Design an Emitter Follower Stage

Table of Contents

- Design an Emitter Follower Stage
  - General Characteristics
  - Parameters
  - Select a Transistor
    - Initial Example Requirements
    - Current Flow
    - Noise Level
    - Transistors
    - Quiescent Current
    - Simulation
  - AC Coupled Emitter Follower
    - Biasing Resistors
    - Coupling Capacitor C1
    - Final Circuit
- Conclusions
  - Input Impedance
  - Output Impedance
  - Source Impedance
- Temperature Stability
  - Rules of Thumb
  - At 55°C

If you want to learn how to bias and use bipolar (BJT) transistors, this is the place to get started. This emitter follower article builds the foundation you will need to design the more often used common emitter configuration, described here.

We will design two circuits using a bipolar (BJT) transistor:

1. The DC coupled emitter follower
2. The AC coupled emitter follower

We will begin with the DC coupled circuit.

General Characteristics

What are some of the general characteristics of the emitter follower circuit?

- Voltage gain is slightly less than unity
- High input impedance
- Low output impedance

The slight loss of voltage gain sounds like a disadvantage but the stage does however have *power* gain. This makes it a good buffer stage.

The high input impedance allows weak signals to be buffered. The emitter follower with its high input impedance and relatively low output impedance, makes it a good impedance adapter.

Parameters

Before you design something, you should list your requirements. You need to specify the answers to the following questions:

- How much current do you want your transistor to conduct? (This is the usual design starting point)
- What supply voltage will you be using?
- What is the largest input signal you expect to amplify?
- Input impedance requirement?
- Output impedance requirement?
- What maximum temperature must our circuit operate at?

Initially, we'll skip the input and output impedance requirements, and just compute what they are. These can be adjusted at the compromise of other factors.

Select a Transistor

The transistor input voltage, current flow, supply voltage and noise requirements will narrow your search to a set of possible BJT (bipolar) transistors. We'll be looking at the design from an audio signal level point of view.

Initial Example Requirements

We're going to assume that we're designing an audio buffer stage, which is something an emitter follower does well. Our example initial requirements will be:

- Input voltage: 3 volts peak-to-peak
- Current Flow: 1 mA (collector current IC)
- Supply Voltage: 12 volts
- Noise Level: as low as possible
- We'll assume our little amplifier may experience a maximum temperature of 55C (it is inside a box).

The maximum input voltage of the signal and the power supply voltage are usually quite obvious. The other parameters will need more explanation, which we'll address later on.

Current Flow

You might be wondering "How do I know what current flow to specify?". This will be discussed more later on but noise is often a factor in the choice.

Small signal stages usually use 0.1 mA up to about 10 mA. If you use low collector currents, then the biasing resistors are usually higher in resistance. This makes them difficult to obtain and contribute to Johnson noise (more resistance means increased noise). At higher currents, temperature and stabilization becomes a larger factor, not to mention battery life.

Noise Level

Audio amplifiers usually require that the noise be reduced as much as possible. There are exceptions to this, like perhaps in an intercom circuit where low cost may be more important. You'll also learn that the level of current flow through the transistor affects noise.

Despite the above, noise is usually only a serious concern for amplifying very weak signals. Often only the first stage in a preamplifier is given any real consideration to this. However, it should always be born in mind when choosing resistors (the most controllable source of noise).

Transistors

We're going to look at two commonly used NPN transistors, for illustrative purposes.

- Fairchild 2N3904
- Fairchild 2N2222A

Frequency

Normally we'd be very concerned about the part's frequency performance in our selection. Since we're working with an audio application, we'll safely ignore this.

Absolute Maximum Ratings

The first thing we'll notice is the difference in the absolute maximum ratings for the transistor's current:

- 2N3904 - 200 mA continuous
- 2N2222A - 1.0 A (Note 2 says continuous)

Right away we see that the 2N3904 is probably the right part for our example, since we are using a smaller collector current (the 2N2222A allows up to 1 Amp, vs the 200 mA max of the 2N3904).

Both parts operate at a maximum of 40 volts. So a 12 volt supply will not present any problem.

Power Dissipation

One thing we must also keep in the back of our mind is the power handling capability of the part. The rating for the transistor is shown at right. Here we see that the maximum power for the 2N3904 is 625 mW at 25C (TA=25C indicates that the ratings are for an ambient temperature of 25C).

Thermal Characteristics T_A = 25°C unless otherwise noted

Symbol	Characteristic	2N3904	Units
P_D	Total Device Dissipation Derate above 25°C	625 5.0	mW mW/°C

$$\begin{aligned} P_{2N3904} &= 625 - (T - 25)\,(5.0) \\ &= 625 - (40 - 25)(5.0) \\ &= 625 - 75 \\ &= 550mW \end{aligned}$$

Additionally, if the temperature of the transistor increases, we must make certain that the derated power rating is not exceeded. For example, if the transistor must operate at 40C, then according to the calculation at left, we must not exceed 550 mW.

$$\begin{aligned} I_{C_{max}} &= \frac{550mW}{12volts} \\ &= 45mA \end{aligned}$$

Since power is volts times amps, we can do a simple division to work out the maximum current of the transistor (Ic). At left we've calculated a max current of 45mA, which is well above our currently planned 1 mA operating point.

HFE (DC Beta)

When selecting an operating current, you should also review the HFE (DC gain, also known as DC Beta). The Beta of the transistor is shown in the "Min" column for each collector current (Ic) in the column "Test Conditions".

Notice that the Beta is highest when the Ic is 10 mA. The "Max" column shows that the parts can have a Beta as high as 300 (at 10 mA).

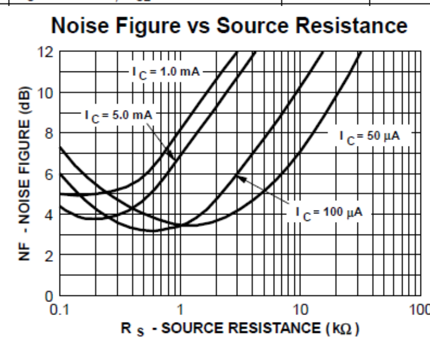
Our required Ic=1 mA shows a minimum Beta of 70.

Electrical Characteristics T_A = 25°C unless otherwise noted

Symbol	Parameter	Test Conditions	Min	Max
ON CHARACTERISTICS*				
h_{FE}	DC Current Gain	I_C = 0.1 mA, V_{CE} = 1.0 V	40	
		I_C = 1.0 mA, V_{CE} = 1.0 V	70	
		I_C = 10 mA, V_{CE} = 1.0 V	100	300
		I_C = 50 mA, V_{CE} = 1.0 V	60	
		I_C = 100 mA, V_{CE} = 1.0 V	30	

Noise

From the chart above, we saw that the 2N3904 has the greatest Beta when operated at 10 mA. We might want to change our requirement because of this fact (we were going to use a 1 mA of collector current). However, let's review the noise chart that Fairchild has kindly provided before we decide.



From this chart we see that the noise is lowest, when the transistor is operated at very low currents (as low as 50 uA). We also see that the highest noise graph appears to occur when Ic=1.0 mA (which was our original choice). Also we notice that we do not have a graph for when the collector current is 10 mA, which is the point where the 2N3904 has the highest gain (Beta).

According to this chart, if we operate the 2N3904 at 5 mA we'll get improved gain (over the 1.0 mA Beta=70) and a lower noise figure (vs Ic=1 mA). Perhaps this is the sweet spot for our application.

So let's revise our requirement to operate the transistor at 5 mA (Ic=5 mA). We're also reminded that 5 mA is well under the 45 mA maximum to allow for safe operation at a temperature of 40C.

Noise Footnote

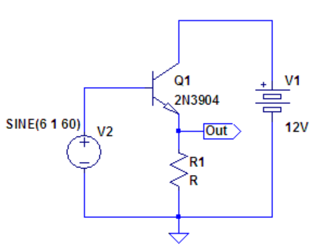
In the above design work, we only concerned ourselves with the transistor noise figure. To truly achieve low noise performance requires that some other compromises to be made that we have glossed over. For example, as you lower current flow in a transistor, there are higher resistances involved. Resistors add to the noise generated and noise increases with resistance. So it is necessary to find an optimal solution in a compromise between the transistor noise level and the resistive components used.

In this example, we will assume the base is directly coupled to the input signal source. So there are no bias resistors to consider from a noise perspective. As the Ic is increased and thus the emitter (Ie) current, we know that the emitter resistance can also be reduced. So increasing Ic=5mA, is helping us improve our noise figure.

Although less of an issue at audio frequencies, stray capacitance is another factor to be considered when using low collector currents. The active component must be able to charge and discharge any stray capacitance as the signal fluctuates. As the operating current is reduced, the ability to charge/discharge is weakened thus affecting bandwidth.

In audio circuits RC constants are significant and should be re-examined when resistances are changed. Watch out for RC constants that affect the circuit's bandwidth (like coupling capacitors, which don't apply until we look at an AC coupled stage).

Quiescent Current



Quiescent current is the amount of current flowing through our transistor when no signal is being amplified. In the preceding section we chose a current of 5 mA, since this offers better noise performance. So how do we determine the value for the emitter resistor R1?

We know that whatever flows through the transistor must also be flowing in the emitter resistor R1 (series circuit). So if we know the quiescent voltage at "Out", we can calculate R1 using ohms law.

Emitter Voltage or V(R1)

The voltage found at the emitter will be the voltage of the transistor's base, minus approximately 0.6 volts for silicon components. For this example, we'll assume that the base is coupled directly (DC coupled) to the preceding stage. For maximum signal swing, let's also assume that the base is hovering at about half of the supply voltage (12 volts divided by 2 here).

So if the base of the transistor is 6 volts, we know the voltage at the emitter (and thus across R1) is approximately 5.4 volts.

$$\begin{aligned} V_E &= V_B - 0.6 \\ &= 6 - 0.6 \\ &= 5.4 \text{ volts} \end{aligned}$$

Calculating R1

$$\begin{aligned} R &= \frac{V}{I} \\ &= \frac{5.4volts}{5mA} \\ &= 1080 \text{ ohms} \end{aligned}$$

With V(R1) known, we can compute R1 using ohms law. It computes to 1080 ohms. But using the Resistor Calculator we find that a standard 10% resistor value is 1k ohms.

```
The closest 10% resistor for 1.08K ohms is 1K ohms.
The error is 7.4 %.
```

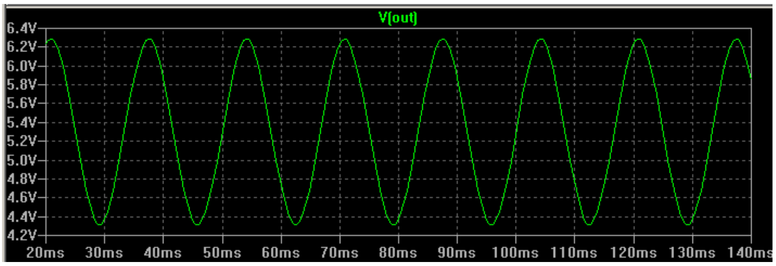
Simulation

With R1 known, we can complete the schematic and run a simulation in LTspice.

In the simulation, the base of the transistor was supplied a sine wave:

- DC offset of 6 volts
- Sine wave of +/- 1 volts (peak)
- Frequency of 60 Hz

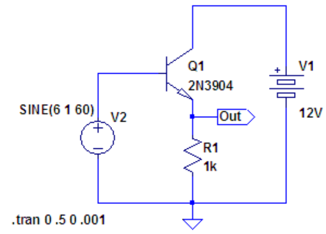
From this you can see that the voltage at the emitter was centered near 5.4 volts as we hoped.



If you do a similar trace of the current through R1, you would also see that it is centered at approximately 5.4 mA, which isn't far from our targeted 5 mA. An increase in current is expected since we calculated 1080 ohms for R1 but used a 1K resistor instead. To get closer to 5 mA, you could use a 1.07K 1% resistor instead, to arrive at Ic=5.04 mA (Ie ~= Ic).

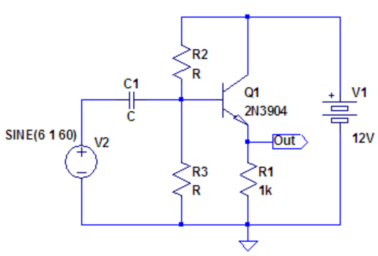
Emitter_Follower.asc

```
Version 4
SHEET 1 880 680
WIRE 448 16 272 16
WIRE 272 96 272 16
WIRE 448 128 448 16
WIRE 208 144 96 144
WIRE 96 224 96 144
WIRE 272 224 272 192
WIRE 320 224 272 224
WIRE 272 256 272 224
WIRE 96 384 96 304
WIRE 272 384 272 336
WIRE 272 384 96 384
WIRE 448 384 448 208
WIRE 448 384 272 384
WIRE 272 416 272 384
FLAG 272 416 0
FLAG 320 224 Out
IOPIN 320 224 Out
SYMBOL Misc\\battery 448 112
WINDOW 123 0 0 Left 0
WINDOW 39 0 0 Left 0
SYMATTR InstName V1
SYMATTR Value 12V
SYMBOL voltage 96 208 R0
WINDOW 3 -148 2 Left 0
WINDOW 123 0 0 Left 0
WINDOW 39 0 0 Left 0
SYMATTR InstName V2
SYMATTR Value SINE(6 1 60)
SYMBOL npn 208 96 R0
SYMATTR InstName Q1
SYMATTR Value 2N3904
SYMBOL res 256 240 R0
SYMATTR InstName R1
SYMATTR Value 1k
SYMATTR SpiceLine tol=10%
TEXT -64 416 Left 0 !.tran 0
```



AC Coupled Emitter Follower

In the preceding circuit, we DC coupled the base of our transistor to the prior stage. Whatever voltage appeared at the base, will cause the follower's emitter to be approximately 0.6 volts less (for silicon). An emitter follower is often used this way, since this avoids the need to bias the transistor and avoids the coupling capacitor.



An emitter follower has a slight loss in voltage gain but makes up for it in current. Thus overall the power of the signal is increased. The follower circuit is often used as a buffer stage for this reason.

Sometimes you must use AC coupling. If you must AC couple the input signal, then you have some additional design work to do. You need plan for:

- frequency response
- bias voltage on the base

You'll notice we've added a voltage divider consisting of R2 and R3. We've also added the AC coupling capacitor C1. We'll need to calculate component values for all of these added components.

Biasing Resistors

One of the design tasks we have is to design the bias circuit consisting of R2 and R3. Our input signal will vary between +1 and -1 volts. We also know that the emitter voltage will be approximately 0.6 volts less than the voltage at the base. So our choice of bias cannot be any less than 1.6 (1.6 - (-1) - 0.6 = 0) volts and no more than 10.4 volts (12 - (+1) - 0.6). The best and safest bias point is midway between the positive supply (+12 volts) and ground. So let's aim for a bias voltage of 6 volts.

We could easily come up with many different pairs of values for R2 and R3 as a voltage divider. However not all values would be suitable for this circuit. We need a divider that is stiff enough to provide the bias for the transistor but not too stiff that it unnecessarily wastes power. For example you could make R2 = R3 = 5 ohms, but that would mean a steady drain of 12 volts divided by 10 ohms (total), for 1.2 Amps! So just how stiff does our divider need to be?

We must satisfy the relationship at right. R1 is the emitter resistor (Re). The bias resistors in parallel must be much smaller than the emitter resistor than the emitter resistor times Beta.

$$R_2 \| R_3 \ll \beta R_1$$

The first thing we need to determine is how much current will be drawn by the base of the transistor from the divider. If we know the Beta (HFE) for the transistor we can arrive at a good approximation. Reviewing the Beta figures for the 2N3904 transistor (see right), we have a minimum Beta of 70 (when current is 1 mA). We don't have a figure for 5 mA, but we know it must be between 70 and 100. Splitting the difference, let's assume a minimum Beta of 85 at 5 mA.

ON CHARACTERISTICS*

h_{FE}	DC Current Gain	I_C = 0.1 mA, V_{CE} = 1.0 V	40	
		I_C = 1.0 mA, V_{CE} = 1.0 V	70	
		I_C = 10 mA, V_{CE} = 1.0 V	100	300
		I_C = 50 mA, V_{CE} = 1.0 V	60	
		I_C = 100 mA, V_{CE} = 1.0 V	30	

$$I_B = \frac{I_C}{\beta} = \frac{5mA}{85} = 0.06 \text{ mA}$$

Using a calculation involving Beta, we can approximate the current going into the base of the transistor. In the calculation shown at left, we see that it amounts to approximately 0.06 mA.

You'll recall that the general rule for a voltage divider is that the divider resistors should conduct approximately 10 times the current of it's load (otherwise the resulting voltage will not be accurate). So now we know that the resistors R2 and R3 in series should conduct 0.06 mA times 10, or 0.6 mA (or more).

$$R3 = \frac{6volts}{0.6mA} = 10k \text{ ohms}$$

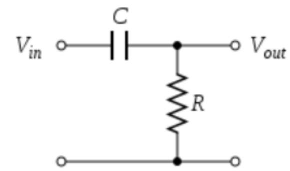
In our example R1 = R2, since we're splitting the supply voltage (+12 volts) in half. So focusing on R3, we know the voltage across it is 6 volts and we now know it's current should be 0.6 mA. Using ohms law we calculate that R3 should be 10k ohms (which happens to be a standard resistor value). Since R2 and R3 are equal, we now have the voltage divider we need.

Coupling Capacitor C1

Our example is an audio amplifier, so we'll assume the frequency response must be 20Hz to 20khz. So what value of the coupling capacitor should we use for C1 in our AC coupled circuit? The input circuit is like a high pass filter (see right).

$$f_c = \frac{1}{2\pi RC}$$

You'll recall that the reactance of a capacitor reduces as the frequency is increased. So we only need to worry about the lowest coupled frequencies. We can compute the low end cutoff frequency using the calculation at left.



Using Thevenin's Theorem, we know the input impedance of our emitter follower circuit is approximately R2||R3, which is 5k ohms (we can ignore the base of the transistor here because the input impedance is high). So the R in the highpass filter is 5k ohms. We know the frequency cutoff should be 20Hz. So what we want to calculate is C.

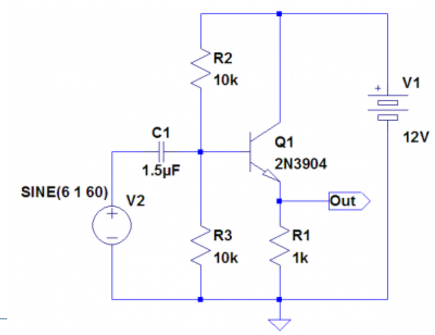
You can rearrange the calculation algebraically and calculate C yourself, or just use the Online RC Filter Calculator. Plug in Frequency and Resistance and click on "Calculate". Remember the resistance must be given in ohms and the frequency is simply 20. The calculator should return a result of 1.592 uF. Given that the standard component values (see Standard Capacitor Values) are 1.5 uF or 2.2 uF, you might choose 1.5 uF.

As a designer, you also have the option of tweaking the voltage divider. You can make the divider (R2 and R3) a bit stiffer, causing R in the high pass filter to change. You don't want to make the divider weaker but you can make it stiffer. A stiffer divider lowers the input impedance of the amplifier stage (lowering R). This also affects the frequency response or the capacitor value used. Reducing the resistors R2=R3=9k ohms for example requires a minimum capacitor of 1.769 uF.

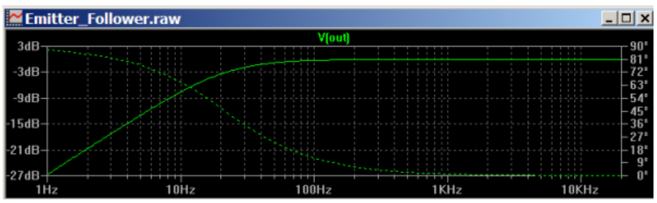
Another design approach is to plug in the 20Hz for frequency and supply 2.2 uF (perhaps 2.2 uF capacitors are cheaper). When you run this through the online calculator, it tells you that R should be 3619 ohms. So then you could decide on a standard resistor value that comes out close to this parallel resistance value for R2 and R3. Remember the calculated R is *parallel* resistance.

Final Circuit

The final emitter follower circuit is shown at right. Checking the bias resistors, we see that R2||R3 is 5k, which is well under the Beta times R1. Even if Q1 had the lowest Beta of 30, we see that the bias parallel resistance is still well under 30 * 1k ohm.

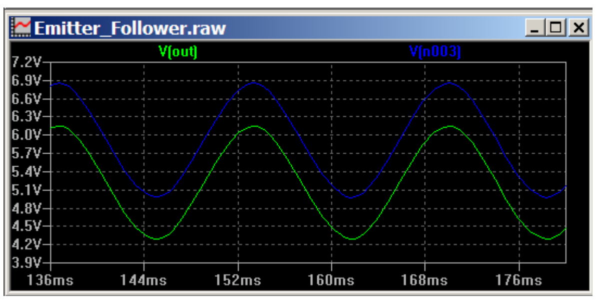


The AC Analysis (LTspice) is provided below (or left) as well:



From this you can see that the -3dB point occurs at 20 Hz as we planned it. The frequency response is otherwise flat from 20 Hz all the way past 20 kHz.

Conclusions



It is fitting that we end with a scope trace of the input and output voltages. The wave form shown in blue is the voltage appearing at the base of the transistor. The yellow trace shows the voltage that emerges at the emitter (and across the emitter resistor R1).

Careful observation will show that the emitter voltage is always approximately 0.6 volts less than the base input voltage. From this we see that the emitter *follows* the base, hence the term "emitter follower".

Input Impedance

$$r_{in} = (\beta + 1)\, R_e$$

The formula at left shows how the input impedance looking into the base of the transistor. This is affected the Beta (HFE) of the transistor and the emitter resistor used. Many transistors have a Beta of 100 or more.

For the DC coupled emitter follower, if we assume a minimum Beta of 85 and an emitter resistor of 1k, we get a minimum input impedance (at the base) of 85k ohms. Essentially the emitter follower impedance multiplies what it sees in the emitter resistor by it's Beta.

The *AC coupled* emitter follower has the bias resistors in parallel with the base of the transistor. The base impedance of 85k ohms when in parallel with the bias resistors, is high enough that it can be ignored. Essentially the input impedance (ignoring coupling capacitor) is R2||R3. A more complete treatment of input impedance is provided in the common emitter stage example. The emitter follower (AC coupled) has the same input impedance as the common emitter configuration (note that the bias resistors are labeled R1 and R2 in that example).

Note also that while the formula above uses simple resistances, complex impedances follow the same rule. Any inductance or capacitance added to the emitter circuit would be reflected to the input.

Output Impedance

$$Z_o = (r_e + \frac{Zs \| R_2 \| R_3}{\beta + 1}) \| R_1$$

Zs represents the source impedance at the input. R2 and R3 when present, are the bias resistances that are attached to the input (in the Thévenin sense). These resistances are divided by the Beta + 1, which tends to make this component insignificant and can be ignored.

This leaves intrinsic emitter resistance re. The resistance re is in parallel to emitter resistor R1. But re dominates since it's value is near 25 ohms. Practically speaking, you can say that the output impedance will be re.

Clearly the emitter follower has low output impedance!

Source Impedance

We mentioned source impedance briefly in the topic of Output Impedance. Source impedance plays a role in the overall performance of an amplifier stage but this is perhaps too much to expand upon here. Chapter 7 "Small Signal A.F. Amplifiers" in the book "Principles of Transistor Circuits" by Stan Amos & Mike James (ISBN 0-7506-4427-3) covers this topic well.

That chapter also makes a brief mention of linearity concerns related to source impedance. The article Linearity of the Transistor Amplifier makes the point that a small amount of negative feedback does an extremely effective job of eliminating this concern. Using an emitter resistor for example, allows the designer to ignore this issue.

Temperature Stability

One important topic that was ignored is the need for temperature stability. Besides simplifying the design discussion above, the emitter follower tends to be relatively immune to temperature instability.

In addition to increased Beta, the bipolar (BJT) transistor undergoes two main changes as temperature increases:

- The VBE decreases
- The IC increases

So as a BJT dissipates power (producing heat) the heat in turn causes it to consume even more current and thus dissipate more power. If the condition is left unchecked, it can lead to thermal runaway in some designs.

The emitter follower is generally resistant to this because as the current in the collector increases, the voltage across the emitter resistor also increases. This acts as negative feedback, since it reduces the voltage difference between base and emitter (VBE). This counteracts the transistor's tendency to conduct more current. Thus thermal stability is maintained.

To check this, let's run some numbers.

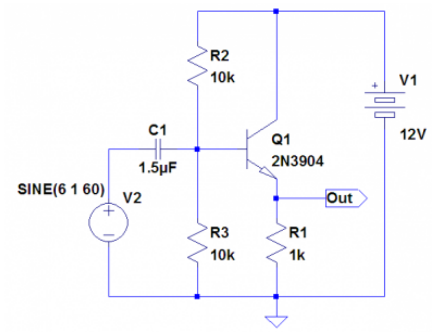
Rules of Thumb

The following chart summarizes some rule-of-thumb figures for temperature change. The left pair of columns indicate that (for silicon):

- for each degree of increase leads to 2.1 mV reduction of VBE
- for each 60mV of VBE reduction, the collector current increases 10 times.

The third column shows that (for silicon) for each 6°C increase, the collector current doubles. For germanium transistors, it takes a 10°C increase to double its collector current.

VBE	ICx10 per VBE	IC doubles per increase of	Transistor Type
-2.1 mV/°C	60 mV	6°C	Silicon
-2.5 mV/°C	125 mV	10°C	Germanium



The design we came up with (shown at right), had a collector current of approximately 5 mA. This design assumed an operating temperature of 25°C. Under these conditions, we have the base biased at 6 volts by the voltage divider R2 and R3. Under quiescent conditions then, the voltage at the emitter would be VBE (0.6 volts) below that, or 5.4 volts. We can precisely calculate a current of 5.4 / 1k ohm ⇒ 5.4 mA.

Given the transistor is silicon, we note that if we increase the temperature by 6°C, that the collector current wants to double at 10.8 mA. But can it do that in this circuit?

If the current were to increase to 10.8 mA, the voltage across R1 would then be 10.8 mA * 1k ohm ⇒ 10.8 volts. The base voltage would need to be 10.8 + VBE to support this (approx 11.4 volts). We know that the base is held steady at 6 volts, so this is unachievable. If the emitter voltage was somehow to rise to 5.4 volts + VBE, the transistor would cut off.

We also know that VBE drops with temperature. Assuming a room temperature VBE of 0.6 volts for silicon, then if the temperature increased by 6°C, we'd have 0.6 volts - 2.1 mV * 6 ⇒ 0.587 volts. With the bias held steady at 6 volts, the voltage seen at the emitter should be 6 volts - 0.587 volts ⇒ 5.41 volts. Therefore we do see a temperature induced increase in current (5.41 volts / 1k ohm ⇒ 5.41 mA). But this is still well within the selected transistor's current limits. The collector current has not doubled and shows no reason to “run away”.

Finally, we note that the current has increased about 0.2% for a 6°C increase in temperature for this particular circuit. This will give us a convenient shortcut in our power calculations.

At 55°C

While it would appear that we have no thermal run away to worry about, we still have not really answered the question:

- will the design function ok at it's highest temperature?

Thermal Characteristics T_A = 25°C unless otherwise noted

Symbol	Characteristic	2N3904	Units
P_D	Total Device Dissipation Derate above 25°C	625 5.0	mW mW/°C

Let's assume a maximum temperature of 55°C, which is rather conservative. For power amplifiers, you may need a much higher margin. You may also need to design for a higher temperature if the circuit is enclosed with poor ventilation.

The 55°C temperature is 30°C higher than our room temperature of 25°C. A quick “back of the envelope” calculation tells us that 30°C is 5 times the 6°C increase figures. So if a 6°C increase leads to a 0.2% increase in collector current, then we know that a 30°C increase is 5 times that. That is 5 times 0.2%, which is a 1% increase.

A 1% increase of 5.4 mA would be 5.45 mA for the transistor.

Checking the power, we know the 2N3904 transistor has an absolute maximum rating of 625 mW. This is the rating for 25°C (the ambient temperature). At 55°C, we have to derate this figure by 5 mW/°C. Since we are 30°C over the ambient temperature, we derate the maximum by subtracting 5 * 30 ⇒ 150 mW/°C. So at 55°C the maximum power dissipation is 625 - 150 ⇒ 475 mW.

At 55°C we estimated a current (average) of 5.45 mA, operating at approximately 12 volts - 5.2 volts ⇒ 6.8 volts. So the device is dissipating approximately 6.8 volts * 5.45 mA ⇒ 37 mW. This is well within the 475 mW maximum.

So we conclude that we are operating the device safely at 7.8% of it's maximum rated power (at 55°C).