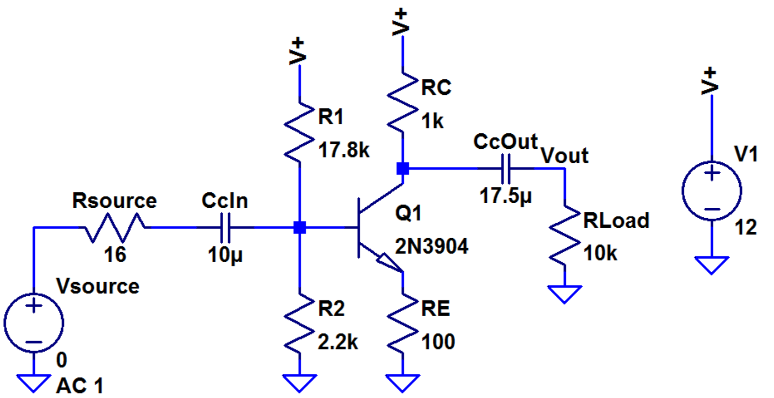


Figure 3. Final schematic of common-emitter amplifier.

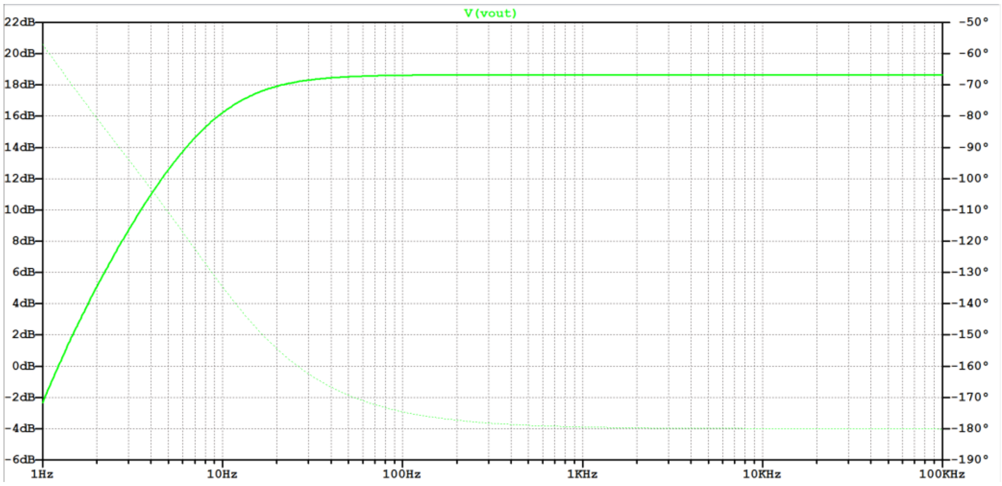


Figure 4. Frequency response showing -3dB frequency of 8.6 Hz.

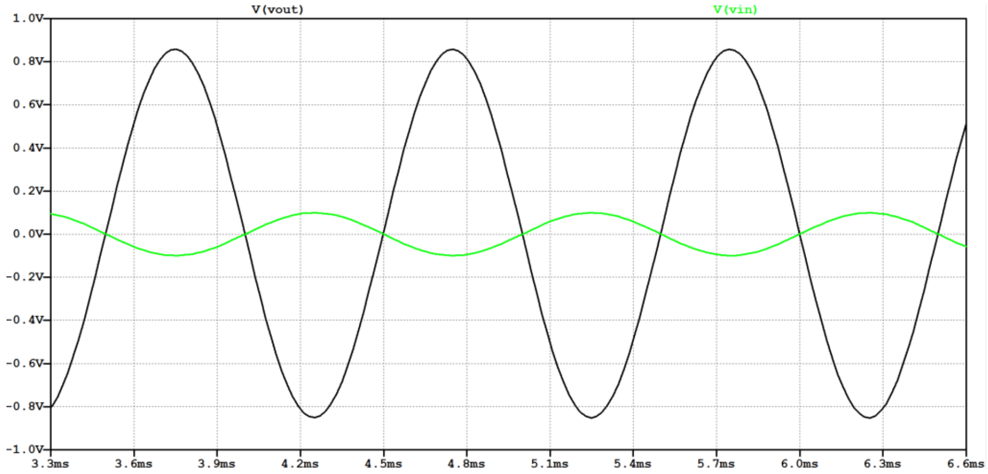


Figure 5. Transient analysis with a 100 mV 1 kHz input signal.

Before continuing with other examples, a small detail regarding the reflected emitter resistance should be clarified. In this example the reflected emitter resistance lumped both the emitter resistance due to the transconductance and the emitter resistor external to the transistor as shown in,

$$Re_{ref} = (\beta + 1) * \left(Re + \frac{1}{gm}\right)$$

Many textbooks separate these two terms and define the reflected emitter resistance with only external emitter resistor,

$$Re_{ref} = (\beta + 1) * (Re)$$

The remaining term is defined as Rπ which is the resistance seen from the base to the emitter of the transistor,

$$R_{\pi} = (\beta + 1) * \frac{1}{gm}$$

The input impedance of the transistor amplifier is then defined as,

$$R_{in} = Re_{ref} + R_{\pi}$$

Both methods are equivalent but it is easier to remember all the resistances in the emitter are reflected to the base by $\beta + 1$ rather than two distinct resistance terms that both contribute to the input resistance.

For the sake of completeness, we will briefly repeat the previous example but with a PNP transistor. Students are often afraid of PNP devices but all the same principles apply. Often, PNP parameters are given as negative values. For example, Vbe=-0.7 meaning that the base is 0.7V lower than the emitter. In the following examples we will avoid negative values and simply define Veb=0.7V. This means that the emitter is 0.7V higher than the base. For example if the emitter is connected to a 12V supply, the base will be at 11.3V. This convention is more modern because most circuits are single supply and CMOS design uses the same convention.

A PNP common emitter amplifier is shown below in Fig. 6. Notice that everything is the same except that the emitter and collector are flipped and so are RC and RE. The output is taken from the collector.

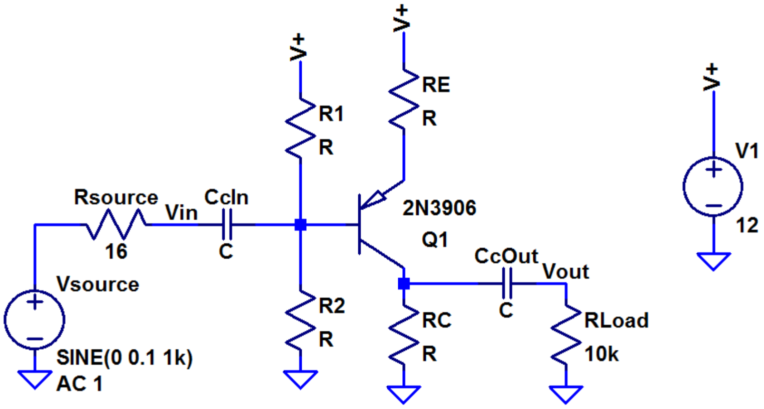


Figure 6. PNP CE Amplifier.

Using the same requirements let’s calculate all the parameters.

$$RC = \frac{1}{10} * RL = 1k$$

Setting the collector to 6V (half of 12V) supply requires 6 mA to flow through the 1k resistor. For a gain of 10, the emitter resistor should be 100 ohms. The voltage at the emitter is,

$$V_E = 12V - (6mA * 100) = 12V - 0.6V = 11.4V$$

$$V_{EB} = 0.7V \rightarrow V_B = 11.4V - 0.7V = 10.7V$$

We need to set the voltage divider to be 10.7V and set the divider current to 10 times the worst case base current draw. Assuming minimum β=100,

$$I_B = \frac{I_C}{\beta} = \frac{6\ mA}{100} = 60\ uA$$

$$I_{Divider} = 10 * I_B = 600\ uA$$

With a 12V supply, R1+R2=20k (12V/600uA=20k). We need R2 to develop 10.7V across it when 600 uA flows through it,

$$R_2 = \frac{10.7V}{600uA} = 17.8k$$

R1 is the remainder, 20k-17.8k=2.2k. At this point it is obvious that the circuit is the same except the resistor values are flipped from the NPN version. Due to this, we can assume that the input and output coupling capacitors will be the same since the equivalent resistances with which they form their time constants are the same as the NPN version. Figures 7, 8 and 9 show the final schematic, frequency response and transient analysis respectively.

If the amplifiers are largely identical, what is the rationale for using one over the other? Generally, NPN devices are faster while PNP devices have lower noise. Of course this statement is a generalization and there are many exceptions. The bigger reason depends on signal swing. Because of the saturation voltage an NPN amplifier can swing from the positive supply rail to within 0.2V of ground plus the voltage drop across the emitter resistor. If it is desired to swing all the way to the positive supply rail an NPN amplifier should be used. Conversely if it is desired to swing all the way to ground, a PNP amplifier should be used. The signal swing limits can be defined with the following equations,

$$V_{swing\_NPN} = V_{+rail}\ to\ \left(V_{CE_{sat}} + R_e * I_e\right) \approx V_{+rail}\ to\ (0.2 + R_e * I_e)$$

$$V_{swing\_PNP} = ground\ to\ \left(V_{+rail} - V_{CE_{sat}} - R_e * I_e\right) \approx ground\ to\ (V_{+rail} - 0.2 - R_e * I_e)$$

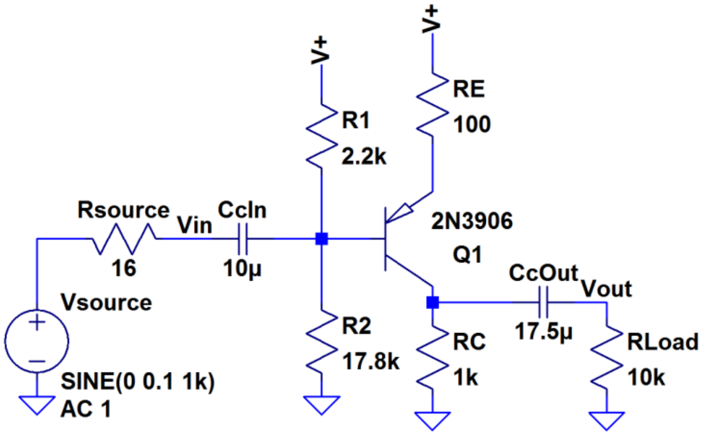


Figure 7. Final PNP CE amplifier schematic.

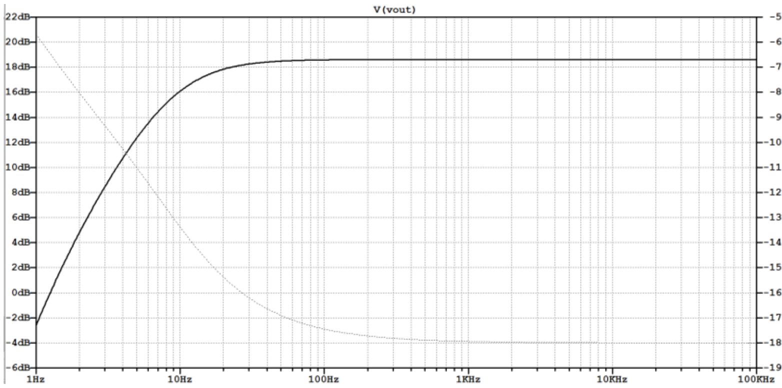


Figure 8. Frequency response of PNP CE amplifier.

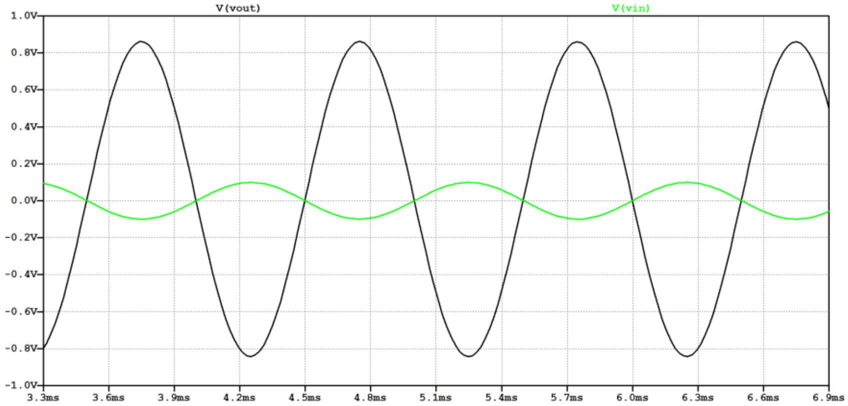


Figure 9. Transient analysis of PNP CE amplifier.

Solved Example Problem #2: Common-Base Amplifier

The common-base amplifier is identical in structure to the common-collector amplifier except that the input signal is applied to the emitter. The input impedance looking into the emitter is quite low, at 1/gm. What is the reason for having a low input impedance for a voltage amplifier? Many systems require impedance matching, such as 50-ohm RF systems and 75-ohm video or networking systems. Cable TV coax cables in our homes have 75-ohm characteristic impedance. Transmission line theory is outside the scope of this lab, but we don't need it to be able to design an amplifier that is compatible within a matched impedance system.

Suppose that we want to make an AC-coupled common-base amplifier with a minimum gain of 50, 75-ohm input impedance and a lower frequency -3dB limit of 10 kHz driving a 100k load. The supply voltage is 12V. This amplifier without values calculated is shown in Fig. 10. Note that the input signal is AC coupled to the emitter of the transistor and the output signal is AC coupled from the collector of the capacitor. The common-base amplifier is a non-inverting amplifier. To intuitively verify this, consider a positive going input signal applied to the emitter. The base voltage is fixed by the voltage divider and therefore Vbe is reduced. A reduction in Vbe reduces the collector current which reduces the voltage drop across the collector resistor resulting in the collector voltage to increase. Since an increase in input (emitter) voltage caused an increase in output (collector) voltage, the amplifier is noninverting.

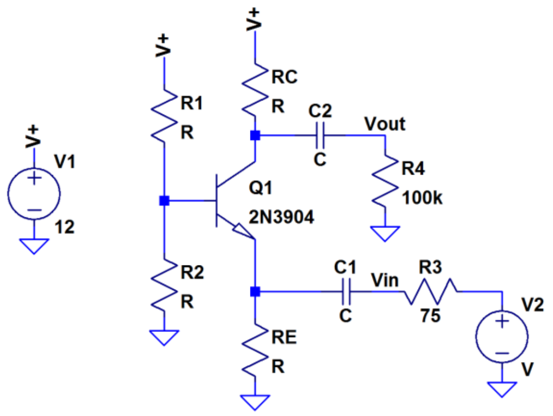


Figure 10. NPN CB amplifier.

The first step to solving our problem is to set the collector current because this directly sets the impedance seen at the emitter. For a 75 ohm input impedance,

$$\frac{1}{g_m} = \frac{V_{thermal}}{I_{Collector}} = \frac{26mV}{I} = 75 \rightarrow I = \frac{26mV}{75} = 346\ uA$$

This is quite a small value of current and will limit the load driving ability of the amplifier. Next, let's calculate the collector resistor to result in a voltage drop of 6V which is half the supply voltage.

$$R_C = \frac{6\,V}{346\,uA} = 17.3k$$

Immediately we should notice that this value is not small compared to the 100k load resistance. Since it is almost 20% of the load resistance, we should expect some attenuation in the final circuit although in this circuit the goal is to achieve a minimum gain. As mentioned before single-stage amplifiers are not a good way to get an exact gain. However, we could reduce the value of the collector resistor to 10k if desired with the trade-off that the signal swing and gain will be reduced.

The next step is to calculate the gain. This is the biggest difference between the common-base and common-emitter amplifiers. The gain doesn't simplify down to RC/RE. Instead, the gain simplifies down to RC/(1/gm). This is because the same current that is fed into the emitter is also flowing into the collector. Since the collector resistor is much higher than the emitter's 1/gm the gain for a common-base amplifier tends to be very high.

$$A_V = \frac{Rc||Ro}{Re||\frac{1}{gm} + R_s} \approx \frac{Rc}{\frac{1}{gm} + R_s} = \frac{17.3k}{75 + 75} = 115$$

The source resistance significantly affects the gain in a common-base amplifier unlike in a common-emitter amplifier. Selecting the emitter resistor depends on the maximum input signal and bias stability. The emitter resistor, assuming it is large compared to the (1/gm) term, does not affect the gain. Instead, the purpose of Re is to set the operating voltage of the emitter. Suppose that the maximum input signal from the source would be 1V. The emitter resistor would need to be sized such that the voltage developed at the emitter is 1V with no input signal. This would allow a 1V AC input signal to swing the emitter down to 0V and up to 2V. Of course, with a gain of 115 as calculated above, only a 50 mV input signal would cause our amplifier to clip. Instead the main concern is bias stability. Higher emitter resistor values contribute to more stable biasing with regard to variations and temperature effects. At the same time, a high value emitter resistor will limit the amplifier's negative going signal swing. For this example, we will arbitrarily pick a emitter resistor value that is 1/10th of the collector resistor which is 1.7k. The voltage at the emitter is,

$$V_e = 346\,uA * 1.7k = 0.59V$$

The base voltage should be 0.7V higher than this which is 1.29V. The voltage divider at the base should result in 1.29V. Assuming a β of 100, the base current should be around 3.46 uA. Using the 10 times base current rule results in (R1+R2=346k). With a divider current of 34.6 uA, R2 is equal to 37.2k (1.29V/34.6uA). The R1 is the remainder or 308k (346k-37.2k).

Finally, let's calculate the coupling capacitors. The input coupling capacitor can be solved by,

$$f_{Cin} = \frac{1}{2\pi C_{in} * ((Rb_{ref}||Re||\frac{1}{gm}) + R_{source})} = 10kHz$$

$$= \frac{1}{2\pi C_{in} * (((\frac{R1||R2}{\beta})||Re||\frac{1}{gm}) + R_{source})} = \frac{1}{2\pi C_{in} * (((\frac{308k||37.2k}{100})||1.7k||75) + 75)}$$

$$\rightarrow C_{in} = 117nF$$

In this example, the resistances on the base side reflect to the emitter side. They are related by the β value. This is different from the common-emitter example where the emitter resistances reflected to the base side. This calculation was tedious and we could use an approximation in most cases,

$$f_{Cin} \approx \frac{1}{2\pi C_{in} * (\frac{1}{gm} + R_{source})} \approx \frac{1}{2\pi C_{in} * (75 + 75)} \approx 10kHz \rightarrow \ C_{in} = 106\,nF$$

The output coupling capacitor is calculated the same way as the common-emitter amplifier,

$$f_{Cout} = \frac{1}{2\pi C_{out} * (R_C||R_L)} = 10\ kHz = \frac{1}{2\pi C_{out} * (17.3k||100k)} \rightarrow C_{out} = 1\,nF$$

The schematic of the final amplifier is shown below in Fig. 11. Capacitor C3 has been added for stabilizing the DC bias voltage. C3 helps reduce noise and ripple at the base since it is important that the base voltage stays fixed. When constructing this circuit in the lab you must include C3. You can think of this as a decoupling capacitor. A transient analysis is shown in Fig. 12. Due to the large gain it is difficult to see the input voltage on the same scale. The gain equation earlier assumes the input voltage measured from the voltage source and factors in the source impedance. In the lab, the function generator gives an output voltage value when terminated into its source impedance. This already accounts for the voltage division between the source impedance and the load impedance. If your gain results are double what you calculated, this is probably the reason. To resolve this, simply drop the Rs term from the gain equation. Finally, the frequency response is shown in Fig. 13. The lower frequency limit of 10 KHz was met. While we didn't calculate for a particular upper frequency limit, we can see that it is quite good at 5 MHz. Reducing the collector resistor value would likely increase the bandwidth. The value of the CB amplifier is apparent from the plot. We could not get 39 dB of gain and 5 MHz of bandwidth from the general purpose op-amps in our lab. In the interest of brevity, we will not analyze a PNP CB amplifier in this lab.

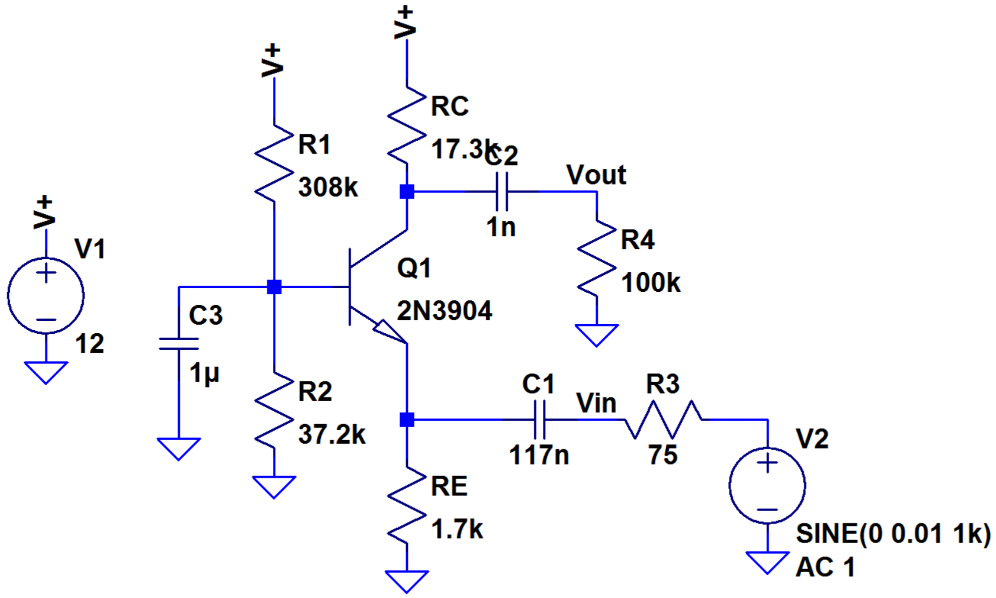


Figure 11. Final NPN CB amplifier schematic. C3 is for decoupling.

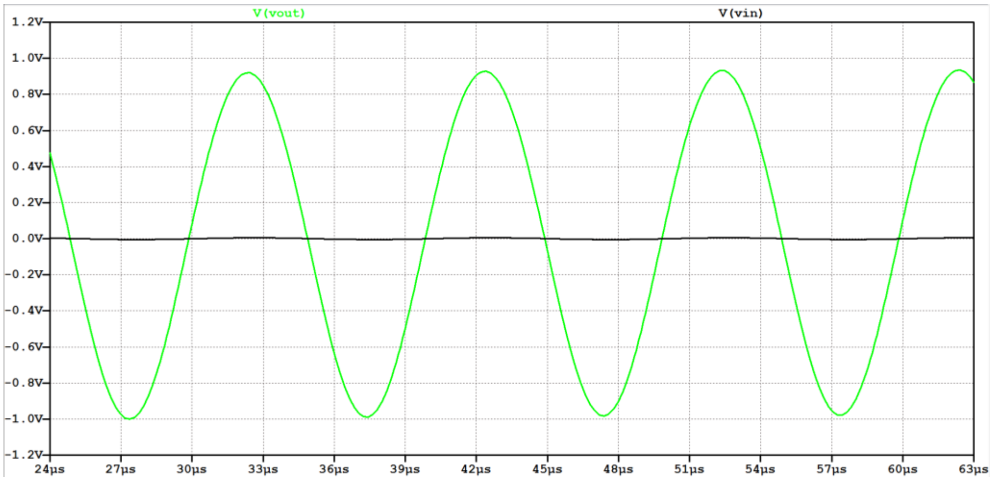


Figure 12. Vout and Vin with a 10 mV 100 kHz input signal.

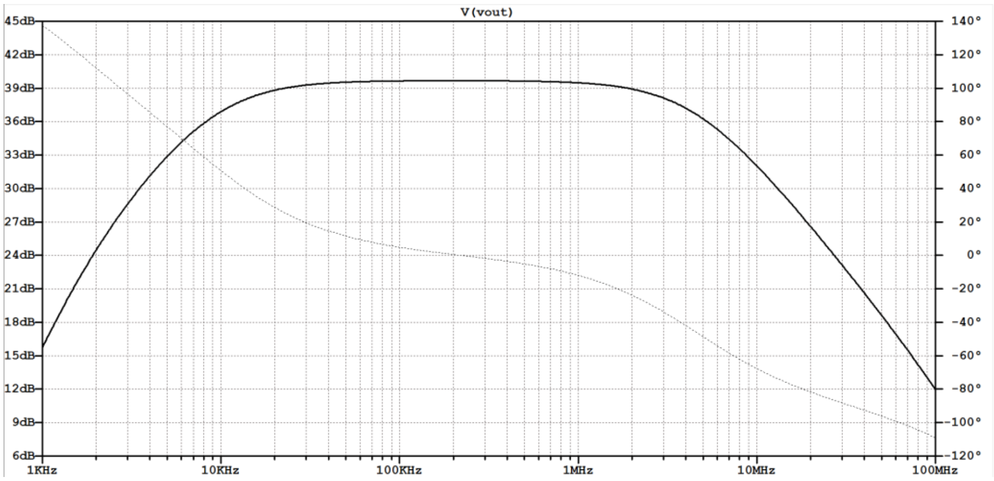


Figure 13. Frequency response of NPN CB amplifier.

The last amplifier topology we will be examining is the common-collector amplifier. This is more commonly known as the emitter-follower and it will be referred to as such. The emitter follower has no voltage gain. The maximum gain that it can offer is 1, although most of the time the gain is around 0.8 or 0.9. The usefulness of the emitter follower is that it can provide the current needed to drive a heavy load. For example, the previous CE and CB amplifiers could not drive a 50 ohm load because their output impedance is too high. The solution would be to connect an emitter follower between the amplifier and the load.

Suppose that the NPN CE amplifier designed earlier needs to be connected to a 25 ohm headphone/speaker. The output impedance of this amplifier is 1k which is too high to drive a 25 ohm load. This amplifier stage is modeled with a voltage source in series with a 1k resistor. The amplifier needs to be able to provide enough current to develop a 1V p-p signal across the speaker. This particular speaker can't reproduce frequencies below 300 Hz so we'll set that as the lower frequency limit. The amplifier schematic representing this problem is shown below in Fig. 14.

To develop 1V p-p across the load, the current in the 25-ohm speaker needs to be 40 mA. We'll try to set the voltage at the emitter to be 6V or ½ V+. A 150 ohm resistor is needed to develop 6V across it when a current of 40 mA flows through it. What if we selected a different voltage for the emitter? Since we only need to swing 1V p-p, what if instead of 6V the emitter voltage was set to 2V or 10V? The trade-offs will be gain vs. power dissipation in the resistor. For 2V, the emitter resistor needs to be 50 ohm. This is quite close to the 25 ohm speaker and will reduce the gain since it is in parallel with it. The power dissipation is 80 mW (I^2R). For 10V the emitter resistor needs to be 250 ohm. The power dissipation will be 400 mW. This exceeds the quarter-watt rating of the resistors in the lab. For the 6V case, the resistor's power dissipation is 240 mW which is too close to the limit. Let's drop the value of the resistor down to 100 ohm.

This results in 4V at the emitter and 160 mW of power in the resistor. The power ratings of resistors go down with increasing temperature. This has to be verified using the derating chart provided by the manufacturer. Since we don't have that, we'll be conservative and limit the power dissipation to half of the rated value.

The next step is to set the voltage at the base. Adding 0.7V to the 4V at the emitter results in 4.7V at the base. The base current is 40mA/100=400uA which is quite large. The divider current should be set to 10 times this value or 4 mA which again, is quite large. The divider R1+R2=3k for a 12V supply and 4 mA of divider current. R2 should be set to 1.17k (4.7V/4mA) and R1 should be the remainder or 1.83k. Immediately we run into the problem that our input impedance is less than our source impedance and therefore the input signal will be reduced. For this design example we will accept this issue and continue analysis. There are always trade-offs involved in design and a few iterations and tweaking are needed to arrive at the best solution given all the constraints. The input impedance is given by,

$$R_{in} = R_1 \left|\right| R_2 \left|\right| Re_{ref} = 1.83k \left|\right| 1.17k \left|\right| 2.08k = 531\ ohm$$

The resistors in the emitter side are reflected to the base,

$$Re_{ref} = (\beta + 1) * \left((Re||Rl) + \frac{1}{gm} \right) = 101 * (20 + 0.65) = 2.08k$$

We need to include the load resistance because it is significant compared to the other values. The input impedance in this case is less than our source impedance which is certainly not an ideal case. Despite this, there will still be useful load driving ability. A common way to improve the input impedance is to use Darlington or Sziklai pairs which are outside the scope of this lab.

The output impedance is calculated as,

$$R_{out} = \frac{1}{gm} \left|\right| Re \left|\right| Rload || Rb_{ref} \approx \frac{1}{gm} \approx 0.65$$

Where Rb_{ref} is the base resistances reflected to the emitter side. This is the complete solution, although the 1/gm term dominates and the other terms can be ignored for an approximate solution.

The gain of the emitter follower is,

$$A_V = \frac{Re||Rload}{\frac{1}{gm} + Re||Rload} = \frac{20}{0.65 + 20} = 0.97$$

This is only valid when the input signal is measured directly at the base. It ignores the voltage division between the source impedance and the input impedance of the transistor amplifier circuit. These should be included since we have a high value of source impedance as shown previously. Adding these terms results in a more complete gain equation when viewed from the signal source,

$$A_V = \frac{R_{in}}{R_{in} + Rsource} * \frac{Re||Rload}{\frac{1}{gm} + Re||Rload} = \frac{531}{531 + 1k} * \frac{20}{0.65 + 20} = 0.34$$

Again, this is only the case due to the high source impedance and does not indicate a poor performing emitter follower.

The last step is to calculate the coupling capacitors. The input coupling capacitor can be calculated with,

$$f_{Cin} = \frac{1}{2\pi C_{in} * Rin + Rsource} = 300\ Hz = \frac{1}{2\pi C_{in} * 531 + 1k} \rightarrow C_{in} = 0.34\ uF$$

The output coupling capacitor can be calculated as follows,

$$f_{Cout} = \frac{1}{2\pi C_{out} * Rout||Rload} = 300\ Hz = \frac{1}{2\pi C_{in} * 531||25} \rightarrow C_{in} = 22\ uF$$

The final schematic of our emitter-follower is shown in Fig. 14. The input and output voltages are shown in Fig. 15. The gain is 0.93 which is quite close to the calculated value of 0.97. Note that Vin is measured at the input of the amplifier and not the signal source. There is significant attenuation at the signal source due to the voltage division between the source impedance and the input impedance. The frequency response is shown in Fig. 16. The first plot shows Vout/Vin, and the second plot shows Vout with the reference voltage as the signal source. In this case, there is 9 dB of attenuation.

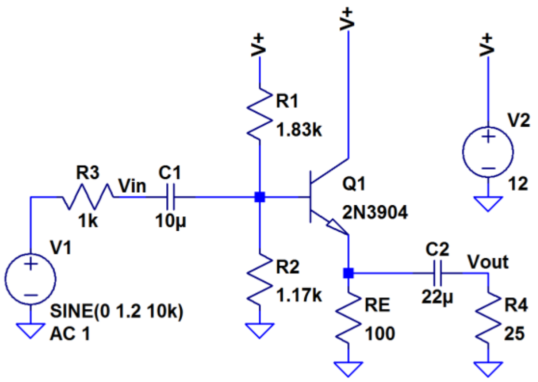


Figure 14. Final schematic of emitter follower.

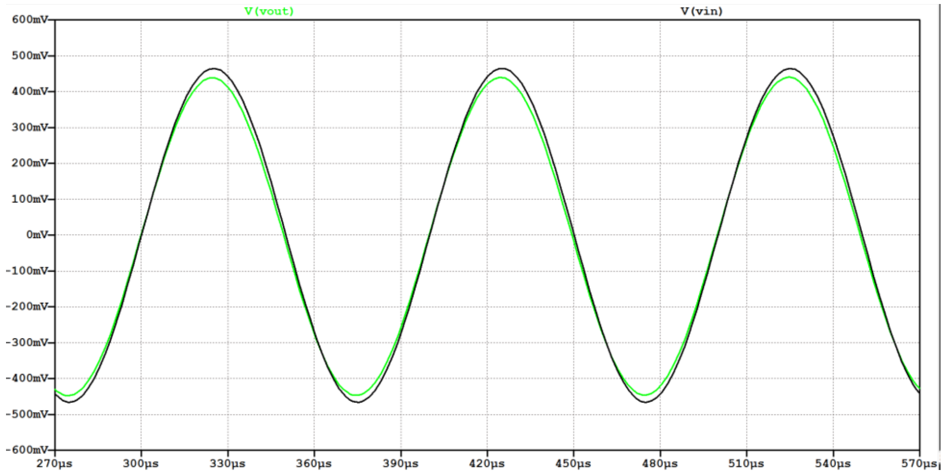


Figure 15. Plot showing input and output signal. Note that the input signal is at the base of the BJT. There is significant attenuation at the actual signal source.

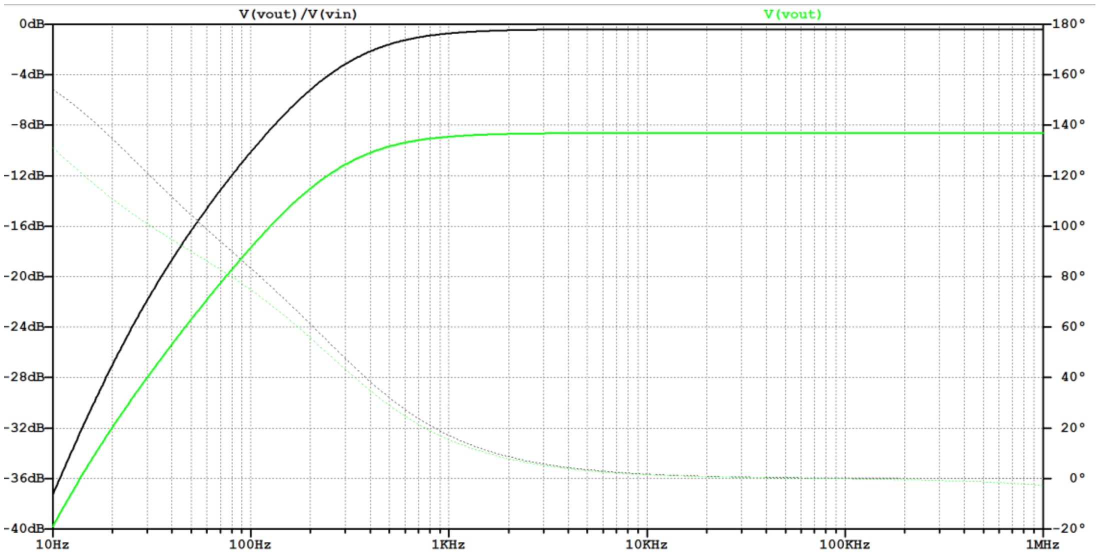


Figure 16. Frequency response of the emitter-follower. Note the attenuation in the green plot due to the voltage division between the source and input impedances.

Prelab #1: NPN Common-Emitter Amplifier

Design an AC-coupled common emitter amplifier with a gain of 10 and a lower frequency limit of 100 Hz. The amplifier will operate from a 15V power supply and the collector should be biased to 7.5V. The signal source will be a 50-ohm function generator. Make sure that the input impedance of the amplifier is at least 10 times greater than the source impedance. Use an NPN 2N3904 transistor. The amplifier will drive a 10k load resistor.

Simulate your schematic with a transient analysis showing input and output. Use a small input signal around 100 mV and show that the gain is around 10.

Simulate your schematic with an AC analysis showing that the lower corner frequency is near 100 Hz.

Turn in hand calculations (approximations are fine), schematic, transient analysis and AC analysis.

Prelab #2: PNP Common-Emitter Amplifier

Repeat prelab #1 but with a PNP 2N3906 transistor. All the requirements are the same as above.

Simulate your schematic with a transient analysis showing input and output. Use a small input signal around 100 mV and show that the gain is around 10.

Simulate your schematic with an AC analysis showing that the lower corner frequency is near 100 Hz.

Turn in hand calculations (approximations are fine), schematic, transient analysis and AC analysis.

Prelab #3: NPN Common-Base Amplifier

Design an AC-coupled common-base amplifier with a minimum gain of 50, a 50-ohm input impedance and a 10 kHz lower frequency limit while driving a 100k load. The amplifier will operate from a 15V supply. Set the collector voltage at ½ the supply voltage. Pick an emitter resistor that is 1/10th the collector resistor.

Simulate your schematic with a transient analysis showing input and output. Use a small input signal around 10 mV and show what your gain is. You may have to reduce the input signal if your gain is extremely high to prevent clipping.

Simulate your schematic with an AC analysis showing that the lower corner frequency is near 10 kHz.

Turn in hand calculations (approximations are fine), schematic, transient analysis and AC analysis.

Prelab #4: NPN Common-Collector (Emitter Follower) Amplifier

Design an AC-coupled emitter follower to drive a 25 ohm load to 1V p-p. The amplifier will operate from a 15V supply. Use a source impedance of 1k. This models a previous amplifier stage. Select an emitter resistor that does not exceed the ¼ watt rating of the resistor. This analysis is explained in the background section.

Simulate your schematic with a transient analysis showing input and output. Use an input signal around 1V and show what your gain (loss) is.

Simulate your schematic with an AC analysis showing that the lower corner frequency is near 300 Hz.

Turn in hand calculations (approximations are fine), schematic, transient analysis and AC analysis.

Prelab #5: Combined Amplifier

Connect the output of the amplifier from prelab #1 to the input of the amplifier in prelab #4. This is a common-emitter amplifier with an emitter follower.

Simulate your schematic with a transient analysis showing input and output. Use an input signal around 100mV.

Simulate your schematic with an AC analysis showing what the lower corner frequency is.

No hand calculations are required. Turn in schematic, transient analysis and AC analysis.

Postlab #1: NPN Common-Emitter Amplifier

Construct the amplifier that you simulated in prelab #1. Use the nearest resistor and capacitor values you can find in the lab to your calculated values. Show input and output on a scope. Roughly verify that the corner frequency is near what was calculated. **Turn in a picture of the circuit, scope picture of input/output and a picture of the lower corner frequency on the scope (70% of reference value).**

Postlab #2: PNP Common-Emitter Amplifier

Construct the amplifier that you simulated in prelab#2. Use the nearest resistor and capacitor values you can find in the lab to your calculated values. Show input and output on a scope. Roughly verify that the corner frequency is near what was calculated. **Turn in a picture of the circuit, scope picture of input/output and a picture of the lower corner frequency on the scope (70% of reference value).**

Postlab #3: NPN Common-Base Amplifier

Construct the amplifier that you simulated in prelab#3. Use the nearest resistor and capacitor values you can find in the lab to your calculated values. Show input and output on a scope. Roughly verify that the corner frequency is near what was calculated. **Turn in a picture of the circuit, scope picture of input/output and a picture of the lower corner frequency on the scope (70% of reference value).**

Postlab #4: NPN Common-Collector (Emitter Follower) Amplifier

Construct the amplifier that you simulated in prelab#4. Use the nearest resistor and capacitor values you can find in the lab to your calculated values. Show input and output on a scope. Roughly verify that the corner frequency is near what was calculated. **Turn in a picture of the circuit, scope picture of input/output and a picture of the lower corner frequency on the scope (70% of reference value).**

Postlab #5: Combined Amplifier

Construct the amplifier that you simulated in prelab#5. Use the nearest resistor and capacitor values you can find in the lab to your calculated values. Simply combine postlab #1 and postlab #4. Show input and output on a scope. Roughly verify that the corner frequency is near what was calculated. If your TA provides a speaker you can listen to your amplifier. **Turn in a picture of the circuit, scope picture of input/output and a picture of the lower corner frequency on the scope (70% of reference value).**

****Don’t try to be exact, if you’re within 20% it’s good enough!****