ECE – 532 Hspice Tutorial

I. **The purpose of this tutorial is to gain experience using the Hspice circuit simulator from the Unix environment. After completing this assignment, the student should be able to:**

    A. Use Hspice to find the operating point for an analog circuit.

    B. Use Hspice to plot an amplifier's open-loop frequency response and be able to measure low frequency gain, gain bandwidth product, phase margin, and gain margin.

    C. Use Hspice to plot the closed-loop transient response of an amplifier, and be able to plot and measure both the large-signal (e.g. slew-rate) and small-signal (e.g. 10-90 risetime) step response.

II. Preliminary Setup

    A. First you must add Hspice to your search path (all commands to be entered from the unix prompt, from now on assume that the '%' symbol refers to the unix prompt)

> *% source ~cad/.cshrc*
>
> *% synopsys_tools*

    B. Next create a simulation directory and copy over the required simulation files (start from home directory)

> *% mkdir sims*
>
> *% cd sims*
>
> *% mkdir spice_tutorial*
>
> *% cd spice_tutorial*
>
> *% cp ~scterry/analog/sims/ece532/test.sp .*
>
> *% cp ~scterry/analog/sims/ece532/cmosamp.spice .*

C. Finally create a models directory and copy over the model files (again start from your home directory).

% mkdir model

% cd model

% cp ~scterry/analog/model/ece532/*ami05u* .

D. Helpful Unix Commands

1. To edit a text document in Unix type (this assumes you have already sourced ~cad/.cshrc)

% edit filename

2. To print a text or postscript file from the Unix machines in the FH-508 electronics lab type

% lp –dfh507 filename

III. Operating Point Calculation

A. Background: When simulating analog circuits, it is very good practice to generate an operating point and study it carefully before performing an AC sweep, DC sweep, or transient simulation. The operating point shows all the bias voltages in the circuit, and also all of the important large- and small-signal parameters for the transistors. Almost all analog circuit design problems can be quickly found by examining the operating point.

B. First edit the file 'test.sp' so that the '.inc' statements point to the correct places in your account. Note that if you are including a file that is in the same directory as the '*.sp' file, then you only have to use the file name.

C. Fig. 1 shows the correct configuration for measuring the operating point of an amplifier. Confirm that the host-file reflects this configuration.

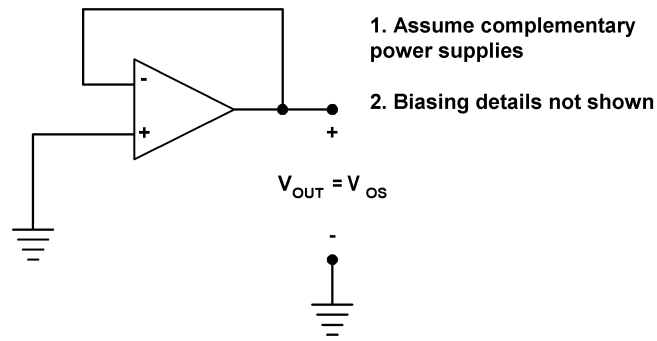D. To generate an operating point type at the command line:

1. Assume complementary power supplies

2. Biasing details not shown

$V_{OUT} = V_{OS}$

Fig. 1. Circuit for measuring an amplifier's operating point

% hspice test.sp > ! test.out &

E. **Assignment**: Open the file 'test.out' and use the information to construct a table showing these important characteristics:

1. Amplifier offset voltage

2. Drain current (I_D), drain-to-source voltage (V_{DS}), saturation voltage (V_{DSAT}), gate-to-source voltage (V_{GS}), transconductance (g_m), output resistance ($r_o = 1/g_{ds}$), and operating region for each transistor (find the inversion region and saturation region, also ignore the Hspice listed operating region because it might be wrong).

IV. **AC Analysis**

A. Background: AC analysis is used to plot an amplifier's frequency response. It can be used to find an amplifier's small-signal bandwidth, phase margin, common-mode rejection, etc. Note that an AC analysis is performed by linearizing a circuit about its bias point, and can therefore not give any information about non-linear effects such as slew-rate, distortion, output limiting, etc.

B. Preliminary: Edit the '.sp' file so the amplifier's open loop gain can be measured. Use the configuration shown in Fig. 2. Additionally you must add an AC sweep statement like this one
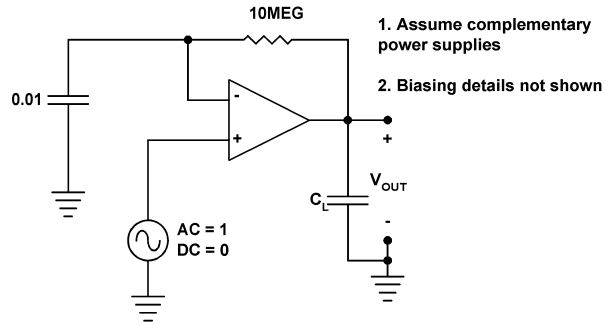
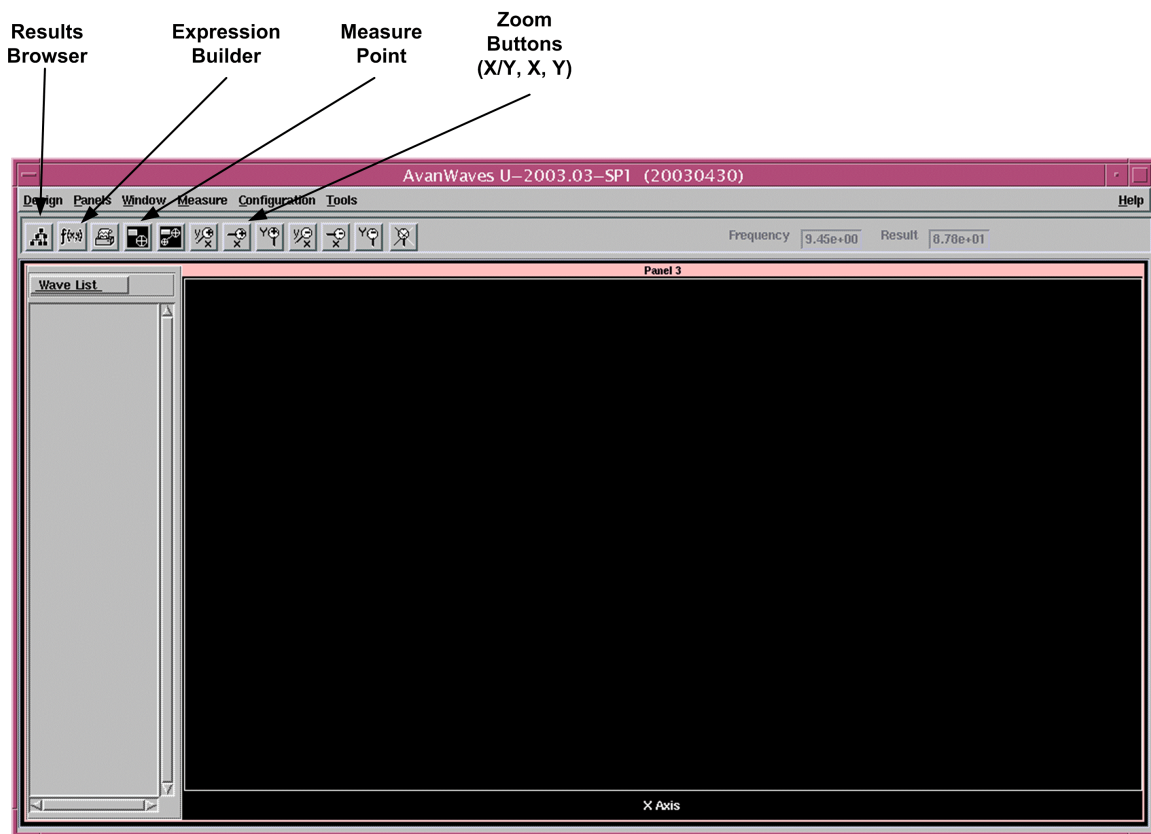**Fig. 2. Circuit for measuring an amplifier's AC response.**

Fig. 3. Awaves window with annotation

.AC DEC 10 1 10X

C. Running the simulation and plotting the output

1. Run the simulation from the command line as before. After the simulation is complete, you should always check your output file to be sure the simulation did not have any problems.

% hspice test.sp > ! test.out &

2. The program AvanWaves will be used to view the output. Fig. 3 shows a blank Awaves window with several important buttons annotated. To launch AvanWaves type at the command line

% awaves &

3. Once Awaves is open, click on 'DESIGN' and then 'OPEN'. Next click on 'test.sp' and the Results Browser will open automatically. In the Results Browser, click the words 'AC: ece-532 …' that appear next to the ac0 and all of the variables which are available for plotting will be shown.

4. First we will plot the magnitude of the frequency response. First click on the 'Expression Builder' button in the main window and then click the decibel function listed under the Functions column. Next go back to the 'Results Browser' and select 'Volts Mag' from the 'Types:' column. Then go the 'Curves:' column and middle click 'vout' and then drag and drop it into the expression field in the 'Expression Builder' window. Next you must give the expression a name (e.g. GAIN) in the Result field and hit return. To view the magnitude response, simply double click on the 'gain' expression in the Expressions column from the 'Expression Builder' window. You can now close the expression builder window.

5. Now we will plot the phase response. First go back to the main Awaves window and click 'Panels' and then 'Add'. Now go back to the 'Results Browser' window and under the 'Types:' column click 'Volts Phase.' Finally double click 'vout' in the Curves: column.

6. The next step is to clean up the plot so it will be readable. First, go the main Awaves window and group the two windows by clicking 'Panels' and then 'Group Panels' and then click on both panels so that the title of each is highlighted in pink. Next right click on the x-axis in each plot and select logarithmic. Often this will scale the x-axis so that it is not readable. To fix this problem, click on the 'Zoom x-axis' button and go to one of the plots. Left click in the plot near 1 Hz, and then hold and drag to the far right edge of the plot. The frequency axis of your plot should now be in decades and should run from 1 Hz to 10 MHz.

7. To measure the gain bandwidth product click on the 'Measure Point' button and then go the magnitude response plot. Sweep over the plot until you see the magnitude go to zero and left click at this point. Likewise to measure the phase margin click on the 'Measure Point' button and go to the unity gain crossover frequency and left click. To save your plot click on the Print button and select Print To: File and PS format and then click Print. Type in a file name (filename.ps) and make sure that the directory you are trying to write to is in your account. You can print this file directly from unix, or you can make it windows readable by typing at the command line

%ps2pdf filename.ps

8. My plot is shown in Fig. 4.

D. **Assignment**: Report the measured results for the previous step in tabular format, and print out a plot of the frequency response.

V. Transient Analysis

A. Background: Transient analysis models the time domain response of a circuit. It can be used to investigate large-signal effects such as slew rate limiting, clipping, distortion, etc. It can also be used to investigate small-signal effects, such as 10–90 risetime, overshoot, and settling time.

B. Small signal performance
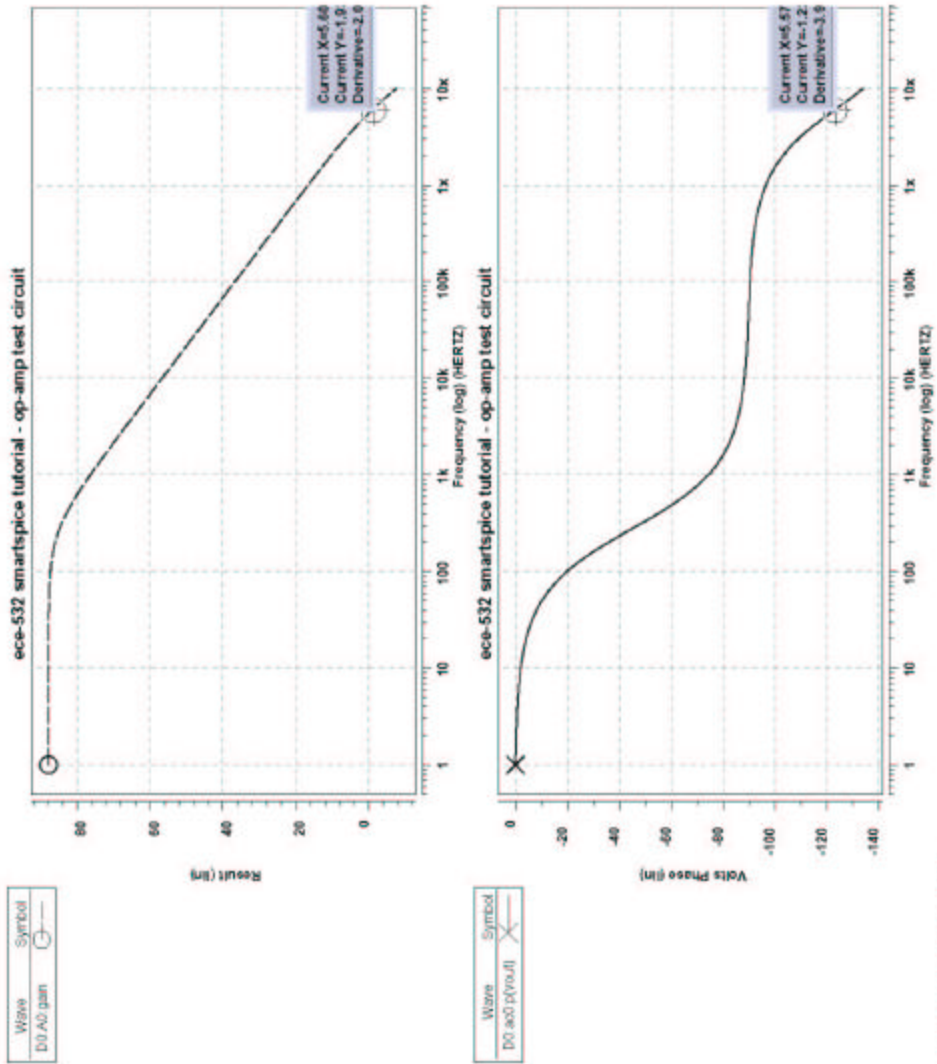
Fig. 4. Magnitude and phase Bode plot generated using Hspice

1. Edit the '.sp' file to measure the small-signal step response using the circuit shown in Fig. 1. Instead of grounding the positive input add a 50–mV pulse voltage source, such as in the following statement

   VIN   VINP   0   PWL(0 0 500NS 0 500.1NS 50MV)

2. Additionally you must add a transient statement

   .TRAN 1N 1000N 0 1N

3. Finally run the simulation from the command line and plot the result in Awaves. To plot the output signal, all you have to do is plot vout directly in the window without using the 'Expression Builder'. My result is shown in Fig. 5.

C. Large Signal Performance

1. The circuit for measuring the large signal transient response is shown in Fig. 6. An inverting gain configuration is used so that the amplifier will operate with a fixed input common-mode level, and the unity gain buffer is used to isolate the amplifier output from the feedback resistance. The buffer could be realized as a simple ESOURCE as

   E1   VOUTX   0   (VOUT,0)   1

2. To measure the slew rate generate a large input pulse (e.g. –2V to +2V). Simulate the circuit and plot the transient response as before. Simulate for both a positive and negative pulse. The following pulse statement can be used

   VIN   VINX   0   PWL(0 2 500N 2 500.1N 2 2500N 2 2500.1N –2)

D. **Assignment**: Generate plots for both the small-signal and large-signal pulse responses and measure the 10–90 risetime and positive and negative slew-rate. Report your results in tabular form.
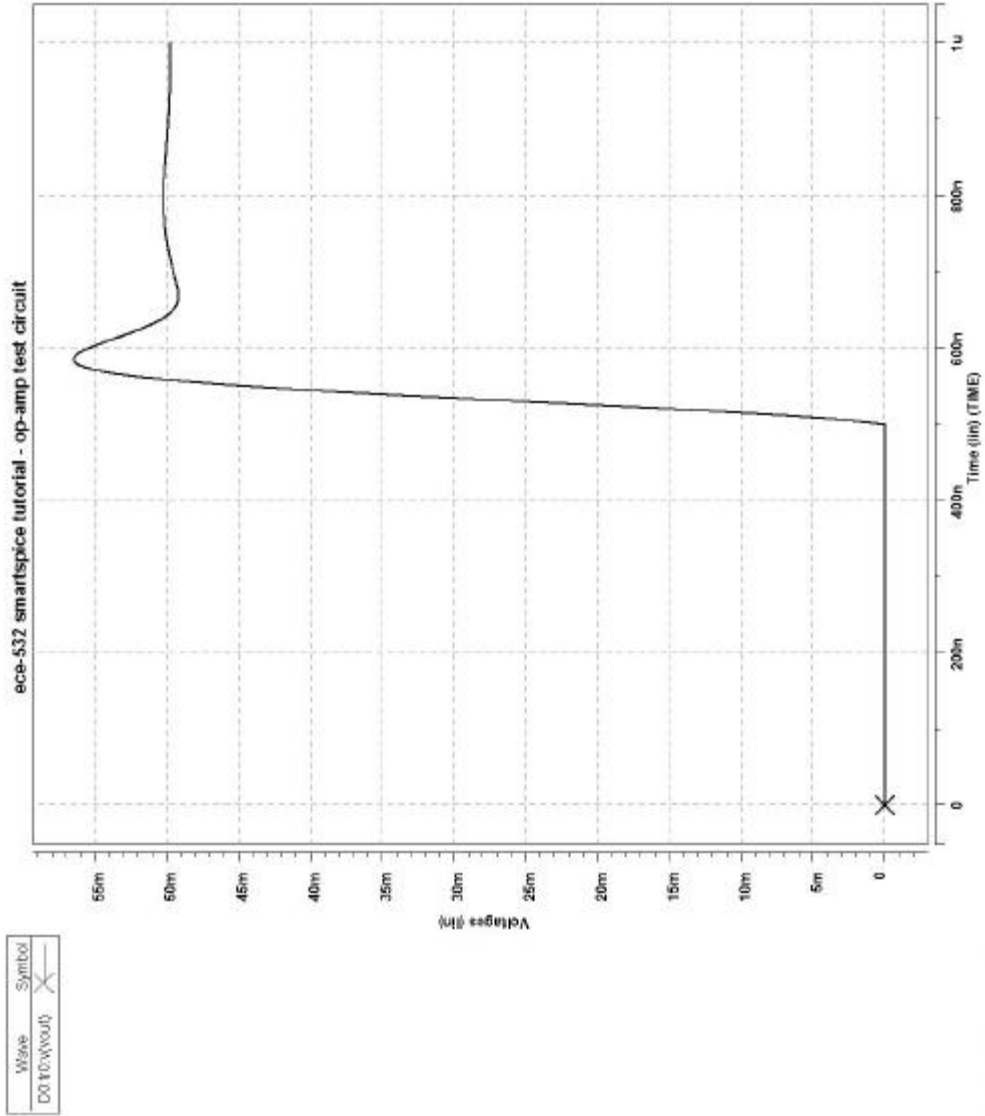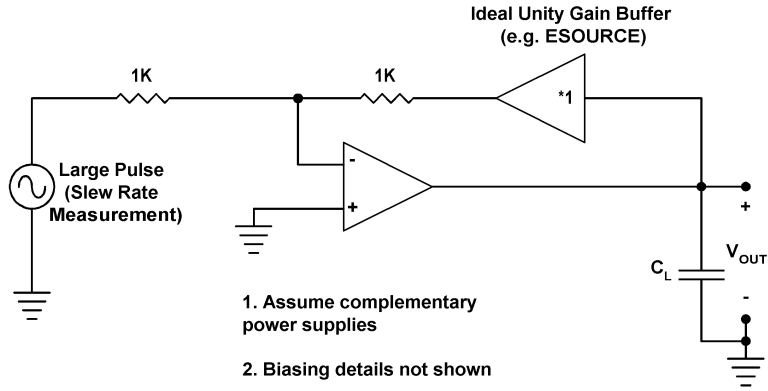
Fig. 5. Small-signal step response
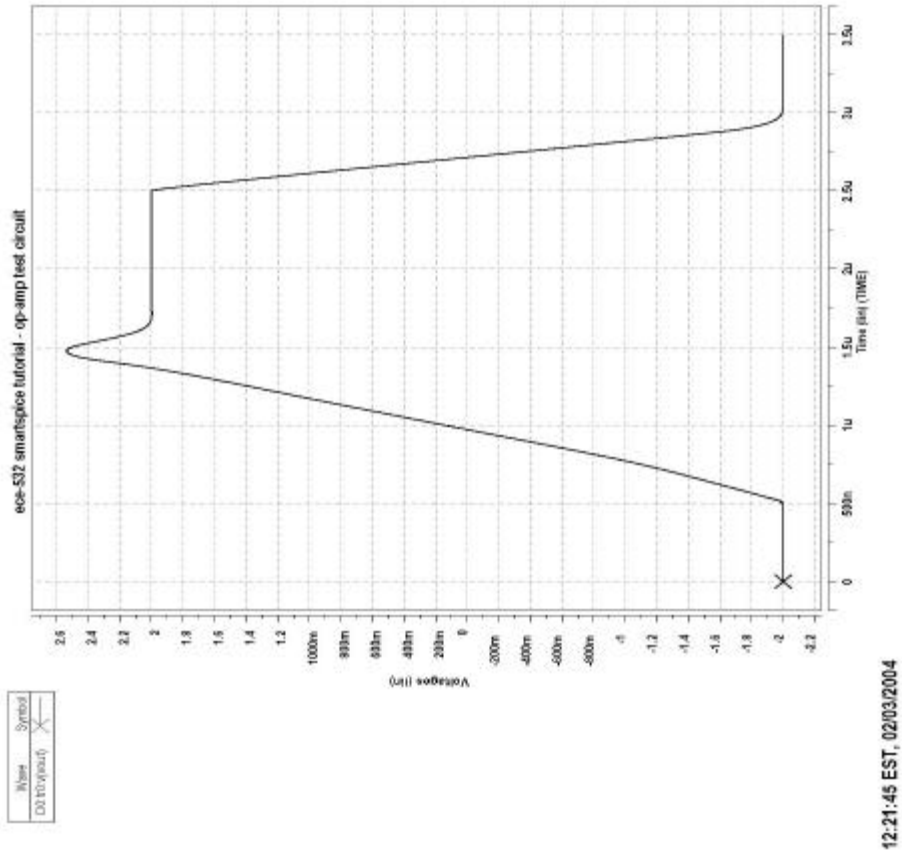
Fig. 6. Circuit for measuring an amplifier's slew rate

Fig. 7. Large-signal step response