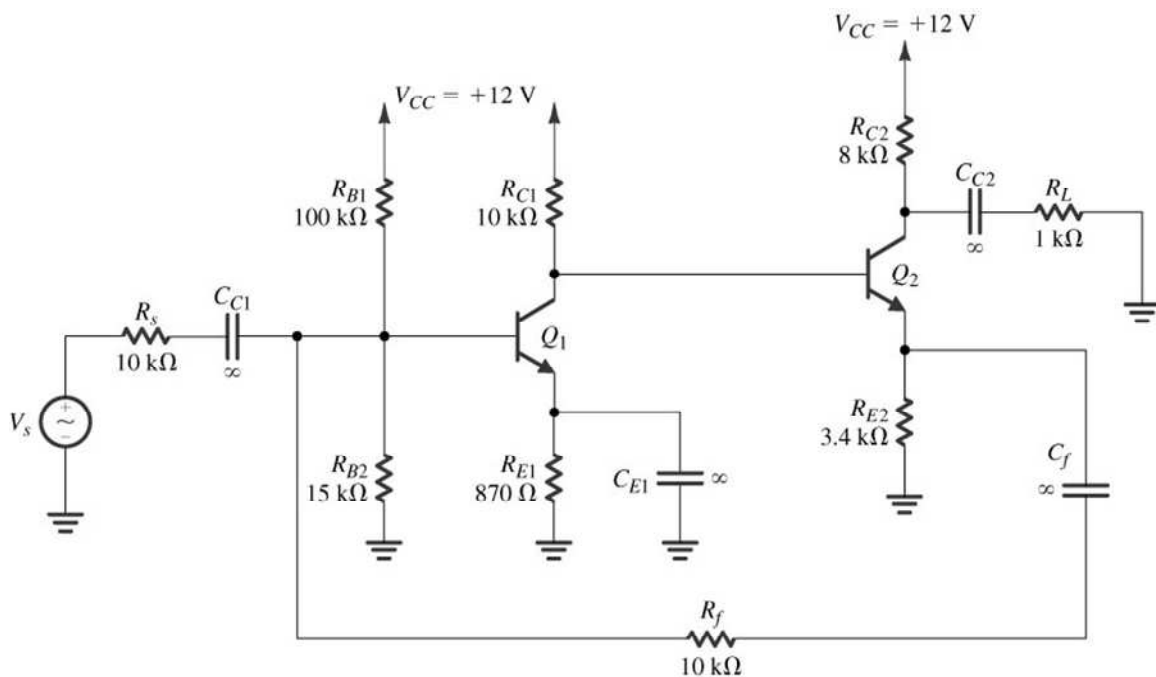# Lab 3   Feedback Amplifiers

Due 4/5/2016 before class

In this lab, we will analyze feedback amplifiers using SPICE simulation. The device models are from the textbook device library **sedra_lib.lib** as in the previous labs. You need analyze the circuits *analytically* first, and then start the simulation.

I. Two-Stage Feedback Amplifier

First, we analyze a two-stage shunt-series feedback amplifier shown above. The circuit is from Sedra&Smith 5th ed., Example 8.4 and 8.7. It uses two NPN bipolar transistors 2N3904 from the textbook device library **sedra_lib.lib** with the following parameters: $\beta=100$, $V_A=75V$. All capacitors and resistors are ideal.

Tasks:

1) Follow the analysis in Example 8.4. How does the amplifier characteristics change when $\beta=416.4$, the value given in the model, instead of the assumed $\beta=100$? What's your conclusion from the observation?

2) Create the SPICE netlist for the circuit. Note that you need to use some practical values for the coupling capacitors like $10\mu F$, instead of infinite. What do you think this change will affect the amplifier characteristics?

3) Run a DC analysis. Verify the dc bias and small-signal parameters matches with Example 8.4.
4) Find the 2-port network parameters for the feedback network.
5) Construct the "new" amplifier without feedback, and find its gain, input and output resistance. Verify that all matches with Example 8.4. Note you need do this using simulations.
6) Following Example 8.7, find the loop gain and plot it in a Bode plot, including both amplitude and phase. Find the dc gain, 3dB bandwidth, phase and gain margin. Is this amplifier stable?
7) Run an AC analysis to find the frequency response of the feedback amplifier, sweeping from 1Hz to 1GHz. Find the small-signal resistance R_{in} and R_{out}.

II. Lab Report:

Draw the schematic of all circuits with clear component names and values. Explain your design. Report simulation results except waveforms. Attach printouts of the SPICE files and simulation results including spectrum and waveforms with clear explanations. Discuss any discrepancy from your expectation. Submit all SPICE files to TA by email. Note that your SPICE file should follow the conventions in the lecture.