

ECE 2100 Spice Project #S2 Help

A. Stolp
4/12/02, 3/13/03

- a)** Draw your circuit with capture or schematics. Remember to name the ground "0". Use the part values that you actually used in the lab if they differ from those shown in the handout. The model for the 2N3904 transistor is in the eval library as Q2N3904. I used VSIN for the v_s .
- b)** To check or modify the transistor model: Highlight the transistor. Edit -> PSpice Model or right-click on the highlighted part and select "Edit PSpice Model". Look for the following parameters: Cjc and Cje. These should be 3 to 4 pF and 4 to 5 pF respectively. I used the default values of Cjc=3.638p and Cje=4.493p. Leave the model editor open.
- c)** Find the Bf parameter, Bf stands for the "forward Beta". Change it to 100 and save the file.
- d)** The following paragraphs detail how I found R_{in} and the frequency response, your methods may differ.

i) I Set the VSIN parameters, then set up a transient analysis (PSpice -> Edit Simulation Profile). I ran the analysis to 40 ms but only started to collect data at 38 ms. That way the true transients should have settled out, leaving me with a steady-state response.

When I got a good plot showing no clipping or distortion, I found the amplitude of the source current. Trace -> Eval Goal Function, I chose the goal function on the right, Max(), and then the source current on the left, I(Vs), to get Max(I(Vs)) in the window at the bottom, then hit OK. (I had named my VSIN source "Vs".) this gave me a peak source current value. I divided the amplitude of my VSIN source by this current amplitude and then subtracted the value of R_s to get the input resistance, R_{in} . For a better number use the Max goal function twice to get Max(V(Vs+))/Max(I(Vs)). Don't forget to subtract R_s . You can also try Max(V(Rs:2))/Max(I(Vs)) or Max(V(Rs:1))/Max(I(Vs)). If you get the right one, you won't have to subtract R_s .

You could also place another resistor in the circuit (say 500 to 1 k Ω) in series with the input. Then use the two voltages on the two sides of this resistor to find R_{in} .

ii) I edited the simulation profile and changed the analysis type to "AC Sweep/Noise". I set the AC Sweep Type to logarithmic, decade, starting at 10 and ending at 100Meg, with 10 points per decade. (Note: in SPICE, M = m = milli, Meg = mega).

Now, if you just run the simulation this way, you'll get errors. To do an AC analysis, the circuit needs an AC source. The VSIN is a time domain/transient source. AC analysis won't recognize it unless you give it an AC value. This is a very weird aspect of SPICE that you must remember or it will give you lots of trouble. **Many parts, especially sources, need to have separate values for AC and DC or Transient analysis.** Sorry, I didn't write the program, I think it's confusing too.

I selected Vs, right-clicked, then selected Edit Properties. I selected the AC property and set it to 1. This will be the source amplitude used for the AC analysis, not the VAMPL. Yes, I know that the output would be horribly clipped and distorted if the input amplitude were 1 V, but SPICE doesn't seem to care in AC analysis. Then I hit PSpice -> Run.

The frequency plot I got will meet the basic requirements of this assignment. You can take data from this plot to find the upper and lower cutoff frequencies (try Trace -> Cursor -> Display) and be done. You could make a little better plot by changing the axis settings (under Plot) or you can get a really nice plot (at least you can in PSpice 9.2). Try this:

Window -> Display Control -> Templates -> Bode Plot - dual Y axis -> Restore. Hit the little gray and red symbol to pick the voltage or current that you want to display. Now you'll get a nice Bode plot.

Try some of the other plot types. In particular, try Impedance. Use VB(Q1) as the voltage and I(Rs) as the current. You'll get a plot of the input impedance as a function of frequency. If you put the cursor at a midband frequency, you'll get the midband R_{in} . The input impedance is very high at low frequencies because R_E is no longer bypassed and it drops off at high frequencies due to the capacitance between the collector and the base inside the transistor and the miller effect.

e) Expect the R_{in} that you measured in the lab to be somewhere between the two you get from the simulation. Same for the low corner frequency (f_{CL}). However, expect the f_{CH} from SPICE to be considerably larger than what you measured in the lab. You are not modeling the proto-board capacitance, lead capacitance, or scope probe in your SPICE model. It would be interesting to try this, if you want to.

f) Print the schematic and then look at the netlist (PSpice -> View Netlist) to figure out the names of all the nodes in your circuit. Label the nodes on your print-out. The purpose of this is so you can see how the netlist corresponds to the circuit. The netlist is the normal input to any SPICE program, many of which don't have fancy schematic drawing programs. Edit the netlist as necessary to print out **only** the part of the netlist that shows the node numbers.

If you look on my web site, you'll find an example of what I think you should turn in, just a few pages— 1 schematic, a small piece of the netlist, 2 frequency curves, & a comparisons table.