SPICE BJT Project (Extra Credit)

Chris Winstead

ECE 3410. Spring, 2015.
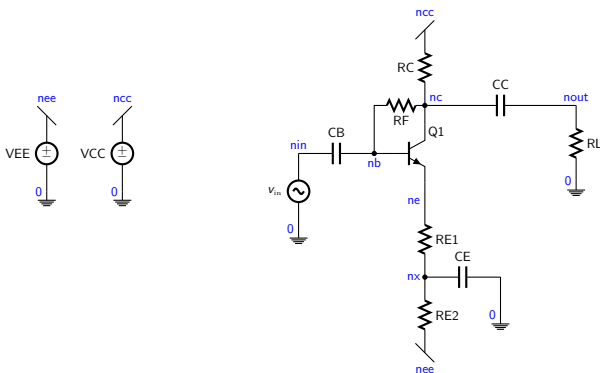
Preparing for the Exercises

Create a new directory for this session:

```
3410/
└── spice/
      └── lab1/
      └── lab2/
      └── lab3/
      └── lab4/
      └── lab5/
      └── lab6/
```

Common Emitter Circuit

In this session, we will design, analyze and verify a BJT Common-Emitter Circuit with Collector-Base feedback bias:

With device values: VCC=10 V, VEE=−10 V, $I_E = 10$ mA, Q1=2N2222 NPN, RF=RL=10 kΩ, VIN=10 kHz sinusoid with a zero-to-peak amplitude of 50 mV.

Design Objectives

The target specification for this design is to achieve a Gain of $-20\,\mathrm{V/V}$.

Since the coupling capacitors act as high-pass filters, we also need to select a minimum frequency in order to solve values for those capacitors. We will target a minimum frequency fmin=100 Hz.

Create a SPICE File

Create a SPICE file called ce_amp.sp to describe the circuit shown on the previous slide. Use parameters to stand in for the unassigned component values RC, RE1, RE2, CB, CE and CC.

Declaring BJT Devices

The physical model for the 2N2222 NPN device is already provided in the
lab_parts.md file. As usual, you should include it near the top of your SPICE file:

```
.include ../lab_parts.md
```

As with MOSFETs, the actual model definition contains many parameters that
influence the real device's behavior:

```
.model Q2N2222 NPN (Is=14.34f Xti=3 Eg=1.11 Vaf=74.03 Bf=255.9 Ne=1.307
+Ise=14.34f Ikf=.2847 Xtb=1.5 Br=6.092 Nc=2 Isc=0 Ikr=0 Rc=1
+Cjc=7.306p Mjc=.3416 Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 Vje=.75
+Tr=46.91n Tf=411.1p Itf=.6 Vtf=1.7 Xtf=3 Rb=10)
```

This BJT device has $\beta = 256$, indicated by the "Bf" parameter (this is an ideal
maximum; the real β will often be smaller). The Early voltage (parameter Vaf) is
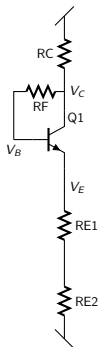74.03 V.

To place a BJT instance in your circuit, the name begins with 'Q', followed by the
terminal nodes and the model name:

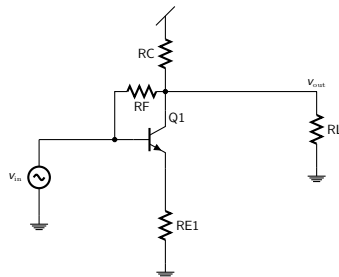```
Q1 <ncollector> <nbase> <nemitter> Q2N2222
```

# Analyzing the Circuit

Thanks to the capacitive coupling and bypass methods, the circuit analysis can be split into two parts, the AC circuit and the DC circuit:



DC Equivalent

AC Equivalent

# Beginning the Analysis

To prepare your analysis, create a Matlab script called `analysis.m`, and enter all of your calculations into that script. This is a good habit to adopt, since it makes it easier to correct mistakes and to experiment with different component values or specifications.

At the top of your script, enter the device parameters ($\beta$ and $V_A$), the given component values (VCC, VEE, RL), and the target specifications (Gain and fmin).

# Analyzing the AC Behavior

Our circuit is greatly simplified under the following conditions:

- RC is much smaller than the resistance seen into the collector.
- RC is much smaller than the feedback resistance RF.
- RC is much smaller than the load RL.

Under these conditions, the total resistance seen above the collector is approximately RC. Due to the bypass capacitor CE, at high frequencies RE2 is shorted out. The total small-signal resistance under the emitter is therefore RE1, so the gain is

$$A_v \approx -\frac{RC}{RE1}.$$

To keep the resistor values small, we may choose RE1=50 Ω. You may then solve the value of RC needed to get a gain of −20 V/V. Write this calculation into you Matlab analysis script.

# Small-Signal Characteristics

In order to validate our conditions, we need to obtain the small-signal transconductance and the resistances seen looking into the collector and base terminals. Since $I_E$ is given as a design specification, we can obtain these values using the equations derived in class:

$$g_m = \frac{I_C}{U_T} = \frac{\alpha I_E}{U_T}$$

$$r_o = \frac{V_A}{I_C} = \frac{V_A}{\alpha I_E}$$

Once these values are obtained, the resistance into the collector is affected by RE1 (note that RE2 is masked by the bypass capacitor):
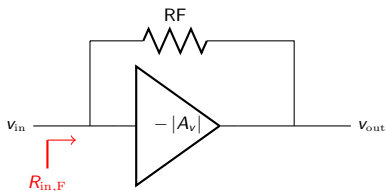
$$R_{\text{coll.}} = R_{E1} + r_o + g_m r_o R_{E1}$$
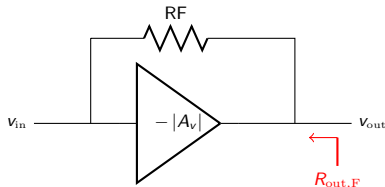
$$R_{\text{base}} = r_\pi + (\beta + 1) R_{E1}$$

Calculate these values and verify that $R_C \ll R_{\text{coll.}}$.

# Effect of the Feedback Resistor I

When RF is placed between the amplifier's input and output, it may have some consequences for the input and/or output resistance. These effects are analyzed using a simplified technique:



$$R_{\mathrm{in,F}} = \frac{v_{\mathrm{in}}}{i_{\mathrm{in}}}$$

$$i_{\mathrm{in}} = \frac{v_{\mathrm{in}} - v_{\mathrm{out}}}{R_F}$$

$$v_{\mathrm{out}} = -|A_v|\, v_{\mathrm{in}}$$

$$\Rightarrow i_{\mathrm{in}} = v_{\mathrm{in}} \frac{1 + |A_v|}{R_F}$$

$$\Rightarrow R_{\mathrm{in,F}} = \frac{R_F}{1 + |A_v|}$$

$$R_{\mathrm{out,F}} = \frac{v_{\mathrm{out}}}{i_{\mathrm{out}}}$$

$$i_{\mathrm{out}} = \frac{v_{\mathrm{out}} - v_{\mathrm{in}}}{R_F}$$

$$v_{\mathrm{in}} = -\frac{v_{\mathrm{out}}}{|A_v|}$$

$$\Rightarrow i_{\mathrm{out}} = v_{\mathrm{out}} \frac{1 + 1/|A_v|}{R_F}$$

$$\Rightarrow R_{\mathrm{out,F}} = \frac{R_F}{1 + 1/|A_v|}$$

# Effect of the Feedback Resistor II

Interpretations:

- On the input side, $R_{\mathrm{in,F}}$ appears as a parallel resistance that may lower the input resistance if it is lower than $R_{\mathrm{base}}$.

- On the output side, $R_{\mathrm{out,F}} \approx R_F$.

To estimate the values of $R_{\mathrm{in,F}}$ and $R_{\mathrm{out,F}}$, you may assume that your design will be successful and use $|A_v| = 20$.

# Input Coupling

On the input side, the main coupling effect is a high-pass filter formed between capacitor $C_B$ and the total equivalent base resistance $R_{B,\text{tot}}$. The total base resistance is

$$R_{B,\text{tot}} = R_{\text{base}} \parallel R_{\text{in,F}}$$

The lower cutoff frequency should be equal to our desired fmin. Based on this constraint, $C_B$ can be solved as

$$C_B = \frac{1}{2\pi f_{\min} R_{B,\text{tot}}}$$

Complete this calculation in your analysis script.

# Output Coupling

On the output side, the coupling effect is a high-pass filter formed between capacitor $C_C$ and the total equivalent collector resistance $R_{C,\text{tot}}$. The total collector resistance is

$$R_{C,\text{tot}} = R_C \parallel R_L \parallel R_{\text{coll.}} \parallel R_{\text{out,F}}$$

The lower cutoff frequency should be equal to our desired fmin. Based on this constraint, $C_C$ can be solved as

$$C_C = \frac{1}{2\pi f_{\min} R_{C,\text{tot}}}$$

Complete this calculation in your analysis script.

# Analyzing the DC Behavior

The DC behavior is relatively straightforward. Since $I_E$ is given, we can easily solve $I_C = \alpha I_E$ and $I_B = I_C/\beta$.

Then the voltages $V_C$ and $V_B$ are obtained by accounting for the voltage drops across RC and RF, respectively. Note that the current passing through $R_C$ is $I_C + I_B$, and the current passing through $R_F$ is $I_B$. Furthermore, since $v_{BE} \approx 0.7\,\text{V}$, $V_E$ can be calculated from $V_B$.

Finally, the remaining voltage drop $(V_E - V_{EE})$ has to be satisfied across $R_{E1} + R_{E2}$ with the specified value of $I_E$. Based on these considerations, solve for $R_{E2}$.

Place these equations in your `analysis.m` file.

# Emitter Bypass Coupling

The emitter coupling is determined by $C_E$ together with the total resistance seen at node nx, including RE2 (RE2 can be neglected in the pass-band, but not when determining the cutoff frequency):

$$R_{E,\text{tot}} = \left( R_{E1} + \frac{\alpha}{g_m} \right) \parallel R_{E2}$$

$$C_E = \frac{1}{2\pi f_{\min} R_{E,\text{tot}}}$$

# Consistency Checks

Once all the solutions are entered into your Matlab script, run it and examine the results. Verify the following assumptions:

- $R_C \ll R_{\text{coll.}}$ and $R_C \ll R_{\text{out,F}}$
- $C_E$, $C_B$ and $C_C$ are realistic values (they should be in the range of nano- or micro-farads).
- $V_C - V_E > 0.3\,\text{V}$.
- Calculate the mid-band gain as $-g_m R_{C,\text{tot}}/(1 + g_m R_{E1})$, in both V/V and dB. It should be close to the desired gain of $-20\,\text{V/V}$.

It might be convenient to place these consistency checks in your Matlab script, so that it automatically prints warning messages if any condition is violated. As usual, record your calculated values in a `log.txt` file.

# Verification: DC Operating Point

Next, enter your calculated values into your SPICE file. Run an operating point simulation using the "OP" command.

View the results with the "`print all`" command, and compare them to your calculations.

They will not be an exact match, but should be in the same general ballpark. Record these and all further measurements in your `log.txt` file.

# Verification: Transient Simulation

Next, run a transient simulation. After the simulation, use the measurement commands to evaluate the gain, like this:

```
meas tran vipp PP v(nin)
meas tran vopp PP v(nout)
let Gain=vopp/vipp
print Gain
```

Also plot v_{in} and v_{out} together. You should see something like this:
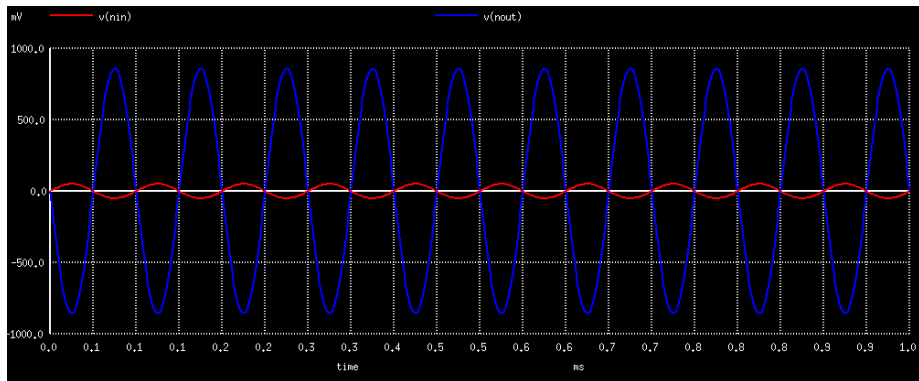
Verification: AC Simulation

Next perform an AC simulation. For this to work, remember that VIN needs to be declared both as a SIN voltage source (for transient simulation) and an AC voltage source with magnitude 1. In other words, your source declaration should look like this:

```
VIN nin 0 DC 0 AC 1 SIN(0 .05 10k)
```

And your AC simulation command line should declare a sweep by decades, with 10 points per decade, from 10 Hz to 100 MHz, like this:

```
AC dec 10 10 100e6
```
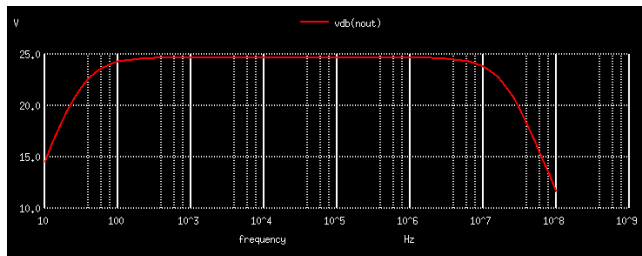
Practical Implementation

As a final step, round off all the calculated resistor and capacitor values so that they can be easily implemented with standard values, like $50\,\Omega$, $100\,\Omega$, $333\,\Omega$, $1\,\mu F$, $10\,\mu F$, etc.

Repeat the simulation and verify that you can obtain results close to the desired gain and bandwidth (you can be the judge of how close is good enough).

Record the final component values in your lab book, along with the gain and frequency response predicted by SPICE.

What to Turn In

Create a zip file containing the following files:

- ce_amp.sp
- analysis.m
- log.txt
- Hardcopy plots from the transient and AC simulations.

Upload these files to Canvas to complete the assignment.