# Problem Set #8

Assigned: Tuesday, November 14, 2000
Due: Wednesday, November 22, 2000 at Recitation

---

| **Reading Assignments:** | 11/14/00 | Sections 8.5 – 8.7 of Howe & Sodini |
|---|---|---|
| | 11/16/00 | Sections 8.8 – 8.9 of Howe & Sodini |
| | 11/21/00 | Sections 10.1 – 10.4 of Howe & Sodini |

---

PLEASE WRITE YOUR RECITATION SESSION TIME ON YOUR PROBLEM SET SOLUTION

1. [50 points] **DC SPICE parameter extraction of an npn-BJT**

This is an exercise in which you use an experimental ***Web-based Microelectronic Device Characterization*** system to characterize an npn-BJT (http://weblab.mit.edu). Professor Jesus del Alamo and his students developed the system. From the measurements, you also extract the DC SPICE parameters of the transistor. The User Manual is available on the ***weblab*** homepage.

In this problem, you will characterize an npn-BJT. The details of how to select and connect the device are available online. Select one of the devices labeled "**6.012 npn BJT**" under the "**Device**" Menu. Take the measurements specified below and download the data to your local machine for additional graphing and further analysis. *Only download the data when you have satisfactory results as would be indicated by the characteristics obtained and displayed through the web*.

For all the following measurements, hold $V_{BE}$ (or $V_{BC}$) between 0 and 0.9 V, and $V_{CE}$ (or $V_{EC}$) between 0 and 4 V. For the SPICE parameter determination you will need the following structural information about the transistor. Here is your assignment.

a. (5 points) Measure and download the common emitter *output characteristics* of the transistor. These are $I_C$ vs. $V_{CE}$ with $I_B$ as parameter ($0 < I_B < 160$ μA). In your local machine and using your favorite software tool, graph the output characteristics. Turn in a printout of this graph.

b. (10 points) Measure and download the common emitter *transfer characteristics* (Gummel Plot) of the transistor in the forward active region. This is a plot of $I_C$ & $I_B$ vs. $V_{BE}$ with $V_{CE}$ = 2.5 V. In your local machine, graph the transfer characteristics and current gain in the forward active region vs. collector current. See pp. 14-15 of Lecture 16. Turn in a printout of these graphs.

c. (5 points) Measure and download the *reverse output characteristics* of the transistor. These are $I_E$ vs. $V_{EC}$ with $I_B$ as parameter ($0 < I_B < 160\ \mu A$). In your local machine and using your favorite software tool, graph the output characteristics. Turn in a printout of this graph.

d. (10 points) Measure and download the *reverse transfer characteristics* (Reverse Gummel Plot) of the transistor in the reverse active region. This is a plot of $I_E$ & $I_B$ vs. $V_{BC}$ with $V_{EC}$ = 2.5 V. In your local machine, graph the transfer characteristics and current gain in the reverse region vs. emitter current. See pp. 14-15 of Lecture 16. Turn in a printout of these graphs.

e. (5 points) From linear plots of $I_B$ vs. $V_{BE}$ and $I_B$ vs. $V_{BC}$, determine $V_{BE,\,on}$, $V_{BC,\,on}$ and $V_{CE,\,sat}$.

f. (5 points) From the *transfer characteristics (Gummel Plots),* extract the SPICE parameters **IS, BF and BR** for the transistor (See equations on Page 12 of Lecture 17 notes). You should be careful in selecting the values of BF and BR. For BR use the maximum value and for BF select a value corresponding to the “flat” part of the current gain vs. current plot.

g. (5 points) From the *output characteristics*, extract the SPICE parameters **VAF** for the transistor (See Section 7.5 of text and recitation on 11/08/00).

h. (5 points) Using the SPICE parameter set just derived, play back the characteristics of the transistors (See equations on Page 12 of Lecture 17 notes and Equation 7.65 of Howe and Sodini) and compare them with the measurement data. The most effective way to do this is to construct graphs that depict the measured data as individual dots and the model as continuous lines.

- Graph together the measured common emitter *output characteristics* of the npn-BJT and those predicted from the SPICE model. Turn in this graph. Comment on the accuracy of the model.

- Graph together the measured *transfer characteristics (Gummel Plots)* of the npn-BJT and those predicted from the SPICE model. Turn in this graph. Comment on the accuracy of the model.

The SPICE parameter determination should not demand extensive numerical analysis. There is no need to do regressions or least-squares fits. The graphs you have to turn in need not be too sophisticated, just simply correct. They must have proper tick marks, axis labeling and correct units. If there are several lines, each one should be properly identified (handwriting is OK).

Additional information and assorted admonitions:

The system will be up between 8 AM on Wednesday November 15 and 8 AM on Wednesday November 22. The systems will be shut down after that. For short periods of time, the system may be down for maintenance and to change the device, if needed.

If you encounter network and system problems, please contact hardison@mit and cc jkfioren@mit.edu and akinwand@mtl.mit.edu. For other problems related to this homework, contact jkfioren@mtl.edu and cc akinwand@mtl.mit.edu.

The device is real and it can be damaged. Please be careful with the voltages that you present to this device. If the characteristics look funny, let us know. The device might have been damaged. We will replace it as soon as we realize it is damaged. Note that at any time, there are two devices you could characterize. The devices are labeled "**6.012 npn BJT**" and they could be selected from under the "device" menu. If one of the devices appears damaged, check the other device.

The login page uses your Athena username (as provided by the Registrar) as username and your MIT ID as password. You should login as soon as you can to confirm that you have access to the system. If you have login problems, contact hardison@mit.edu

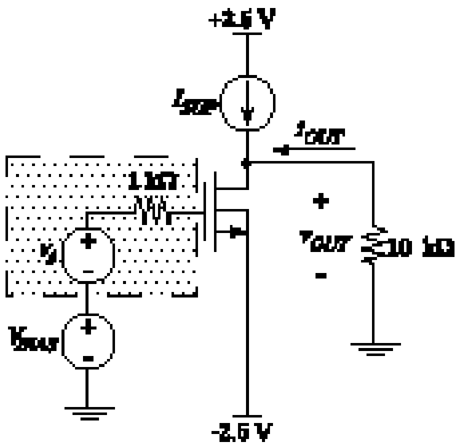
You should try to do the experimental portion of this homework early. We have built a queuing system in the server but we have never tested it with these many users. Besides, if the queue is long, you might need to wait for a while before you order is executed. It is to your advantage to get the data early and to download it to your local machine. Later on, you can work on parameter extraction and the rest of the homework.

The system keeps a log of all logins and the scripts that each user executes.

This is an experiment. Please be patient and give us plenty of feedback. If you are interested in participating in the further development of this system, please approach Prof. del Alamo (alamo@mit.edu).

2. [50 points] **CS Amplifier Design**

You are given a CS amplifier and NMOS device parameters shown below. The current source supply provides 250µA and has an infinite output resistance, (i.e. $I_{SUP}$ = 250 µA and $r_{oc}$ -->∞). The current source supply must have at least 0.5 V across it in order to maintain the high output resistance.




$W/L$ = 150/1.5
$V_{Tn}$ = 0.7 V
$m_n C_{ox}$ = 50 µA/V$^2$
$t_{ox}$ = 15 nm
$\lambda_n$ = 0.067 V$^{-1}$ @ $L$ = 1.5 µm

a. Calculate $V_{BIAS}$ such that $V_{OUT}$ = 0 V.

b. Draw the two-port model and calculate the two-port parameters $R_{in}$, $R_{out}$, and $A_v$.

c. Calculate the overall voltage gain $v_{out}/v_s$.

d. Calculate the output voltage swing.

e. Run a DC SWEEP in SPICE to find the value of $V_{BIAS}$ necessary to operate in the middle of the high gain region and the output voltage swing.

f. Use the value of $V_{BIAS}$ found in (e) to perform a small analysis of the amplifier including the source and load resistors and report $A_v$.