The "T" Model and the Shunt-Series Amplifier

By Thomas T. Lee
Adapted from Chapter 2 of the EE344 Course Reader for Autumn 1999 by Howard Swain

1 Introduction

Most of you should be familiar with the "hybrid-π" small-signal model for bipolar and MOS transistors. Here, we are going to look at a different type of small signal model for bipolar devices known as the "T" model. Although it is not as popular today as it once was, this model is useful for analyzing the shunt-series amplifier you will be building in lab 1. Both the T and the hybrid-π models can be used to analyze the circuit - feel free to choose whichever is more comfortable for you.

2 The "T" model

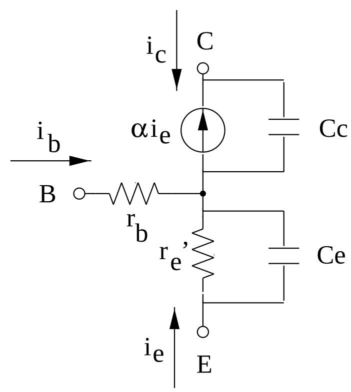

Figure 1: The "T" model

The T model is shown in Figure 1. In this model, r'_e models the relationship between v_{be} and i_e. For a BJT, $Ie \approx Ic = I_0 \left[\exp \frac{V_{be}}{V_t} - 1\right]$, where $V_T = \frac{kT}{q}$. r'_e is the differential change in i_e with v_{be}. In other words, $r'_e = \frac{dV_{be}}{dI_e} \approx \frac{V_T}{I_E}$.

The current-controlled current source feeding the collector is controlled by the emitter current. The coefficient α is known as the *base transport factor*, which is the fraction of carriers leaving the base that actually arrive at the collector, (the rest recombine in the base) and is generally close to unity. For those of you who prefer to think in terms of β (also known as h_{fe}), $\alpha = \frac{\beta}{\beta+1}$.

There are also two major parasitic capacitors, C_c and C_e, which model the charge storage and junction capacitances at between the base and collector, and base and emitter, respectively. We will be using our devices well below the unity-gain frequency (f_T), so these parasitic elements shouldn't matter too much. (But keep in mind that they do exist, just in case you get some screwy data...)

From this core, you can attach any element you wish to any of the terminals. In all cases, the T model and the hybrid-π model should give you the same answers in analysis. However, for certain circuits one model may help you get there faster than the other. One such example is the shunt-series amplifier, which is explained below.

3 The Shunt-Series Amplifier

The shunt-series amplifier is shown in Figure 2. You may recognize it as a variation on the common-emitter amplifier with emitter degeneration. The "shunt" refers to R_f, which provides shunt-shunt feedback, and the "series" refers to R_e, which provides series-series feedback. The principal feature of this circuit is that the

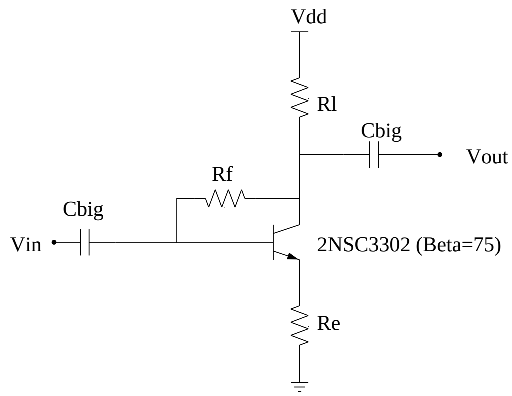

Figure 2: The Shunt and Series Amplifier

feedback resistors allow the designer to control the input and output impedance, providing good impedance matching to the outside world over a broad range of frequencies.

To analyze the shunt-series circuit, we call on our new friend, the T model. A small-signal circuit with the feedback resistors included is shown in Figure 3. For this analysis, we will use the node equations, which will give us "exact" answers, and lead us to some compact design equations. Note that we have chosen to neglect the parasitic capacitances and the base resistance. Note also that the definition for R_{in} does not include the source resistance, nor does the definition for R_{out} include the load resistance. The reason for this is that we want to find out what impedance the amplifier presents to the source and load circuits.

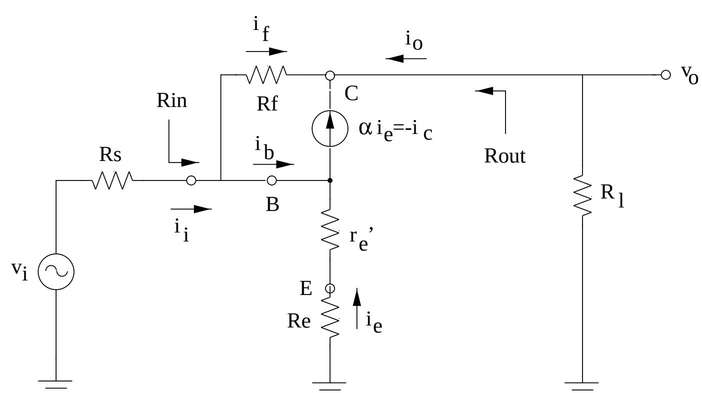

Figure 3: A T model representation of the shunt-series amplifier

The parameters we wish to extract from this circuit are gain, input impedance, and output impedance. To solve for gain, we can simply write the node equations. Throughout our analysis, we will assume that the base resistance and parasitic capacitors are negligible. We will also lump the two emitter resistances into one, $R_E = R_e + r'_e$.

We first note that $i_e = \frac{-v_i}{R_e + r'_e} = \frac{-v_i}{R_E}$ and $i_b = -i_e(1-\alpha) = \frac{v_i(1-\alpha)}{R_E}$.
The currents at the input are:

$$i_i = v_i\left(\frac{1}{R_F} + \frac{1-\alpha}{R_E}\right) - \frac{v_o}{R_F} \tag{1}$$

At the output node, we have:

$$0 = -\alpha\left(\frac{-v_i}{R_E}\right) - \frac{v_i}{R_F} + v_o\left(\frac{1}{R_L} + \frac{1}{R_F}\right) \tag{2}$$

Solving for v_i and v_o, we obtain:

$$R_{in} = \frac{v_i}{i_i} = \frac{R_E\left(R_L + R_F\right)}{R_E + R_L + R_F\left(1-\alpha\right)} \approx \frac{R_E\left(R_L + R_F\right)}{R_E + R_L} \tag{3}$$

The input resistance can be approximated by using the Miller Effect, in which case $R_{in} = \frac{R_f}{1+|A_V|}$.
To obtain a gain expression:

$$\frac{v_o}{v_i} = -\frac{R_L\left(\alpha R_F - R_E\right)}{R_E + R_L + R_F\left(1-n\alpha\right)} \tag{4}$$

$$A_v = \frac{v_o}{v_i} = \frac{v_o}{i_i}\frac{i_i}{v_i} \approx -\frac{R_L\left(\alpha R_F - R_E\right)}{R_E\left(R_L + R_F\right)} \tag{5}$$

Note here that the gain expression does not include the loading effects of R_S, which will reduce the gain seen by the signal source. (If $R_S = R_{in}$, then the gain will decrease by half, or 3 dB. Just remember that when you do your design, you need to add 3 dB to account for this loss.)
To calculate output impedance, we perform a similar analysis and obtain:

$$R_{out} = \frac{v_o}{v_i} = \frac{R_E\left(R_S + R_F\right) + R_S R_F\left(1-\alpha\right)}{R_E + R_S} \approx \frac{R_E\left(R_S + R_F\right)}{R_E + R_S} \tag{6}$$

Here we see something interesting. Note that when $R_S = R_L$, $R_{in} = R_{out}$. If we have equal source and load impedances, then the input and output impedances are equal! This makes impedance matching very easy when working with systems that have a given characteristic impedance (such as 50Ω). If we set $R_S = R_L$, we can rewrite the expressions for R_{in} and R_{out} as:

$$R_0 \triangleq R_{in} = R_{out} = \frac{R_E\left(R_0 + R_F\right)}{R_E + R_0} \tag{7}$$

$$R_0 = \sqrt{R_E R_F} \tag{8}$$

From equation 5, we have:

$$|A_v| \approx -\frac{R_0\left(\alpha R_F - R_E\right)}{R_E\left(R_0 + R_F\right)} \tag{9}$$

These equations are nice for analysis, but somewhat cumbersome if you try to use them for design. After doing some heavy algebra on the above equations, we arrive at some nice design equations:

$$R_F \approx \left(|A_v| + 1\right) R_0 \tag{10}$$

$$R_E = \frac{R_0}{|A_v| + 1} \tag{11}$$

So now we have some quick and dirty design equations to use for the RF part of our design. The next step is to design the DC bias circuitry.

4 Biasing

There are a myriad ways to bias a single-transistor amplifier, each with their advantages and disadvantages. The one in Figure 4 is particularly nice because it has few components, produces bias voltages that are largely independent of β, and does not load the RF circuit heavily.

One thing to remember about bias circuits is that the DC resistor values may differ from the RF values. Note that the subscripts on the resistor labels in Figure 4 are capitalized, indicating that they are meant for DC bias. In some cases, we can make the DC and RF portions independent, in other cases we may have to

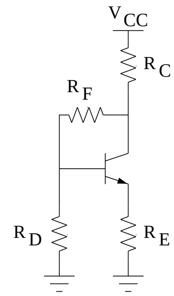

Figure 4: A simple bias circuit

account for the DC bias circuitry when designing the RF section. We will see how this is done later in the handout.

The circuit works as follows: V_{BE} sets the voltage across R_D, making $I_{R_D} = \frac{V_B}{R_D}$. If we choose R_D such that $I_{R_D} \gg I_B$, then $I_{R_F} \approx \frac{V_B}{R_D}$ and thus less dependent on β. We now know the collector voltage: $V_C = R_F I_{R_F} + V_B$, and can thus determine the emitter current.

$$I_E = -\left(\frac{V_{CC} - \frac{V_B(R_C+R_F+R_D)}{R_D}}{R_C}\right)\left(1 + \frac{1}{\beta}\right) \tag{12}$$

$$V_{CE} = V_{CC} - R_C\left(-I_E + \frac{V_B}{R_D}\right) \tag{13}$$

Again, these equations are good for analysis, but are too complicated for design. To make life simpler, we can choose values for R_C and R_F using the following equations:

$$R_C = \frac{V_{CC} - V_{CE}}{I_E + V_B/R_D} \tag{14}$$

$$R_F = \frac{V_{CE} - V_{BE}}{V_B/R_D + I_B} \tag{15}$$

Where $V_B = V_{BE} + I_E R_E$.

In most cases, the designer (YOU!) can choose V_{CE}. A good choice generally lies slightly above halfway between $V_{CC} - V_{BE} - V_E$ (not $V_E E$!). This minimizes distortion due to supply clipping or transistor saturation, while leaving some room for biasing.

As mentioned previously, we want to separate our RF circuit from our DC circuit as much as possible. First, we don't want any DC offsets in our input signal or at the input to the next stage to interfere with out biasing. Second, some of the DC resistor values may be different from those at RF. (e.g, in many cases, $R_{F,dc} < R_{f,rf}$, or $R_{E,dc} > R_{e,rf}$) We can solve these problems by using bypass and coupling capacitors, as in Figure 5. (BFC = "Big, Fat Capacitor") The capacitors are chosen such that they have a small impedance in the frequency range of interest, and thus do not affect the RF circuit.

The last issue we need to address is loading by R_C and R_D, which we did not account for when discussing the RF circuit. If unaccounted, the loading will decrease gain and degrade both the input and output match. Fortunately, the solution to this problem is fairly simple. All we need to do is modify R_in and R_out to make sure that the source and load see a matched impedance.

To do this we note that the source sees $R_D \parallel R_{in}$ and the load sees $R_C \parallel R_{out}$ at the input. To match this, simply make each of the quantities equal to the source and load impedance, respectively:

Input	**Output**
$\frac{1}{R_s} = \frac{1}{R_{in}} + \frac{1}{R_D}$	$\frac{1}{R_l} = \frac{1}{R_{out}} + \frac{1}{R_C}$
$R_{in} = \left(\frac{1}{R_s} - \frac{1}{R_D}\right)^{-1}$	$R_{out} = \left(\frac{1}{R_l} - \frac{1}{R_C}\right)^{-1}$
$R_{in} = \frac{R_D - R_s}{R_D R_s}$	$R_{out} = \frac{R_C - R_l}{R_C R_s}$

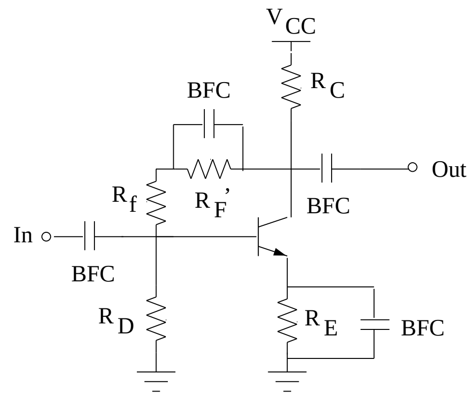

Figure 5: Complete Amplifier, with bypass and coupling capacitors. ($R'_F = R_{F,dc} - R_{f,ac}$)

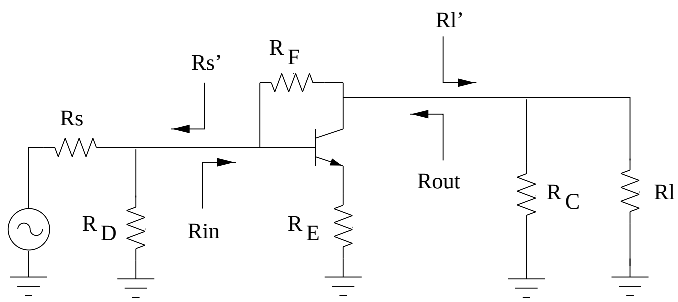

Figure 6: The Loaded RF Circuit

Note that if $R_s = R_l$, we must have $R_C = R_D$ in order to match both the input and output. Since we now have $R_{in} \neq R_{out}$, we cannot use Equations 10 and 11. To keep your design process simple, first use the Miller Effect along with your gain specification to determine R_f. From there, you can solve Equation 5 for $R_{E,rf}$. Once you have R_E, use Equation 3 to check that R_{in} is correct. (Be sure to replace R_l with R'_l.)

5 Design Steps

Here is a summary of design steps, some of which may be performed out-of-order:

- Choose R_C and R_D, and determine R_{in} and R_{out} based on your specifications for R_s and R_l.
- From your specifications for power dissipation and supply voltage, choose I_E, keeping in mind that $P_d = V_{CC}\,(I_E + I_{R_D})$.
- Choose V_{CE}
- Determine $R_{F,dc}$ and $R_{E,dc}$.
- Design RF circuit.
- Simulate in SPICE, make sure that it confirms your hand calculations. If not, check your SPICE deck and calculations before you change ANYTHING. Find out where the discrepancy comes from.

Congratulations! You are now on your way to becoming a true-blooded RF engineer.
Good luck, and HAVE FUN!