ECE 304
Using PSPICE with WORD and EXCEL

Table of Contents

ECE 304: Using PSPICE with WORD and EXCEL

Introduction

This exercise familiarizes you with Microsoft WORD ′97 and 2000 and Cadence PSPICE, version 9.2[1]. Your job here is to duplicate the Table of Contents, all figure captions, and PSPICE and EXCEL figures (not the figures of screen dumps). This document reviews the basic plots you will need in this course. PSPICE background is in Herniter, *Schematic Capture with Cadence PSPICE[2].*

Using WORD is simplified if you open your new WORD file with FILE/NEW and select the template provided on the department computers named PSPICE.dot. This template includes some format styles and some macros for copy and paste that will make life easier for you.

Creating a Schematic

Figure 1 shows the schematic of a current mirror that we want to create. To get started you will have to read pp. 1-44 in Herniter, *Schematic Capture with Cadence PSPICE.* These pages describe how to open the program, select and wire parts.

The parts needed here are VDC (DC voltage sources), IDC (DC current source) resistors, zero ground connection, two parameter boxes (part PARAM), an AC voltage source (part VAC), a sinusoidal voltage source (part VSIN), and two transistors (part Q2N2222). Except for ground (0/SOURCE), which is selected using the GND [⏚] button from the right-hand toolbar, these parts all can be located using PLACE PART and typing the part name into the PART box. The attributes of the parts are set following the directions of Herniter, § 1.D., p. 19. Setting the attributes of the PARAM part is described in more detail on pp. 206-208. We use the PARAM part to set up variables to describe the current IREF of the DC current source, *etc.* These variables are then inserted in the corresponding part (like IDC) in curly braces {} to tell PSPICE that they are variable values. For example, we can run a number of *I-V* curves for the current mirror for different values of IREF by using the IREF parameter. Such a family of curves is generated in the last section of this document. For a different problem, such a set is described in Herniter § 4.D.5, p. 204.

We will use the circuit shown in the schematic of Figure 1 to illustrate a number of different types of PSPICE simulation.. Click on any figure label to jump to that figure.

1. Q-point (called a BIAS POINT SIMULATION) (Bias values determined as shown in Figure 1)
2. DC current vs. DC voltage (called a DC SWEEP) (Figure 9 and Figure 10)
3. DC differential output resistance vs. DC applied voltage (a DC SWEEP using a built-in PROBE function) (Figure 13)
4. Small-signal output resistance vs. frequency on a log scale (called an AC SWEEP) (Figure 16)
5. Transient output current in response to sinusoidal applied voltage of various amplitudes (called a TRANSIENT ANALYSIS) (Figure 19)
6. Small-signal output resistance at 1 kHz vs. emitter leg resistor value (called a PERFORMANCE ANALYSIS) (Figure 25)

---

[1] When you install PSPICE, take note of some corrections outlined in Appendix 1

[2] Updates and additional material can be found on Herniter's Web Site, http://www.rose-hulman.edu/~herniter/

7. DC compliance voltage vs. emitter leg resistor value (a DC SWEEP vs. a circuit parameter) (Figure 27)
8. DC output current vs. reference current for various values of emitter leg resistor (a DC DOUBLE SWEEP) (Figure 41)

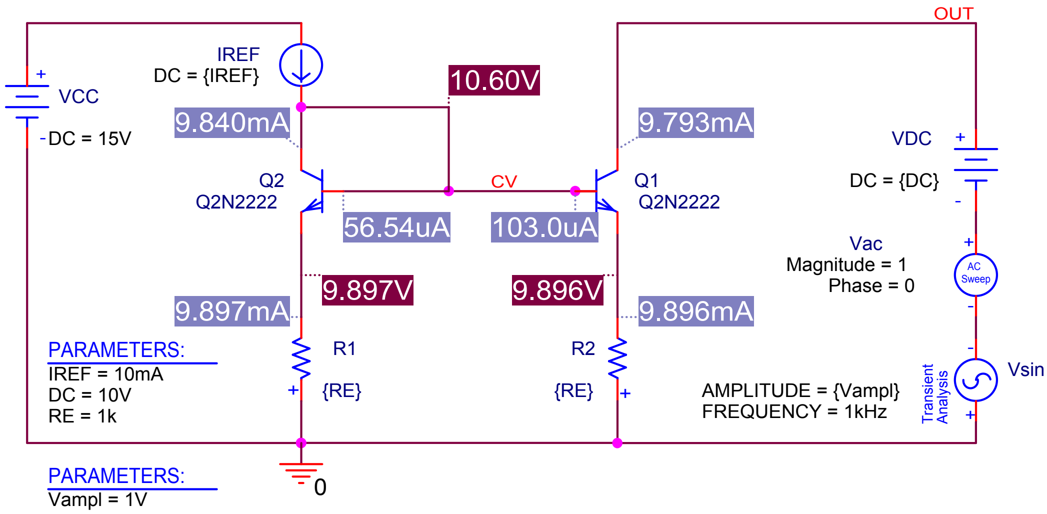


FIGURE 1:

Current mirror schematic. Note the emitter-leg resistors of value R_E = 1kΩ.

(Return to list of PSPICE simulation figures.)

Setting Font Sizes on the Schematic

You may wish to change the font size or style displayed on the schematic. For the voltage or current markers the font choices and the color of the labels is set using the main toolbar and selecting PSPICE/BIAS POINTS/PREFERENCES to obtain the menu shown in Figure 2.

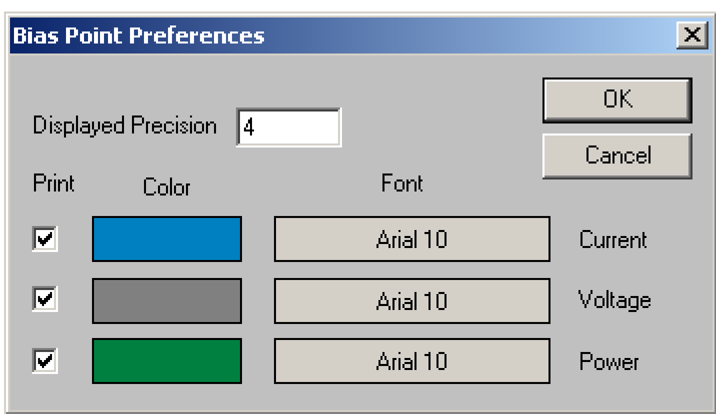


FIGURE 2

Menu for setting font characteristics and labels for currents and voltages on the schematic

For the labels on circuit components, port labels etc., the font size is set on the menu OPTIONS/DESIGN TEMPLATE as shown in Figure 3.

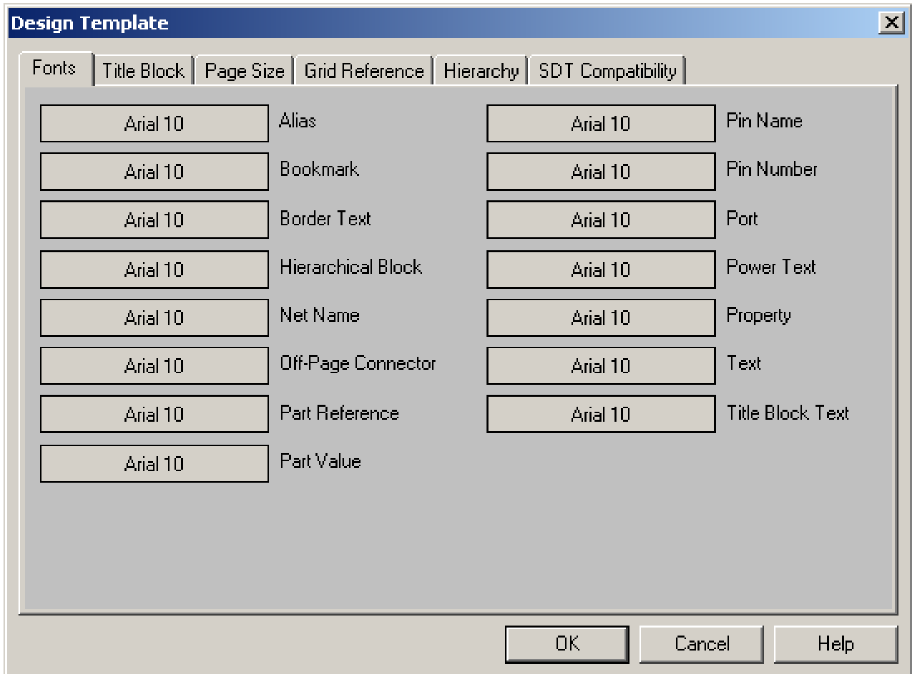


FIGURE 3

Menu found in OPTIONS/DESIGN TEMPLATE from the main schematic toolbar

Running a Q-point Simulation

To obtain the Q-point values shown in Figure 1 we first run a simple Q-point analysis by selecting PSPICE/ NEW SIMULATION PROFILE and choosing the ANALYSIS TYPE as BIAS POINT. I chose "Q-point" for the name of this SIMULATION PROFILE. The simulation settings are shown in Figure 4 below.

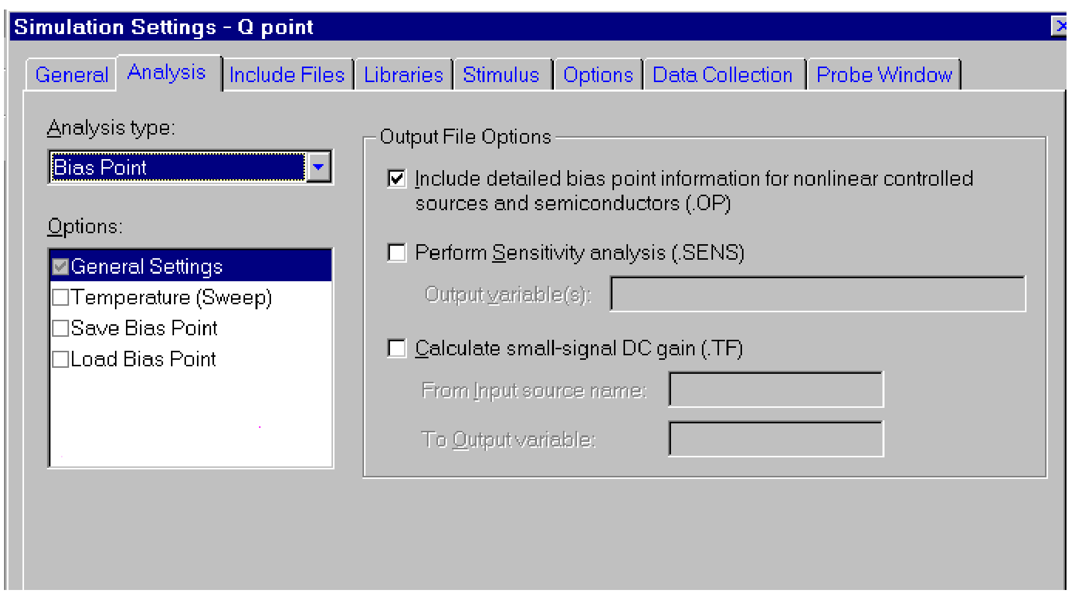


FIGURE 4:

Screen dump of simulation settings for the Q-point simulation.

Once the bias point analysis has been run, we can find the Q-point information for the transistors in the output file, accessed using VIEW/OUTPUT FILE as shown in Figure 5. An excerpt from the output file is shown in Figure 6.

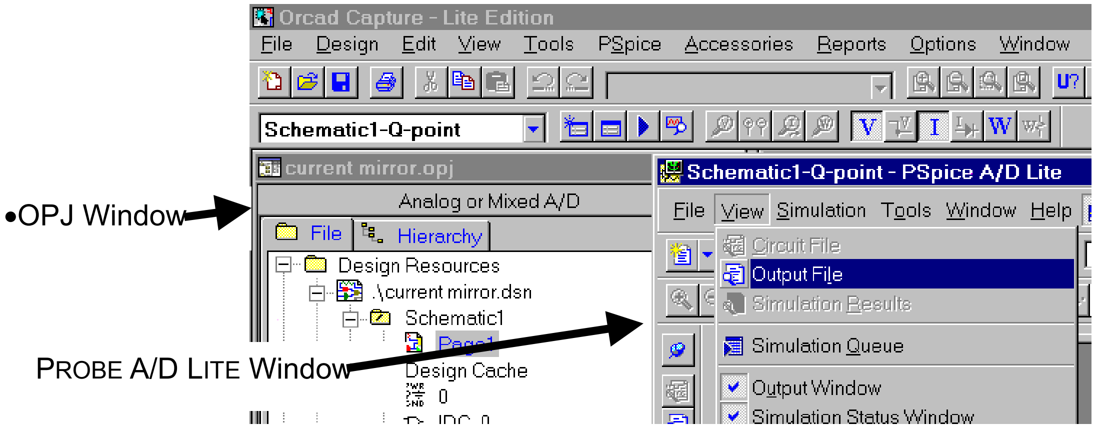


FIGURE 5:
Screen dump showing how OUTPUT FILE can be accessed from PROBE.

**** BIPOLAR JUNCTION TRANSISTORS

NAME	Q_Q1	Q_Q2
MODEL	Q2N2222	Q2N2222
IB	5.58E-05	5.68E-05
IC	9.89E-03	9.89E-03
VBE	0.706	0.706
VBC	-1.350	0.000
VCE	2.060	0.706
BETADC	177	174
GM	0.370	0.370
RPI	4.98E+02	4.89E+02
RX	10	10
RO	7.62E+03	7.49E+03
CBE	1.90E-10	1.91E-10
CBC	5.15E-12	7.34E-12
CJS	0.00E+00	0.00E+00
BETAAC	184	181
CBX/CBX2	0.00E+00	0.00E+00
FT/FT2	3.01E+08	2.97E+08

FIGURE 6:
Q-point data for bipolar transistors when applied DC voltage is DC = 12 V. This data has been formatted by copying it from the PSPICE OUTPUT FILE into EXCEL and using the EXCEL menu DATA/TEXT TO COLUMNS

Pasting Schematics into WORD

In the ECE Computer Lab the PC's have a template called PSPICE.dot. If you use WORD on these computers, the macros described below are installed in this template. You can use these shortcuts to cut and paste by selecting FILE/NEW and choosing the template PSPICE.dot. If you want to incorporate these same macros in your computer, you can either copy the template, or add these macros to your own template following the directions in the document *Using Word.*

First, let's paste a schematic into WORD '97. In PSPICE outline your schematic and use EDIT/COPY. This copies your SCHEMATIC. Then insert your cursor in the WORD document and use the keys Ctrl+P. We obtain Figure 7 below.

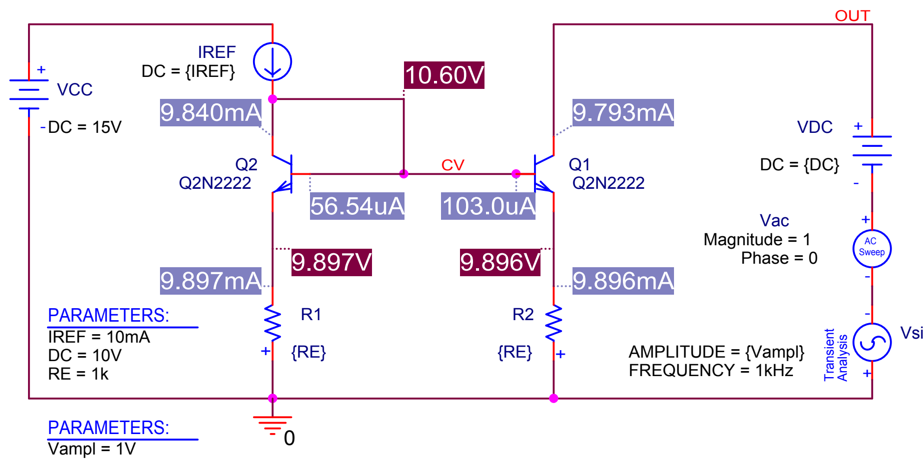


FIGURE 7:

Current-mirror circuit.

If you forget to copy something before you use Ctrl+P, so you are trying to paste something that isn't there, a menu will appear warning you. Simply select the END option, and go back and copy the figure. Then Ctrl+P will work.

DC *I-V* Characteristics of Current Mirror Using PROBE

As an initial PROBE plot we make a basic DC sweep of the mirror current *vs.* the output voltage across the current mirror, the parameter DC. We follow Herniter §4A. The simulation settings are shown in Figure 8 below.

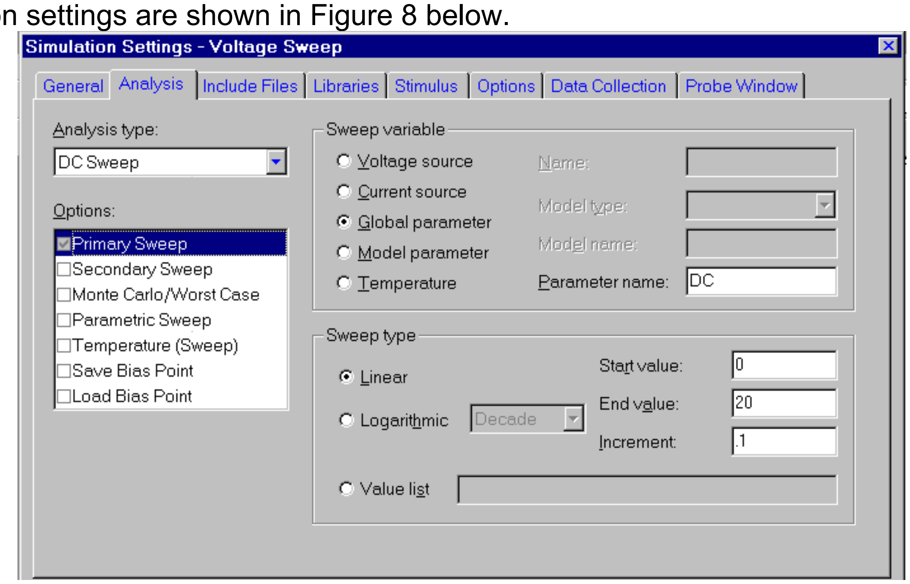


FIGURE 8:

Screen dump of simulation settings for DC sweep of global parameter DC

PASTING A PROBE PLOT INTO WORD

To insert the corresponding output plot from PROBE, in PROBE we use WINDOW/COPY TO CLIPBOARD. In PROBE, we make the following menu choices after selecting

WINDOW/COPY TO CLIPBOARD. Check the BACKGROUND box to make backgrounds transparent.

Assuming the default colors have been replaced according to [Appendix 2](#), choose USE SCREEN COLORS. Moving to WORD, we can use the Ctrl+P macro. The plot is in Figure 9 below.

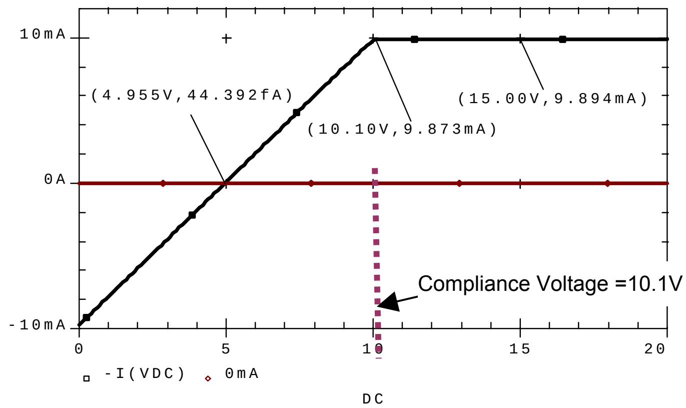


FIGURE 9:

PROBE plot of DC *I-V* curve for the current mirror in Figure 1 using PASTE SPECIAL/PICTURE (macro Ctrl+P). This figure is <u>not</u> WYSIWYG *(what you see is what you get)*. Compliance voltage label is added in WORD.

(Return to list of PSPICE simulation figures.)

There is another viable choice for pasting figures from PROBE, unlike the case with the SCHEMATIC where only one choice exists. Because PROBE figures are copied to the CLIPBOARD, WORD gives us another option, which is to paste as a device-independent bitmap. We use Ctrl+D. Then we get Figure 10 instead of Figure 9.

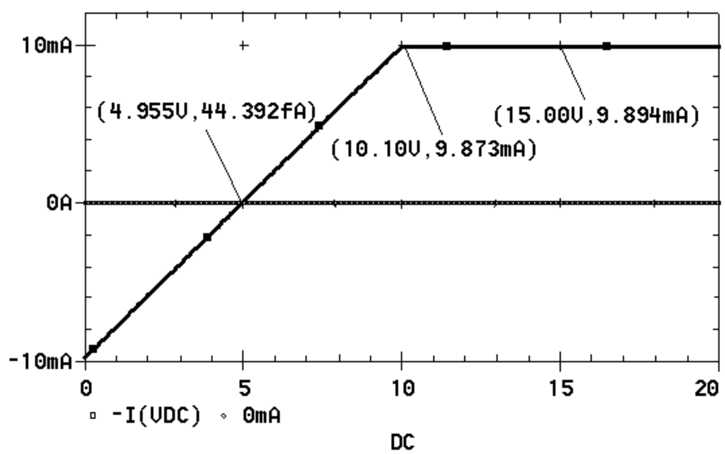


FIGURE 10:

PROBE plot for current mirror in Figure 1 using PASTE SPECIAL/DEVICE INDEPENDENT BITMAP (macro Ctrl+D). This figure is WYSIWYG.

(Return to list of PSPICE simulation figures.)

Figure 10 uses a different typeface, and does not allow colored curves. It also automatically scales the font size so it looks better in the WORD viewer. When printed out, the two figures look quite different as well. The type size in the labels of Figure 9 is much smaller than it appears in the viewer, and a frame appears around the picture. In

Figure 10, the printed picture has no frame and the labels are larger and in bold face. It's a matter of taste: how much do you like color and WYSIWYG *(what you see is what you get)*?
If you want to copy and paste screen dumps using Ctrl+shift+Print Scrn, the bitmap approach (Ctrl+D) is the only choice.

PASTING PSpice Probe DATA INTO Excel
Once a PROBE plot has been made, we can click on the caption label for the generated curve and it becomes highlighted. Next we select EDIT/COPY. This action copies the data for this curve. Next we open EXCEL and click the right mouse button, select PASTE. The PROBE data is copied onto the spreadsheet. The data on the spreadsheet is already highlighted. While it is highlighted, click the CHART WIZARD button  and select a scatter plot as shown in Figure 11.

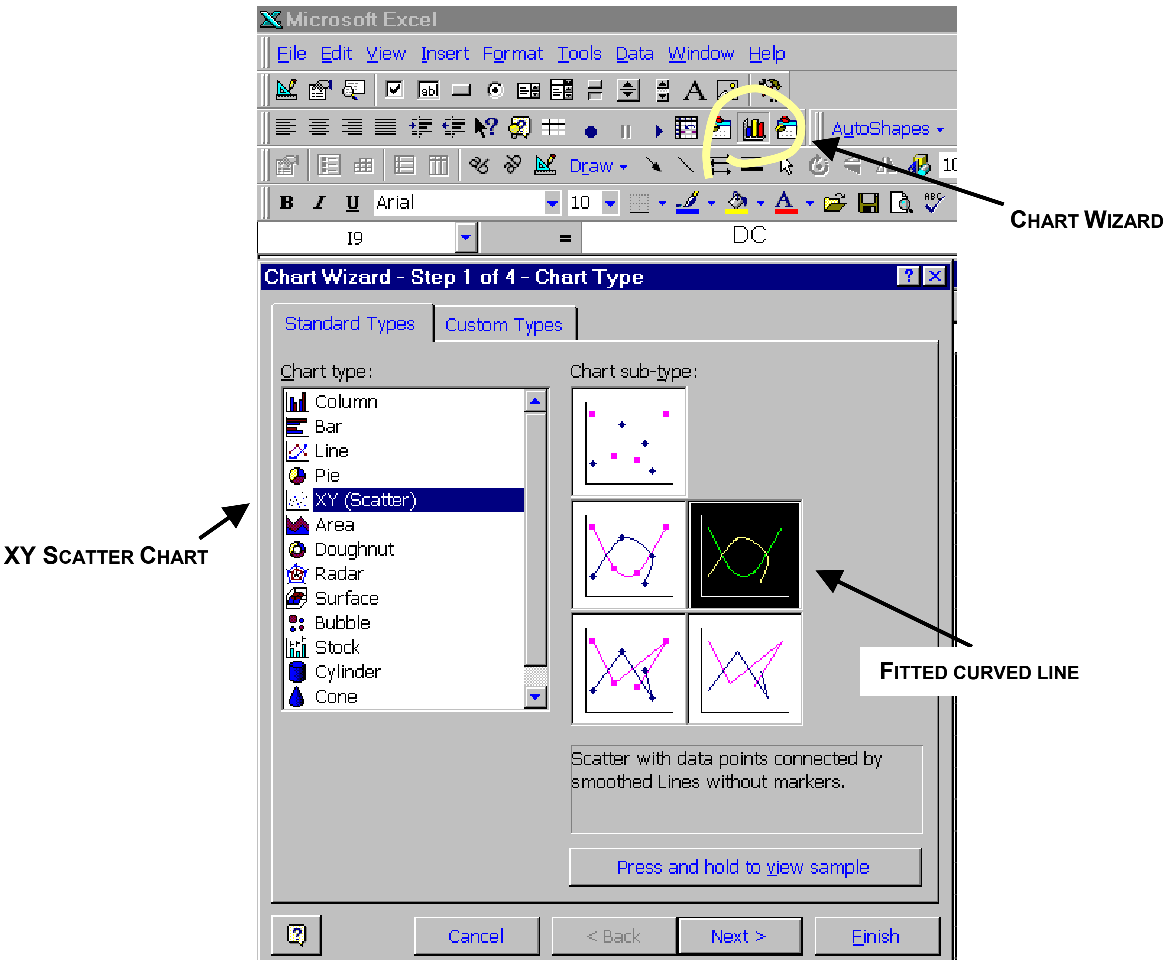


FIGURE 11:
Screen dump from EXCEL showing selection of chart type using the chart wizard.

Click FINISH. Using the right mouse button and various menu selections, you will end up with a chart formatted as you like it. See *Using EXCEL*. An example is shown in Figure 12.

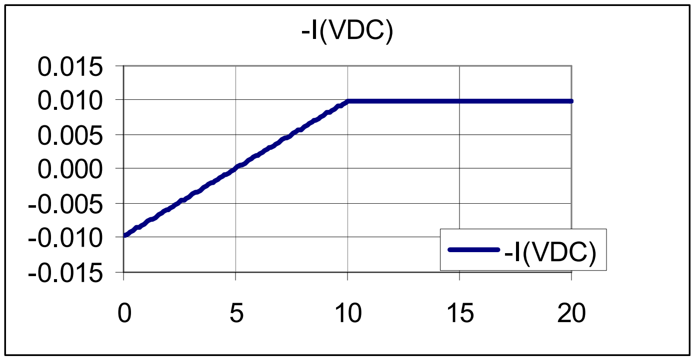


FIGURE 12:
Example EXCEL chart of the data in Figure 10.

PASTING Excel CHARTS AND TABLES INTO WORD

To paste an EXCEL chart into WORD '97 we need yet another paste macro because EXCEL charts are a different type of file (an OLE object). In the PSPICE.dot template this paste macro has the shortcut Ctrl+E. The procedure is to COPY the chart or table in EXCEL, move the mouse to the WORD document, and then press Ctrl+E.

In WORD 2000, charts in EXCEL are copied to the clipboard. The macro PASTECLIP is assigned to Ctrl+Shift+E in PSPICE.dot. If you click on the clipboard command bar to empty the clipboard after the paste, you will avoid some confusion with multiple clipboard entries.

DC determination of Norton Resistance: Using PSPICE D() for Derivative[3]

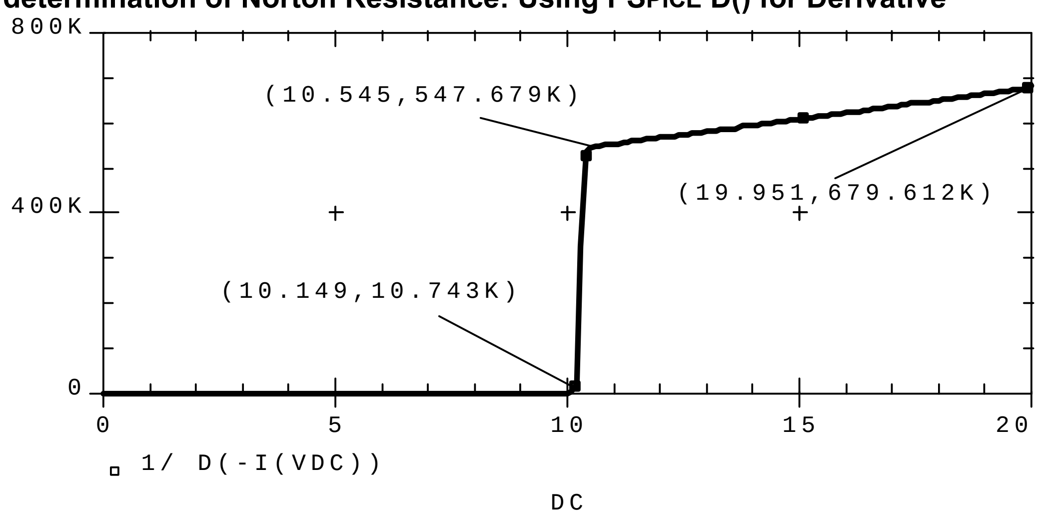


FIGURE 13:
Reciprocal of derivative of the DC *I-V* plot of Figure 9 to find the output resistance of the mirror.

(Return to list of PSPICE simulation figures.)

---

[3] A listing of available PROBE functions can be seen in the Orcad On-line Reference Manual, pp. xvii-xix, "C:\Program Files\OrcadLite\Document\PSpcRef.pdf" if you loaded PSPICE following Herniter's directions

Figure 13 shows how the PSPICE function D(), which takes the derivative of its argument with respect to the x-axis variable, can be used to find the mirror resistance as the inverse slope of the *I-V* curve. D() can be found on the TRACE ADD menu in the right-hand column under ANALOG OPERATORS AND FUNCTIONS. If this window is blank, try scrolling the FUNCTIONS OR MACROS window at the top of the column. At a DC voltage of 10.15V the output transistor begins to leave saturation and by DC = 10.55V it has become active. The *I-V* curve then has a low slope, implying a high output resistance.

Norton Resistance of Current Mirror: AC Frequency Sweep and Macros

Next, we use PROBE to generate the output resistance of the current mirror using an AC analysis. This approach has two possible advantages over a DC analysis: it avoids the numerical differentiation used in Figure 13 that can sometimes lead to "noisy" behavior if the voltage increment in DC is chosen too small, and it provides the frequency dependence of the Norton impedance introduced by the transistor capacitances (these are studied later in the course).

This analysis is a small-signal analysis, that is, it is based on a linearized circuit.

For this analysis, we make a frequency sweep of the AC voltage across the mirror using part VAC as described by Herniter § 5.I.2, p. 292. The simulation settings are in Figure 14.

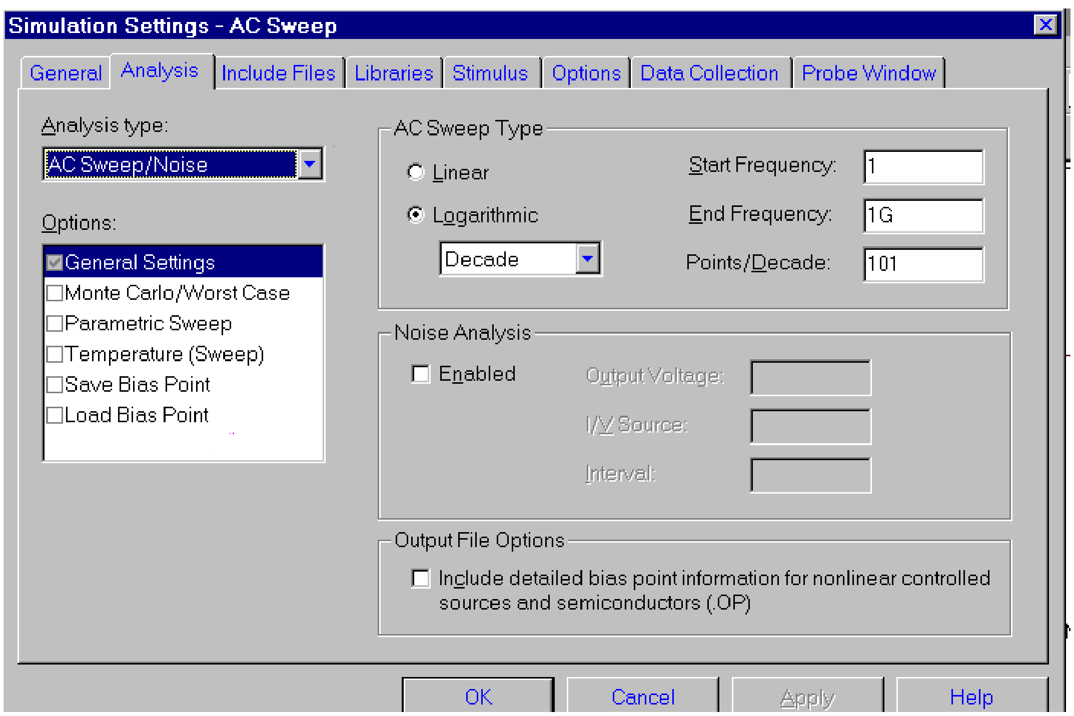


FIGURE 14:
Screen dump showing the AC-sweep simulation settings. We sweep frequency on a log scale by DECADE from 1 Hz to 1 GHz.

In this case, we model the current mirror as a Norton source with a Norton impedance across it. At low frequencies, the Norton impedance is well approximated by a parallel resistor-capacitor combination. In this case, the current flowing into the mirror from the driving AC source is $I = (j\omega C + 1/R)V_{ac}$. As $V_{ac} = 1$V, we can find the resistive portion of the impedance as *R = 1/R(I)*, where R() is the PSPICE function that finds the real part of its argument. Rather than typing this formula over and over, PSPICE allows a macro to be defined and stored to make your function available from the TRACE/ADD option. To store a macro, select TRACE/MACROS to get the MACROS menu shown in Figure 15.

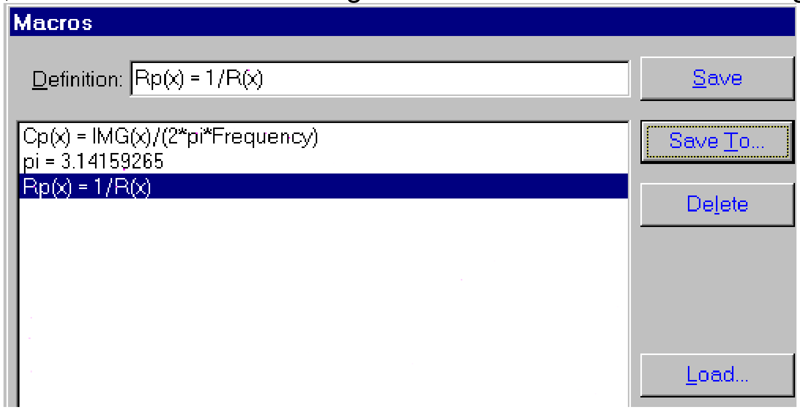


FIGURE 15:

Screen dump of MACROS menu showing the definition of several macros.

To store these macros for general access, use the SAVE TO option and in the resulting menu select GLOBAL FILE.

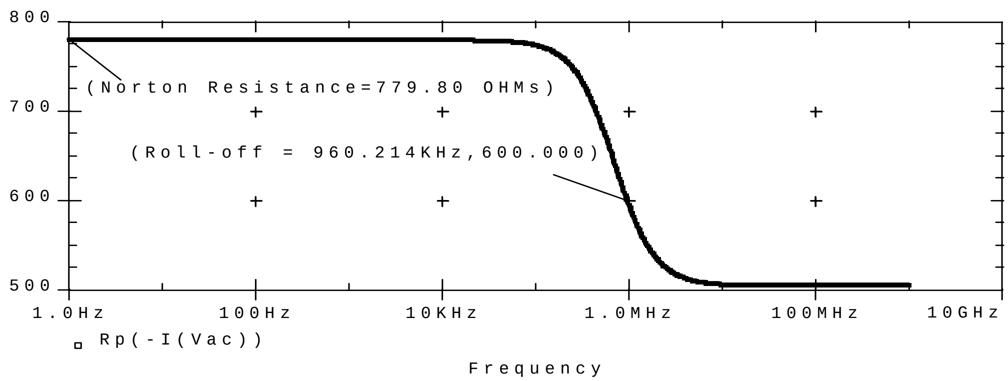


FIGURE 16:

Norton resistance of current mirror *vs.* frequency.

(Return to list of PSPICE simulation figures.)

Performing the simulation, we obtain Figure 16. The cursor is activated using the toggle button, as described by Herniter, § 2.K, p. 119. Labels can be added using the label toggle button, Herniter, § 2K, p. 124. By clicking on the labels, we can edit them to include the WORDS "Norton Resistance" *etc.*

AC Mirror Current in Response to Sinusoidal Output Voltage: Transient Analysis

Next, we do a transient analysis using the sinusoidally varying voltage source VSIN. This analysis is not small signal – that is, it takes into account nonlinear, large-amplitude behavior. The simulation settings are shown in Figure 17 and Figure 18. The meaning of the entries is in Herniter, §6.A.2 p. 297.

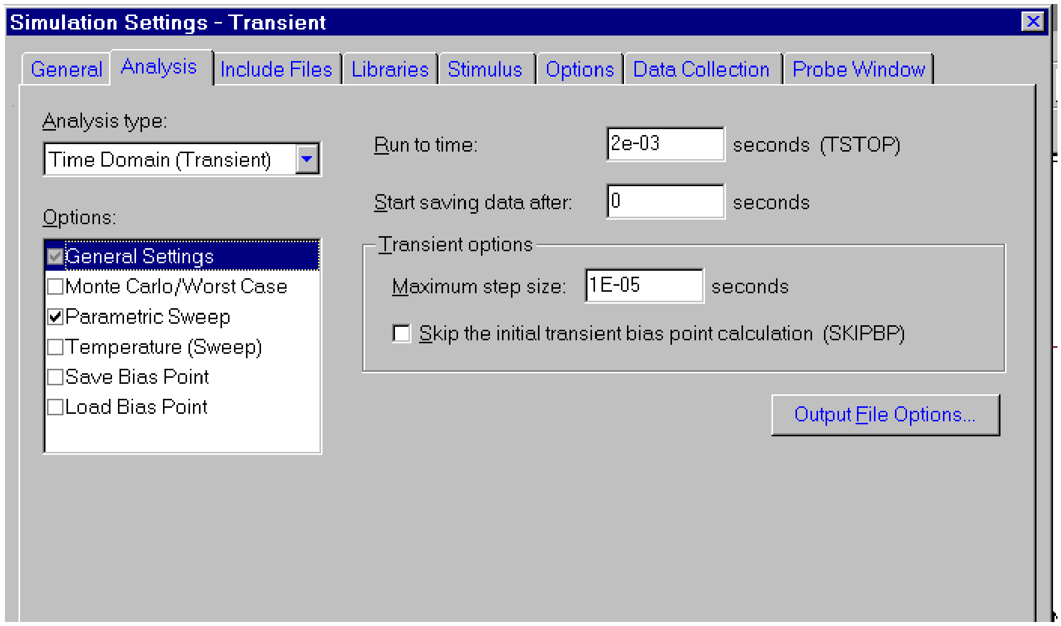


FIGURE 17:

Simulation settings for transient analysis. A parametric sweep of the amplitude of the AC voltage, variable VAMPL is used.

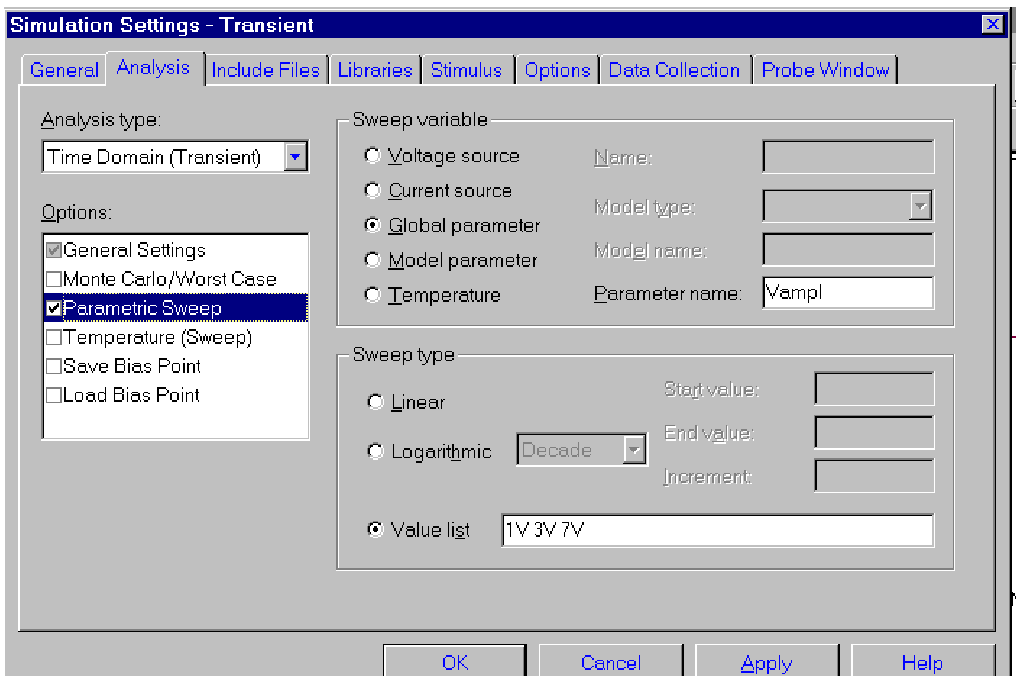


FIGURE 18:

Setup for the global parameter VAMPL that determines the amplitude of the sinusoidal voltage across the mirror. A VALUE LIST of three amplitudes is used to get a family of curves.

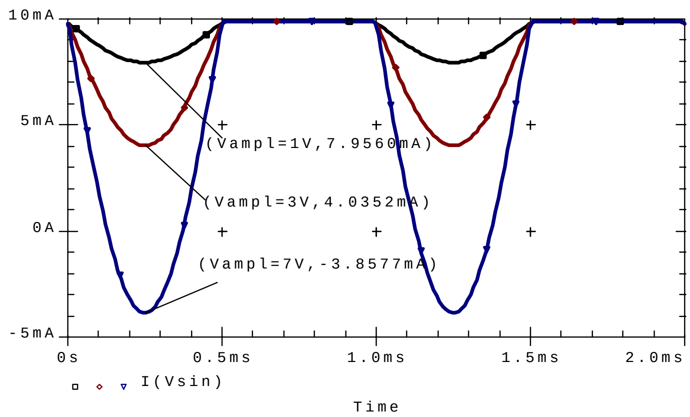


FIGURE 19:

Current mirror transient current output corresponding to sinusoidal output voltage swings of 1V, 3V and 7V amplitudes with DC voltage DC=10V. Clicking on the curve symbol brings up a menu identifying the parameter value for a particular curve.

(Return to list of PSPICE simulation figures.)

Figure 19 shows the three transient output currents of the mirror in response to the three different input voltages. On the upswings of the voltage, the mirror stays above its compliance voltage and the current remains constant. But on the downswings for large enough voltage swings the mirror drops below its compliance voltage and the output transistor Q1 goes into saturation, reducing the mirror current.

In PROBE plots with multiple curves, the curves cannot be distinguished from each other once they are pasted into a WORD document in black and white. To identify the curves the PROBE/TOOLS/OPTIONS menu should have the USE SYMBOLS box checked for ALWAYS. The symbols usually are small, so it's safest to use the TOGGLE CURSOR and MARK LABEL buttons to label the curves. Then click on the labels to add your identifying information.

Fourier Analysis and Harmonic Distortion

Read Herniter §6.G.2, p. 340. In the ECE 304 lab it will be necessary to assess the distortion of an amplifier. PSPICE can help by making a Fourier analysis of the waveform, that is, by determining how many sine waves must be added and with what coefficients to make the waveform. For a linear amplifier, sine wave input leads to single sine wave output. Extra sine waves in the Fourier series indicate harmonic distortion. If PERFORM FOURIER ANALYSIS is selected as shown in Figure 20, the PSPICE output file lists a tabulation of the amplitude and phase of each harmonic, and the total harmonic distortion = power contained in the harmonics (*i.e.* the square root of the sum of squares of amplitudes).

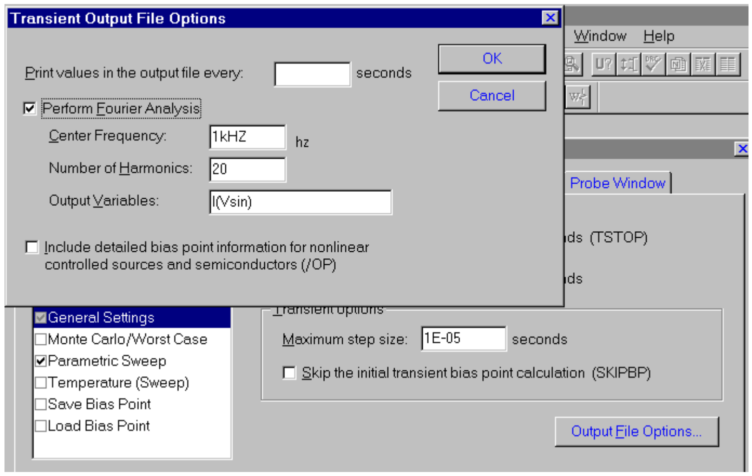


FIGURE 20:

Screen dump of menu activating Fourier analysis. Pressing the OUTPUT FILE OPTIONS bar with GENERAL SETTINGS selected on the TRANSIENT simulation profile accesses this menu.

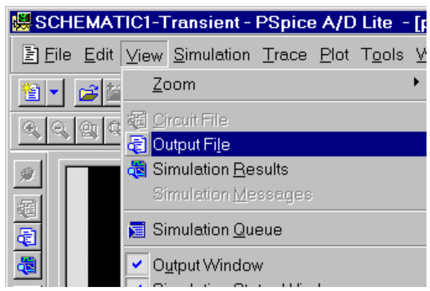


FIGURE 21:

Screen dump showing how to access the OUTPUT FILE from PROBE.

FOURIER COMPONENTS OF TRANSIENT RESPONSE I(V_Vsin)

DC COMPONENT = 9.265492E-03

HARMONIC NO	FREQUENCY (HZ)	FOURIER COMPONENT	NORMALIZED COMPONENT	PHASE (DEG)	NORMALIZED PHASE (DEG)
1	1.00E+03	9.67E-04	1.00E+00	1.80E+02	0.00E+00
2	2.00E+03	3.97E-04	4.11E-01	9.00E+01	-2.70E+02
3	3.00E+03	3.60E-06	3.73E-03	1.75E+02	-3.65E+02
4	4.00E+03	7.18E-05	7.42E-02	9.00E+01	-6.30E+02
5	5.00E+03	4.64E-06	4.80E-03	1.77E+02	-7.23E+02
6	6.00E+03	2.76E-05	2.85E-02	9.00E+01	-9.90E+02
7	7.00E+03	4.07E-06	4.21E-03	1.77E+02	-1.08E+03
8	8.00E+03	1.35E-05	1.40E-02	9.00E+01	-1.35E+03
9	9.00E+03	3.35E-06	3.47E-03	1.76E+02	-1.44E+03
10	1.00E+04	7.42E-06	7.68E-03	9.01E+01	-1.71E+03
11	1.10E+04	2.70E-06	2.79E-03	1.75E+02	-1.81E+03
12	1.20E+04	4.35E-06	4.50E-03	9.01E+01	-2.07E+03
13	1.30E+04	2.14E-06	2.22E-03	1.74E+02	-2.17E+03
14	1.40E+04	2.64E-06	2.73E-03	9.01E+01	-2.43E+03
15	1.50E+04	1.68E-06	1.74E-03	1.72E+02	-2.53E+03
16	1.60E+04	1.64E-06	1.70E-03	9.00E+01	-2.79E+03
17	1.70E+04	1.30E-06	1.35E-03	1.70E+02	-2.89E+03
18	1.80E+04	1.04E-06	1.07E-03	8.97E+01	-3.15E+03
19	1.90E+04	9.97E-07	1.03E-03	1.67E+02	-3.25E+03
20	2.00E+04	6.69E-07	6.93E-04	8.92E+01	-3.51E+03

TOTAL HARMONIC DISTORTION = 4.186126E+01 PERCENT

FIGURE 22:

Harmonic distortion from the PSPICE output file. The output was pasted into EXCEL to format it this way using the EXCEL option DATA/TEXT TO COLUMNS. Total harmonic distortion is ≈ 42%.

Figure 22 shows the harmonic distortion analysis of the current waveform in Figure 19 for VAMPL = 1V. Because this waveform is approximately a rectified sine wave (it is zero for half a cycle) we expect huge distortion.

EFFECT OF MAXIMUM STEP SIZE

The output of the transient analysis simply "connects the dots", joining the values determined at each step. Usually PSPICE selects a reasonable step size by itself. However, sometimes it chooses too coarse a step. A large step size results in an output that has corners, that is, sharp breaks in slope at each step. The Fourier analysis of such a waveform contains many harmonics because it does not resemble a sine wave. This distortion is a numerical artifact, and is not the true output of the amplifier. Too large a step size leads to an incorrect harmonic distortion analysis. If PSPICE selects too large a step, you have to force a smaller step size by setting the maximum step size in the menu shown in Figure 17.

Norton Resistance *vs.* Emitter-leg Resistance: Performance Analysis

The next type of plot we will use is PERFORMANCE ANALYSIS. In our case, we will plot the Norton resistance as a function of the emitter-leg resistor value R_E. Other applications of performance analysis are described by Herniter § 5.G. p. 280.

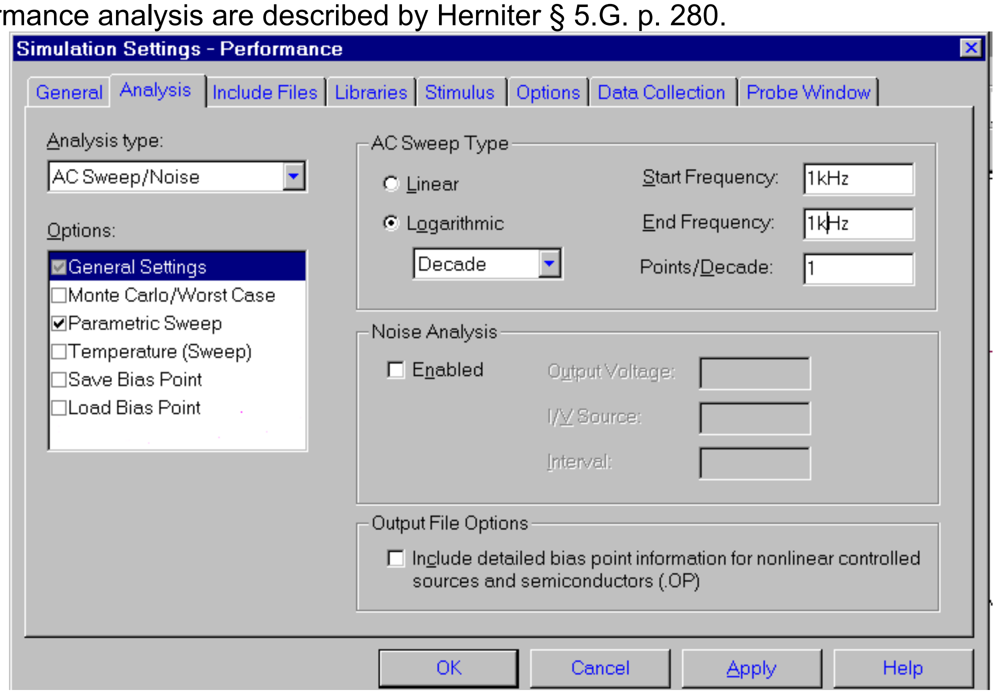


FIGURE 23:

Screen dump for performance analysis plot. Notice that the start and stop frequencies are the same and only one point/decade is requested.

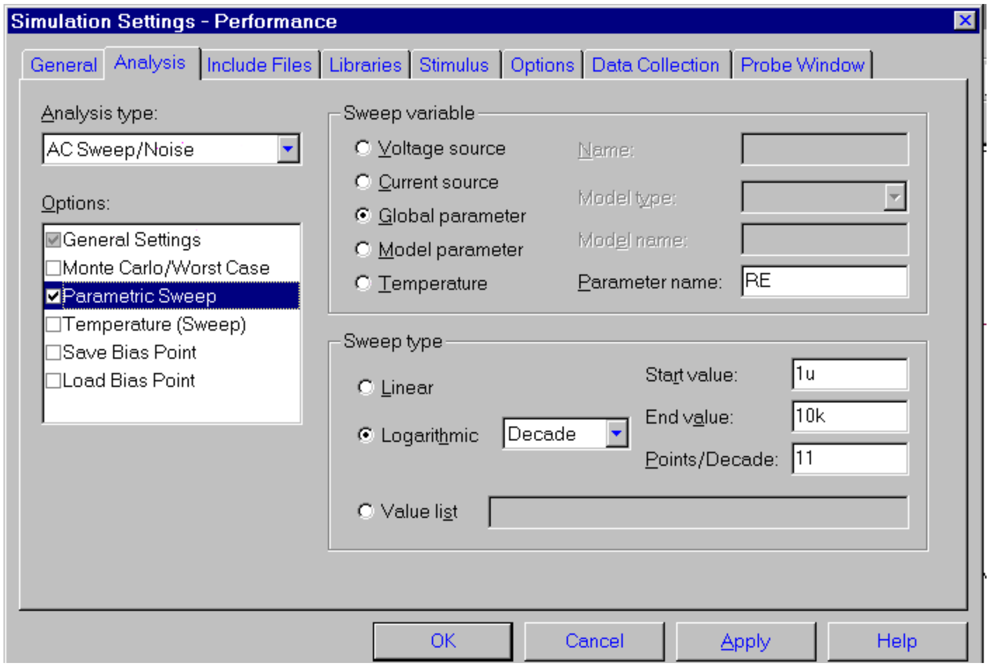


FIGURE 24:

Screen dump for the parametric sweep in the performance analysis

To make this performance analysis, we want to fix the frequency and sweep the parameter value. The simulation setup is shown in Figure 23 and Figure 24. The resulting PROBE plot is shown in Figure 25.

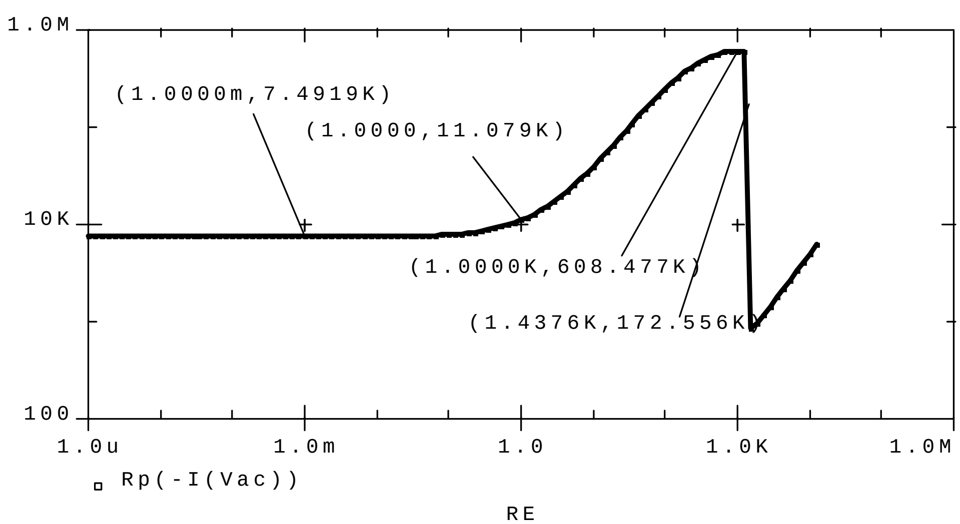


FIGURE 25:

Performance analysis output for Norton resistance at 1 kHz *vs*. the emitter-leg resistor value R_E. The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage.

(Return to list of PSPICE simulation figures.)

Figure 25 shows the Norton resistance is nearly independent of R_E at a value of R_N = 7.5 kΩ for R_E < 1Ω, but as R_E increases, the Norton resistance rapidly climbs to 608.5 kΩ. Then it suddenly drops. What is the cause of this drop in R_N for R_E in the range 1 kΩ ≤ R_E ≤ 1.5 kΩ?

To see whether it is related to the compliance voltage, we look at the compliance voltage as a function of R_E using a DC sweep. The node that sets the compliance voltage is labeled CV in Figure 26.

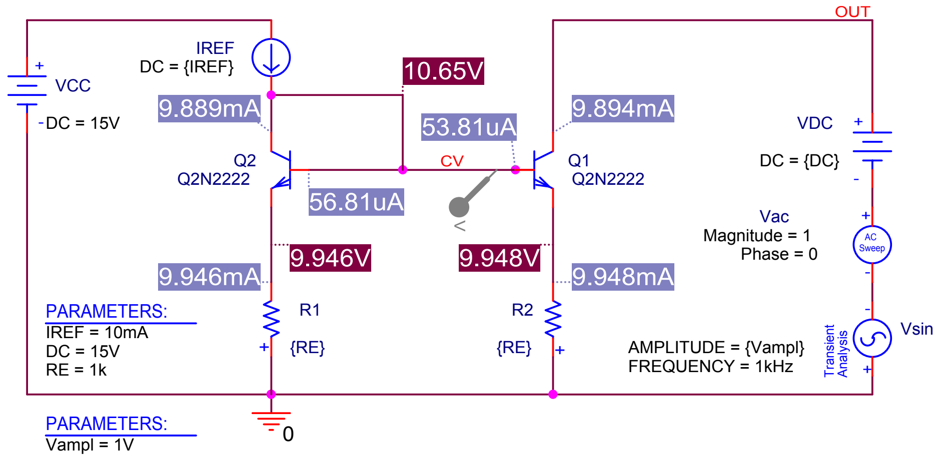


FIGURE 26:

Schematic with node marked CV where compliance voltage is measured. A voltage marker has been placed on that node for the R_E sweep simulation.

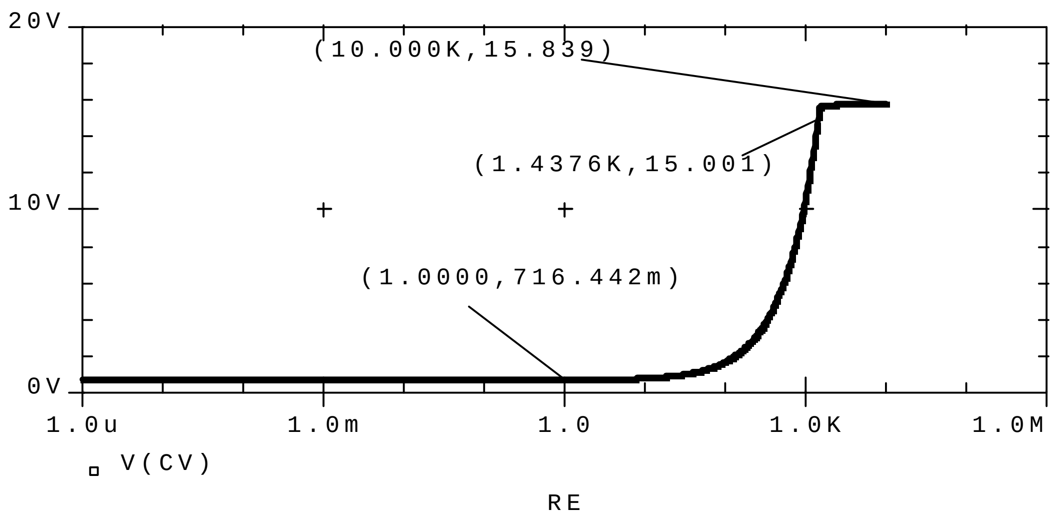


FIGURE 27:

DC sweep of compliance voltage *vs.*emitter-leg resistor value R_E. The compliance voltage reaches the supply voltage when R_E = 1.44 kΩ.

(Back to list of figures.)

From Figure 27 we can see that indeed the compliance voltage exceeds the supply voltage when R_E ≥ 1.44 kΩ, which means that the output transistor Q1 will go into saturation for R_E ≥ 1.44 kΩ, reducing the mirror current. The mirror's Norton resistance also drops because the mirror has left the low-slope region of the *I-V* curve, meaning the output resistance has dropped.

Using Goal Functions

Performance analysis also enables the use of GOAL FUNCTIONS. To illustrate the provided goal functions, we explore the dependence of the gain of a common-emitter amplifier upon C_{OMP}, an external capacitor value. We find the frequency where the phase of the gain is –45º as a function of C_{OMP}. The circuit is shown in Figure 28.

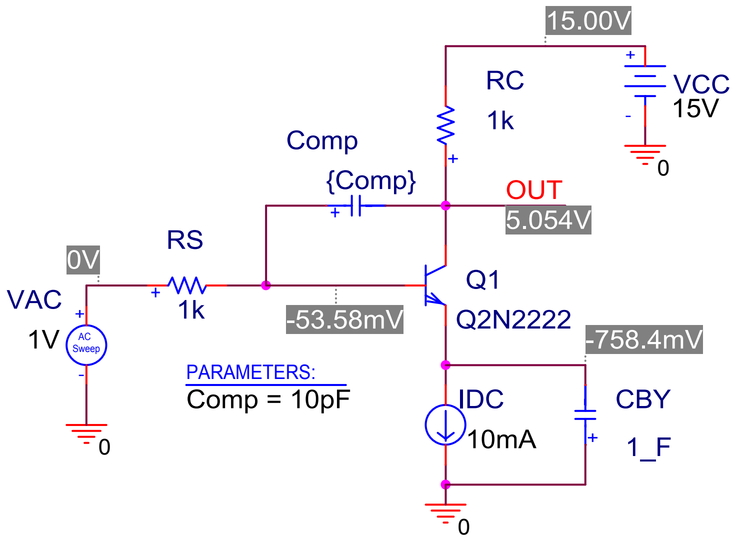


FIGURE 28:

Schematic of circuit for demonstration of GOAL FUNCTIONS; an external capacitor C_{OMP} modifies the frequency response of the amplifier

The phase of the small-signal gain for several values of the capacitor C_{OMP} is shown in Figure 29. The downward shift of the –45º frequency with the increase of C_{OMP} is evident. We suppose that we would like to track this frequency as a function of C_{OMP} without using the cursor to pick off values as has been done in Figure 29. The use of GOAL FUNCTIONS makes this tracking possible.

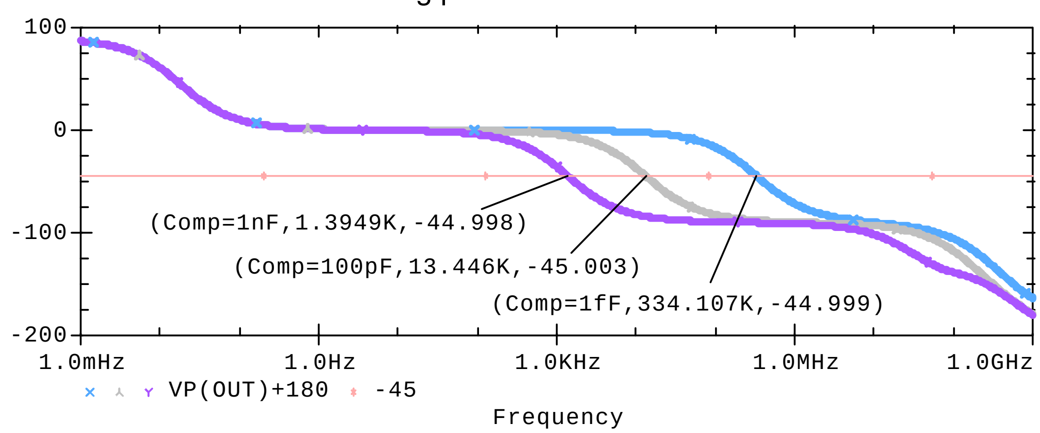


FIGURE 29:

Phase of gain of circuit in Figure 28 (with 180º added to counter the intrinsic sign flip of the common emitter stage)

To implement the goal functions, we first set up a performance analysis. First the main frequency sweep is set up as shown in Figure 30.

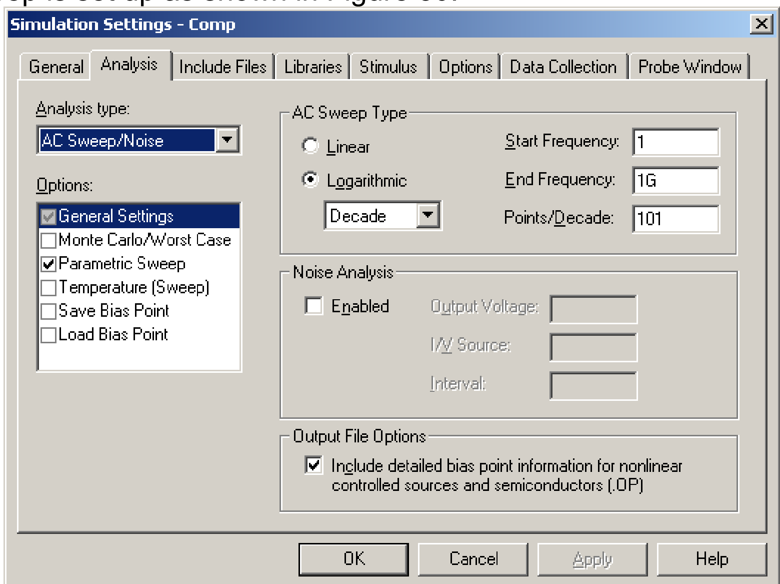


FIGURE 30:

Setting up the frequency sweep for the performance analysis

Next the secondary sweep of the capacitance value C_{OMP} is set up as shown in Figure 31

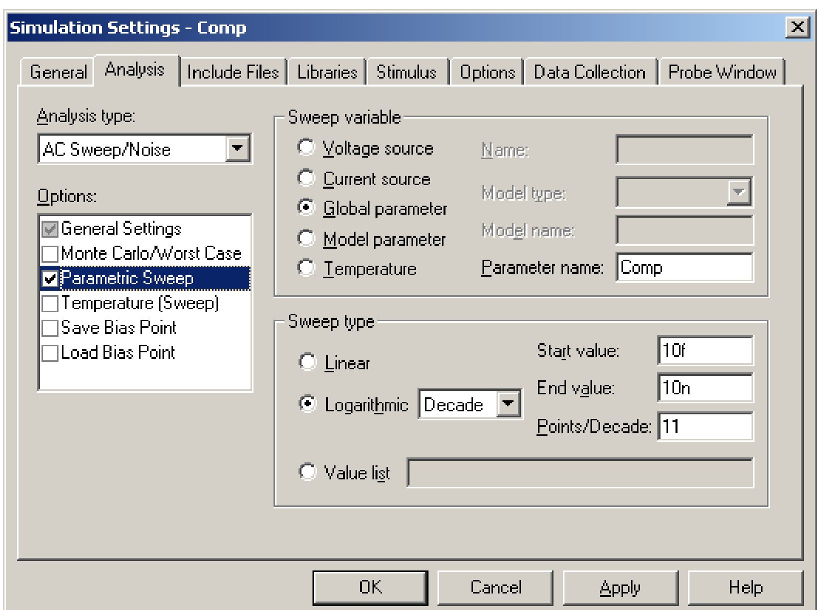


FIGURE 31:

Setting up the parametric sweep of C_{OMP}

We then run the simulation without putting any markers on the schematic. We obtain a blank PROBE screen and select the PLOT/ACCESS SETTINGS menu, shown in Figure 32. The box PERFORMANCE ANALYSIS IS checked. The x-axis now becomes the swept variable (C_{OMP} in this example); see Figure 33.

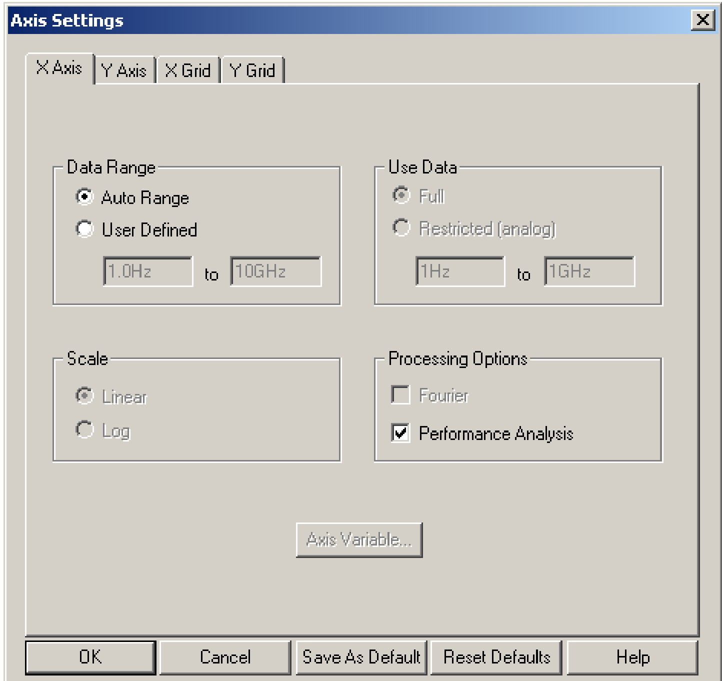


FIGURE 32:
Checking the PERFORMANCE ANALYSIS box on the ACCESS SETTINGS menu

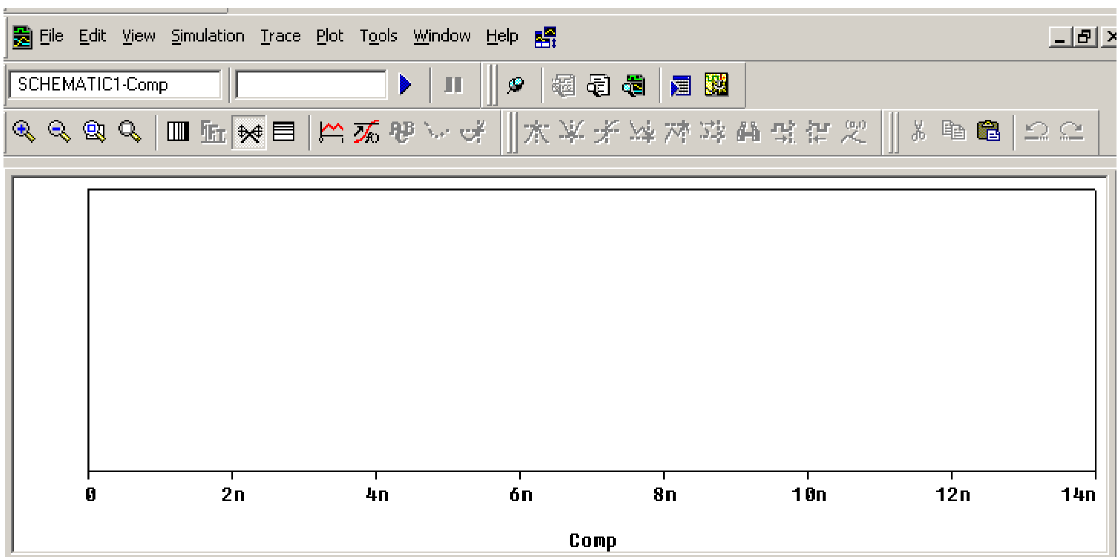


FIGURE 33:
The blank PROBE window with the swept variable as x-axis

Next we want to add a trace. We are going to use a GOAL FUNCTION.

Of course, if you know the goal function you want already, and what its arguments mean, you don't have to consult the code listing for the function. You can use the TRACE/ ADD TRACE menu and make your selection of goal function from the right panel of this menu (see Figure 37). But if you want some directions, or if you want to customize the goal function to your application, you will want to access the code and/or the directions.

The descriptions and code for the available goal functions can be viewed using TRACE menu, as shown in Figure 34, and selecting GOAL FUNCTIONS. As shown in Figure 35, for this example we select the function XatNthY from the list of possible goal functions. If we want to see what the arguments of this function mean and what code describes the function, we click on the EDIT tab to obtain the description in Figure 36.

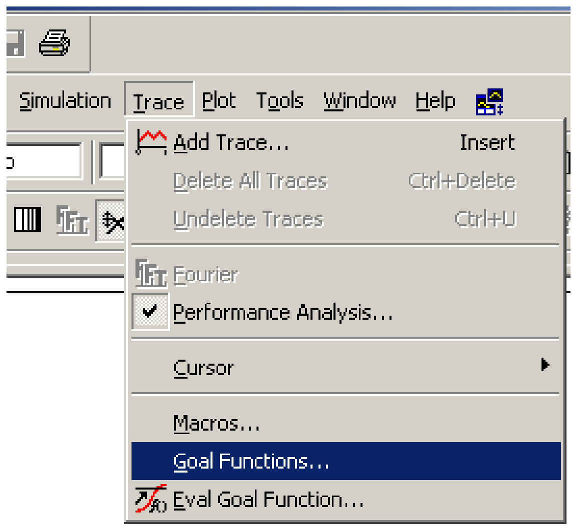


FIGURE 34:
Locating the available GOAL FUNCTIONS

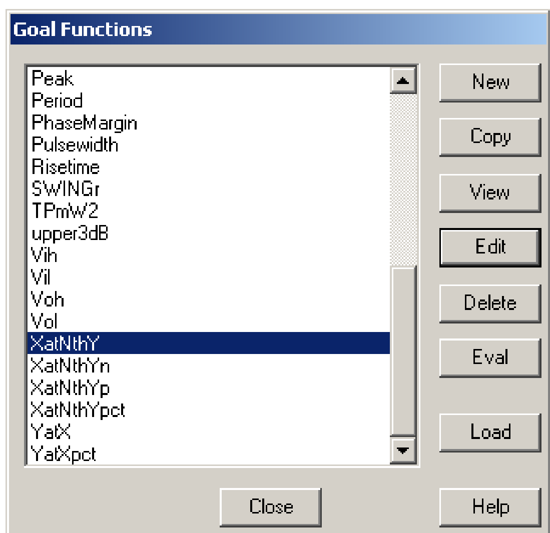


FIGURE 35:
Selecting XatNthY from the listed functions

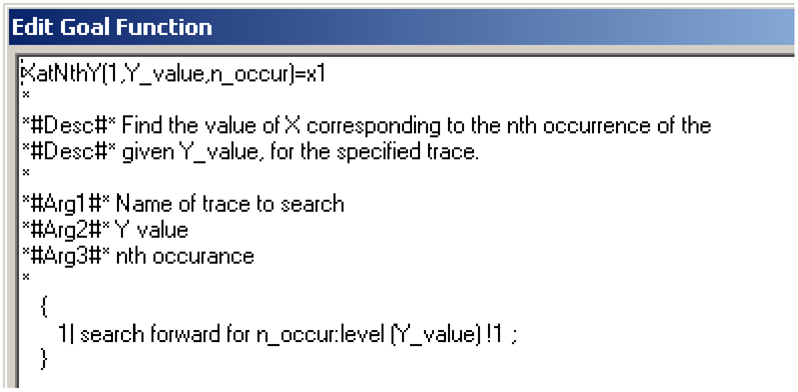


FIGURE 36:

The description of the goal function XatNthY found using the EDIT tab

If we are satisfied that XatNthY is the correct function and we know how to use it, we return to TRACE/ADD TRACE and, set the FUNCTIONS OR MACROS box to GOAL FUNCTIONS, and select the function as shown in Figure 37.

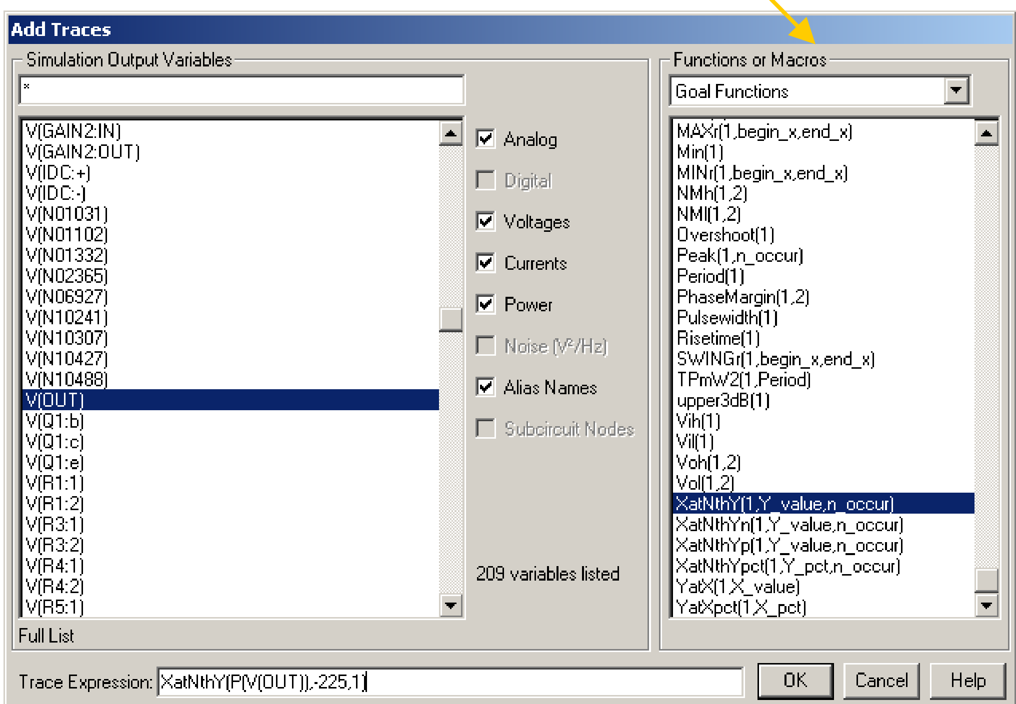


FIGURE 37:

Filling in the arguments of XatNthY; the function P(.) finds the phase of its argument (link back to directions)

Hitting OK, the plot of Figure 38 results. The results agree with Figure 29. The function P(.) for phase is one of the PROBE functions found in the right-hand panel of TRACE/ADD TRACE menu of Figure 37 when the scroll-down box FUNCTIONS OR MACROS is set to ANALOG OPERATORS OR FUNCTIONS. However, this selection is not available in a performance analysis so, to see these functions, you have to open TRACE/ADD TRACE in its customary mode with the PERFORMANCE ANALYSIS box of Figure 32 not checked.

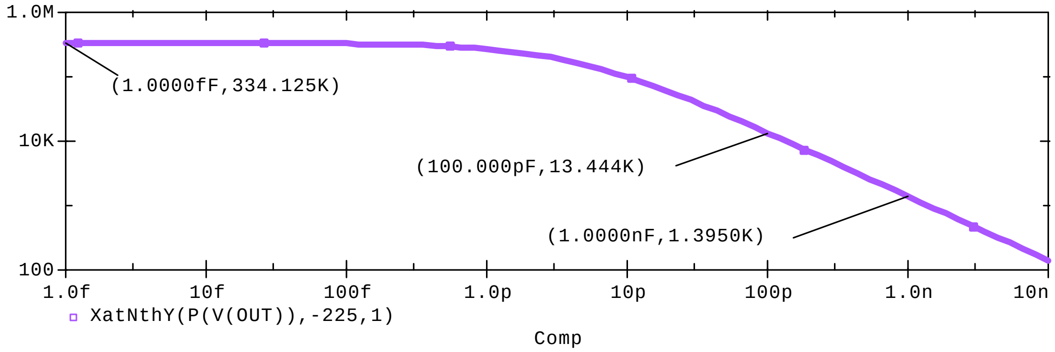


FIGURE 38:
Result of performance analysis using goal function XatNthY to find the values of C_{OMP} where the phase of the gain is –225°

Current Mirror Output *vs.* Reference Current and R_E: Double Sweep

Finally, as indicated at the beginning, we look at the output current of the mirror as a function of the reference current I_{REF}. From the theory of the mirror, we expect that the output current will be almost equal to the reference current. The reason is that I_{REF} sets the V_{BE} of Q2, and because the circuit connection forces $V_{BE}(Q1) = V_{BE}(Q2)$, we expect $I_C(Q1) = I_C(Q2)$ (apart from slight adjustments for base current). To do the simulation, we make a DC sweep of I_{REF} as a global parameter and select R_E from a VALUE LIST to see how R_E affects matters. The use of primary and secondary sweeps is discussed in Herniter §4.D.2, p. 185. In our case, this simulation is named I_{REF} Sweep, and the simulation settings are shown in Figure 39 and Figure 40 below.

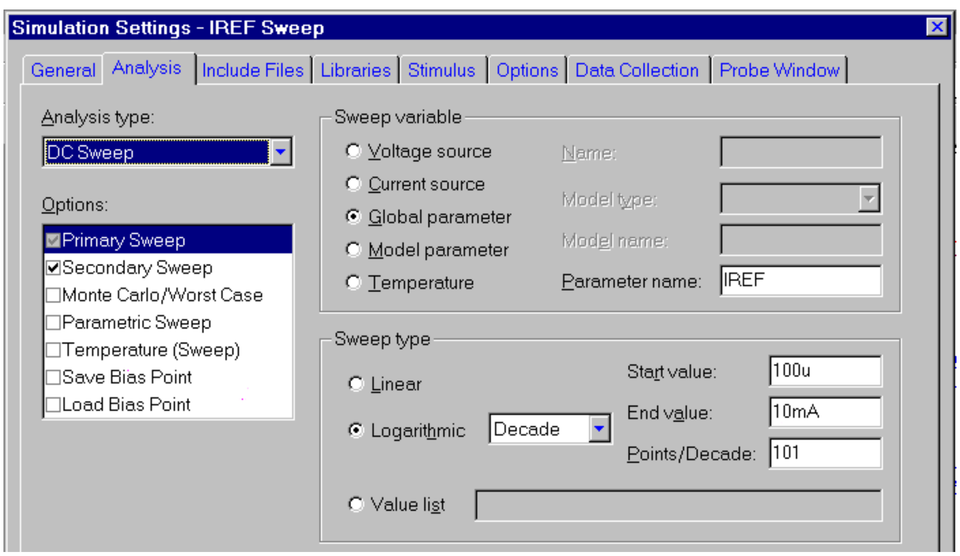


FIGURE 39:
Screen dump for I_{REF} sweep showing the primary sweep settings.

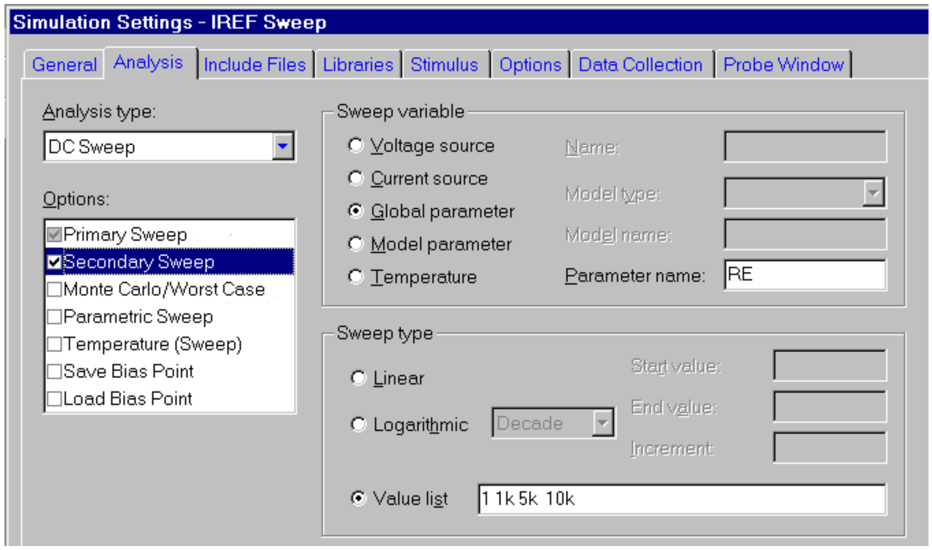


FIGURE 40:

Screen dump for I_{REF} sweep showing settings for the secondary sweep of R_E from a VALUE LIST.

The results of the simulation are shown in Figure 41 below.

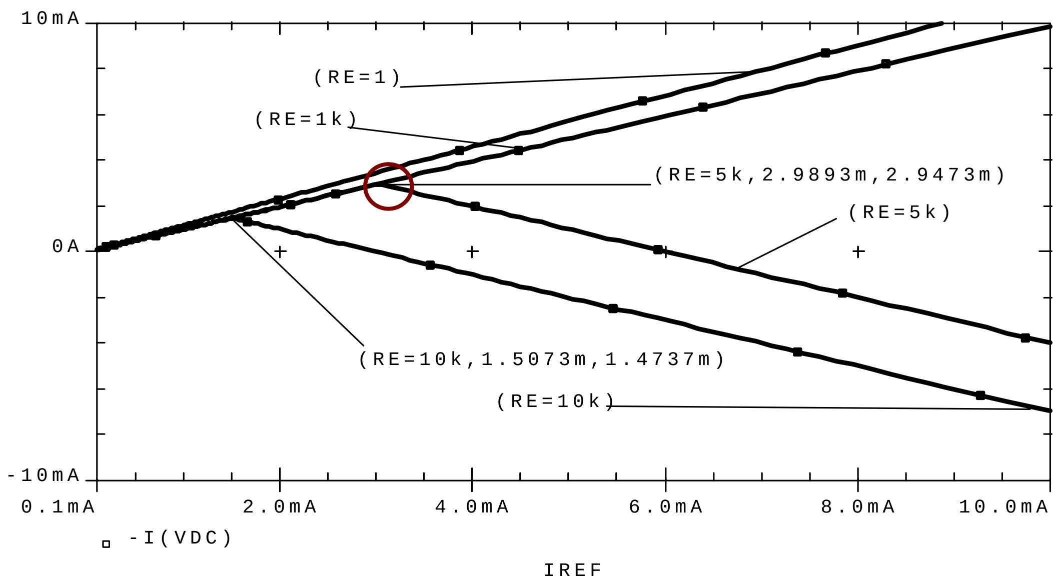


FIGURE 41:

Current from current mirror *vs.* reference current I_{REF} for DC = 15V and several values of emitter-leg resistor value R_E.

(Back to Schematic, or to list of PSPICE simulation figures.)

Figure 41 is a demonstration of undesirable features of the double sweep option. First, it is not possible to separately control the curve identification symbols or curve colors. It also is not possible to identify the parameter value of R_E by clicking on the legend. Finally, it is difficult to move the labels on the screen around because the entire set of curves tends to become selected instead of the label you want to move.

From Figure 41 we can see that the mirror current is very nearly linearly increasing with I_{REF} as expected, at low values of I_{REF}. When I_{REF} becomes large enough that the compliance voltage rises to the voltage across the mirror (15V in this example), the output transistor Q1 saturates and the mirror current actually decreases with further increase in I_{REF}. For example, for the circled point in Figure 41, R_E = 5 kΩ, a break in the curves results at a mirror current of 2.95 mA. The corresponding voltage drop across R_E is 2.95 mA × 5 kΩ = 14.75V. This drop suggests that Q1 has a V_{CE} of 15V – 14.75V = 0.25V or a V_{CB} of $V_{CB} = V_{CE} - V_{BE} \approx 0.25V - 0.7V < 0V$, *i.e.* Q1 is in saturation. You should check for yourself that violation of compliance voltage is responsible for the breaks in the curves by running a sweep of compliance voltage *vs.* I_{REF} with R_E as parameter.

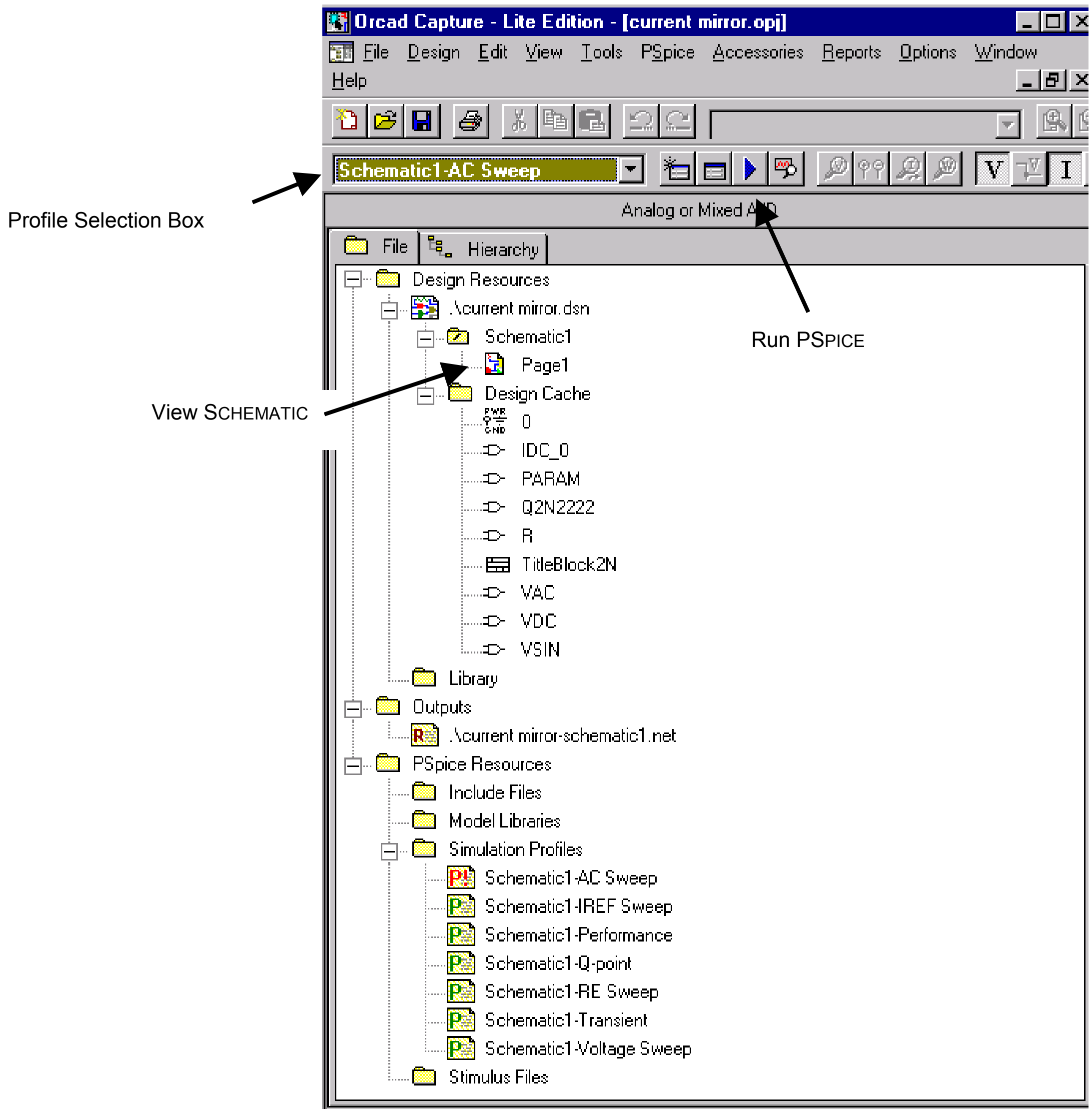


FIGURE 42:

Orcad •OPJ file hierarchy showing the various simulation profiles used in this exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, just for identification in case you have several schematics in this project. You can rename the schematics to be more descriptive if you want.

In Figure 42 is a screen dump of the Orcad file hierarchy (•OPJ) for the project in this exercise. It inventories all the schematics (only one here), parts, and simulation profiles for the project. From this view of the project, clicking on Page1 under the schematic name SCHEMATIC1 accesses the SCHEMATIC. The seven different types of simulation we have used are listed. Selecting a profile using the scroll in the top toolbar and clicking the arrowhead button ( ) runs any SIMULATION PROFILE.

Sending Schematics by E-mail

First, a major caution: if you want to e-mail files, make sure the names of the files <u>contain no spaces or blanks</u>. When files are zipped or unzipped, blanks in the file names often are removed. But <u>inside</u> the files references to these names retain the blanks. Then Orcad cannot identify the files.

To send a file as an attachment by e-mail, you could attach all the project files. You also could zip together all the files and send them, but they still may take a lot of space. An alternative is to use the ARCHIVE utility in PSPICE, as described below. To be sure you have everything, go to the •OPJ view in Orcad, highlight SCHEMATIC1/PAGE1 and SAVE it. Likewise, highlight DESIGN RESOURCES and SAVE it. Then go to FILE/ARCHIVE PROJECT, as shown in Figure 43.

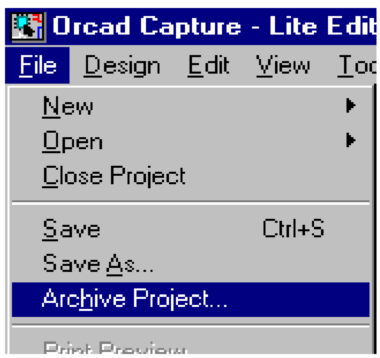


FIGURE 43:

The FILE/ARCHIVE menu selection in CAPTURE.

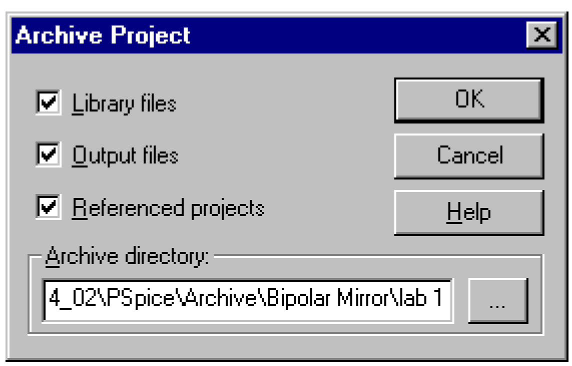


FIGURE 44:

The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The path name for the folder in which the archive is to be placed must be identified in the ARCHIVE DIRECTORY box. If it doesn't already exist, the file is created by the archive process.

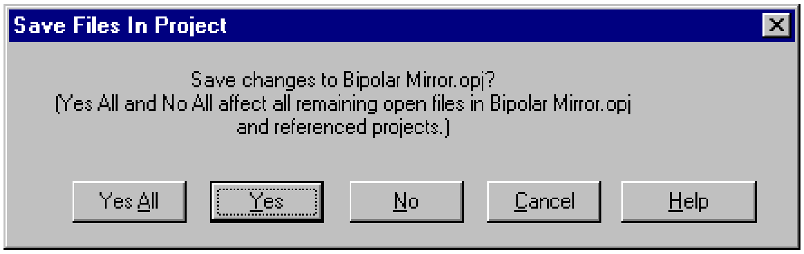


FIGURE 45:
Specify Yes All

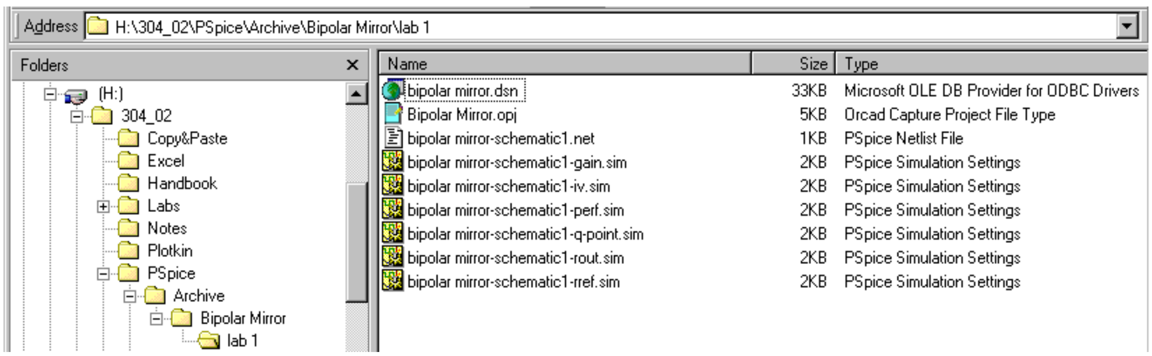


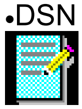
FIGURE 46:
The archived project.

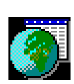
Figure 46 shows the archived project as seen in WINDOWS NT EXPLORER. It contains only the •opj, •dsn, •sim, and the •net files. In this case the archived file is 46 kB, compared to 580 kB for the entire project before archiving. Archiving files not in active use also saves space on your hard drive.

Note on icons: In Figure 46 above, the icons for the various files simply showed up this way on my computer, but they may not look like this on your computer. You can change these icons on Windows NT using the VIEW tab of CONTROL PANEL and selecting OPTIONS/FILE TYPES, selecting the file and using EDIT/CHANGE ICON.

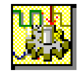
The sure way to identify the correct files is by the file extension, that is , •opj, •dsn, •sim. To see these extensions in the WINDOWS NT FILE EXPLORER, in the TOOLBAR in the EXPLORER select VIEW/FOLDER OPTIONS/VIEW and uncheck the box HIDE FILE EXTENSIONS FOR KNOWN FILE TYPES. Then go back to the EXPLORER TOOLBAR and select VIEW/DETAILS. This is the view seen in Figure 46.

If all you want to send is the schematic, you can get away with sending only the •OPJ file and the •DSN file:


Current Mirror.opj


CURRENT MIRROR.DSN

, where "Current Mirror" is the name of the project. If these two files are all you send, any simulation profiles will be listed by name only, and will have no content. For any simulation profile you want to use, you also



current mirror-SCHEMATIC1-Q-Point.sim

need the •SIM file for the profile: . Here "Q-Point" is the name of the simulation profile in this case, which is expanded by the program to "SCHEMATIC1-Q-Point" in the •OPJ file. Any current or voltage markers you might have placed on the schematic as part of this simulation profile are lost.

ARCHIVING A CIRCUIT WITH BREAKOUT PARTS

In the case of a circuit with custom breakout parts, a warning message is generated during the archive procedure.

FIGURE 47:

Warning message when archiving a circuit with breakout parts

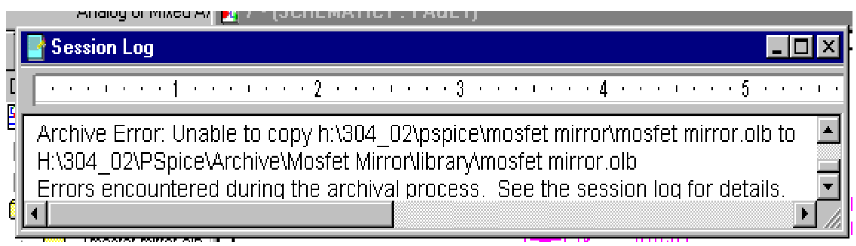


FIGURE 48:

The warning in the SESSION LOG referred to in Figure 47

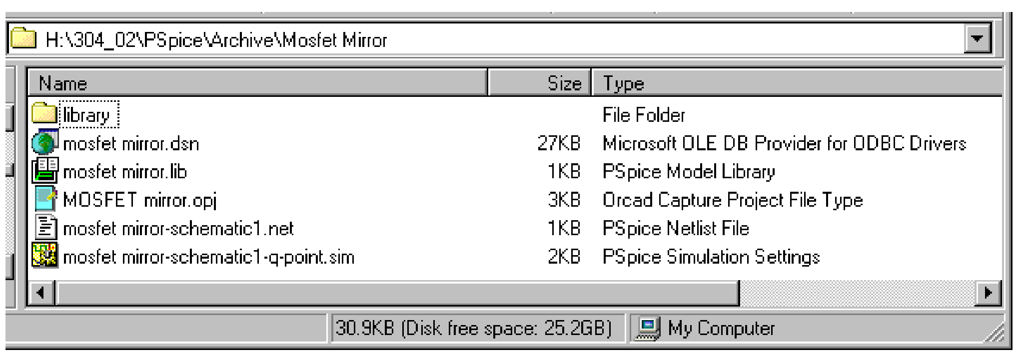


FIGURE 49:

The archived file shows that the •model file for the MOSFET has been copied in the •lib file, but the •olb file is missing. The folder labeled LIBRARY is empty.

When we double click on the •opj file to open the archived project, sometimes an error message appears that some files and libraries cannot be found. If we persist, and double click on the •opj file again, everything works OK.

So, to e-mail circuits with breakout parts, the main difference is that we have to include the •lib files that contain the •model statements for the breakout parts.

Appendix 1:
CORRECTIONS TO PSpice LOADED FROM HERNITER'S DISK[4]

(Back to Intro)

PROBLEM DESCRIPTION

If you have loaded Version 9.2 from Herniter's disk, unfortunately it contains two versions of the Q2N2222 bipolar transistor. In this situation, PSPICE works unpredictably, sometimes using one model for the Q2N2222 and sometimes the other. To avoid jumping back and forth without warning, on the department computers one of these models has been deactivated. Instructions follow on how to do this so that you can make this program on your own computer use the same model as our classes.

There are also two models for the Q2N3906 transistor and for the D1N4148 and D1N914 diodes

FINDING THE MODEL FILE

One of the Q2N2222 models is kept in a library file named *class.lib* and the other version is kept in the file *eval.lib*. We are going to eliminate the version in *class.lib*. Actually, it won't be eliminated, just commented out.

To find the file and edit it we will use the WINDOWS NOTEPAD text editor.

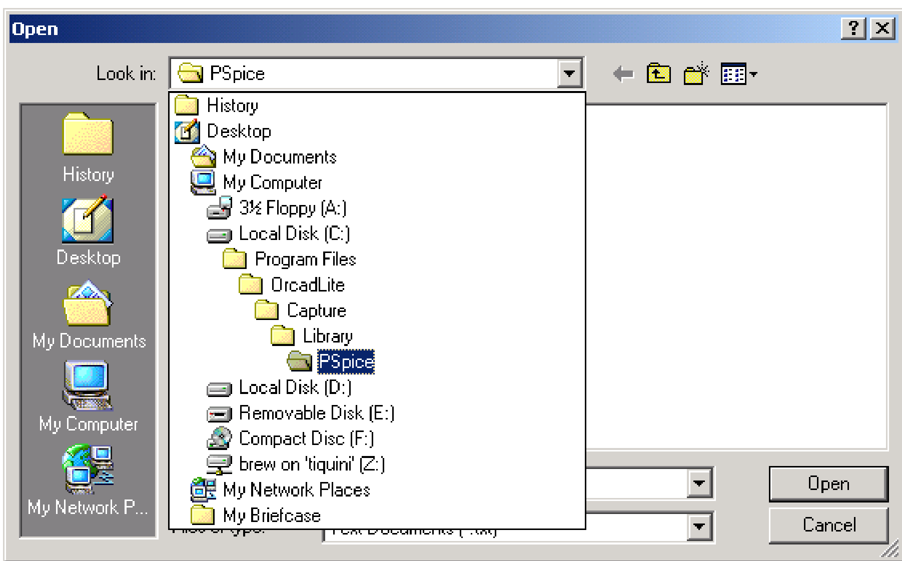


FIGURE 50:
The file hierarchy as seen from NOTEPAD's FILE OPEN menu.

[4] Updates to this book and additional material can be found on Herniter's Web Site, http://www.rose-hulman.edu/~herniter/. I haven't examined the updated •model files on this site.

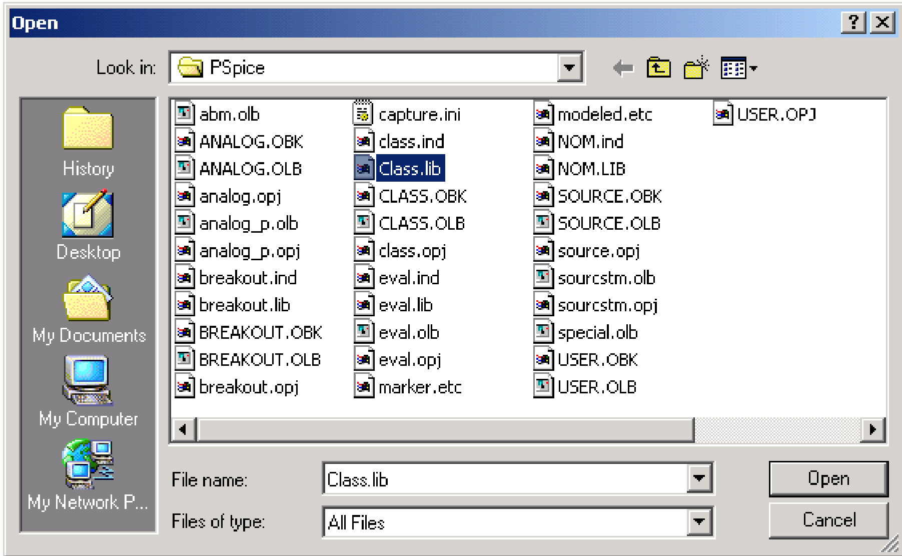
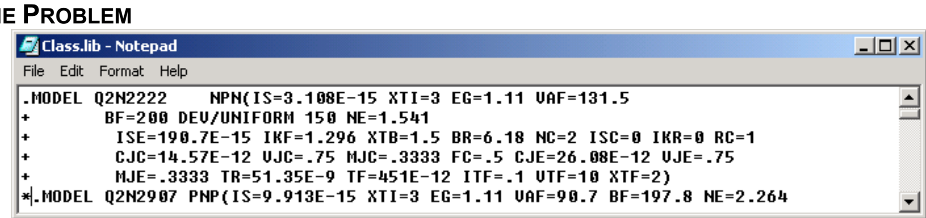


FIGURE 51:

The file *Class.lib* found in NOTEPAD using the selection Files of type ALL FILES in the bottom selection box.

FIXING THE PROBLEM

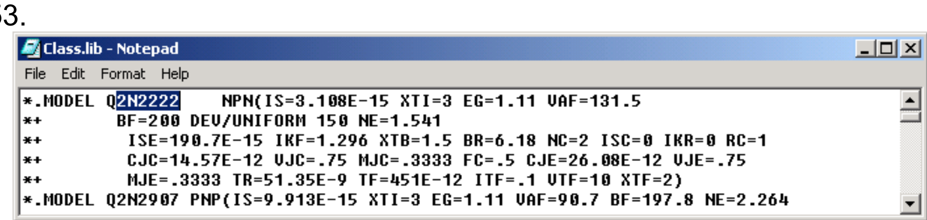



```
.MODEL Q2N2222    NPN(IS=3.108E-15 XTI=3 EG=1.11 VAF=131.5
+       BF=200 DEV/UNIFORM 150 NE=1.541
+        ISE=190.7E-15 IKF=1.296 XTB=1.5 BR=6.18 NC=2 ISC=0 IKR=0 RC=1
+        CJC=14.57E-12 VJC=.75 MJC=.3333 FC=.5 CJE=26.08E-12 VJE=.75
+        MJE=.3333 TR=51.35E-9 TF=451E-12 ITF=.1 VTF=10 XTF=2)
*.MODEL Q2N2907 PNP(IS=9.913E-15 XTI=3 EG=1.11 VAF=90.7 BF=197.8 NE=2.264
```

FIGURE 52:

The Q2N2222 device description in the library *Class.lib*.

Figure 52 shows the file we are going to edit. Rather than delete this file, we will comment it out by putting * at the beginning of each line. The modified file is shown in Figure 53.




```
*.MODEL Q2N2222    NPN(IS=3.108E-15 XTI=3 EG=1.11 VAF=131.5
*+       BF=200 DEV/UNIFORM 150 NE=1.541
*+        ISE=190.7E-15 IKF=1.296 XTB=1.5 BR=6.18 NC=2 ISC=0 IKR=0 RC=1
*+        CJC=14.57E-12 VJC=.75 MJC=.3333 FC=.5 CJE=26.08E-12 VJE=.75
*+        MJE=.3333 TR=51.35E-9 TF=451E-12 ITF=.1 VTF=10 XTF=2)
*.MODEL Q2N2907 PNP(IS=9.913E-15 XTI=3 EG=1.11 VAF=90.7 BF=197.8 NE=2.264
```

FIGURE 53:

The file in *Class.lib* with the model lines for the Q2N2222 commented out by placing * at the start of each line.

We do the same for each of the other duplicate parts, save the file and close it. That's all there is to it.

Appendix 2:

CHANGING DEFAULT VALUES IN PSpice TO OBTAIN Probe FIGURES SUITABLE FOR Word

(Back to Paste PROBE)

Changing the background color to white

PROBE uses a black background in default mode, which does not print well in documents. It is possible to make some adjustments in PROBE using the menus WINDOW/COPY TO CLIPBOARD and selecting MAKE BACKGROUND TRANSPARENT, AND CHANGE WHITE TO BLACK. The result is shown in Figure 54 below.

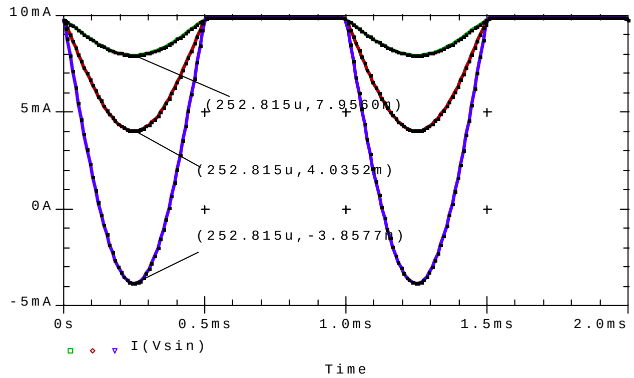


FIGURE 54:
Result of Change White to Black and making Background Transparent. Copied with Paste Special/ Picture (macro Ctrl+P). This is not WYSIWYG.

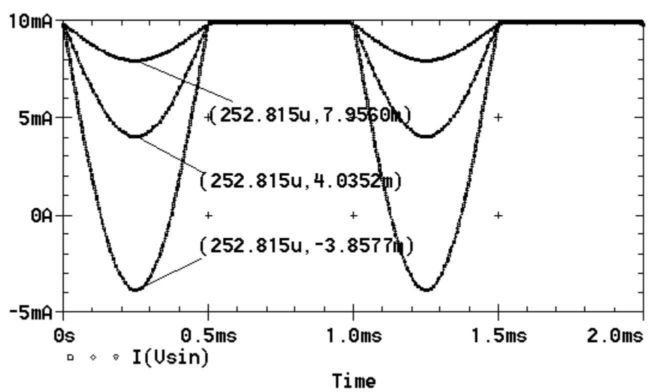


FIGURE 55:
Result of Change White to Black and making Background Transparent. Copied with Paste Special/ Device Independent Bitmap (macro Ctrl+D). This is WYSIWYG.

Changing the line colors in PROBE

If you want more control, the default color values can be changed directly in the PSPICE•ini file found in the PSPICE directory. Your choice of colors in this menu determines the palette of colors you are offered in PROBE when you right click on a curve and select PROPERTIES. Clicking on a trace you obtain the menu in Figure 56.

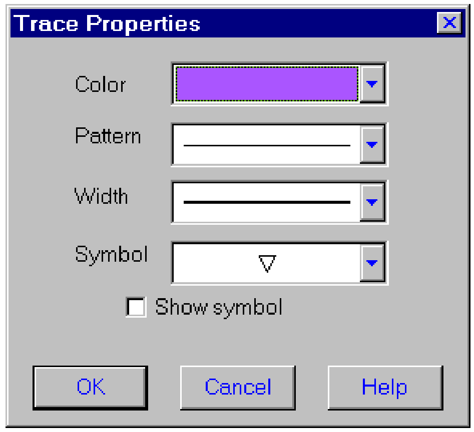


FIGURE 56:

The TRACE PROPERTIES menu

Clicking on the arrow next to the color bar, the palette menu in Figure 57 is exposed.

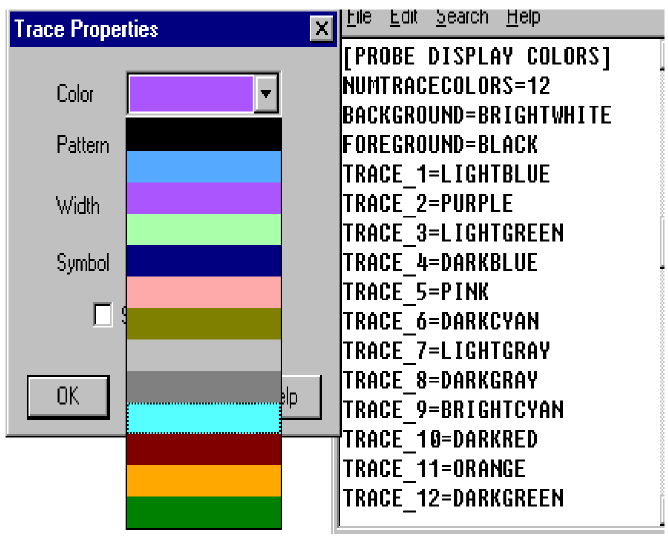


FIGURE 57:

The color palette seen when the COLOR bar is expanded. This palette is controlled by the selection of colors in the PSPICE•ini file on the right.

The palette in Figure 57 results from the •ini listing next to the color palette.

```
[PROBE DISPLAY COLORS]
NUMTRACECOLORS=12
BACKGROUND=BRIGHTWHITE
FOREGROUND=BLACK
TRACE_1=LIGHTBLUE
TRACE_2=PURPLE
TRACE_3=LIGHTGREEN
TRACE_4=DARKBLUE
TRACE_5=PINK
TRACE_6=DARKCYAN
TRACE_7=LIGHTGRAY
TRACE_8=DARKGRAY
TRACE_9=BRIGHTCYAN
TRACE_10=DARKRED
TRACE_11=ORANGE
TRACE_12=DARKGREEN
```

FIGURE 58:

The •ini listing corresponding to Figure 57. This •ini file is available on the ECE 304 web page.

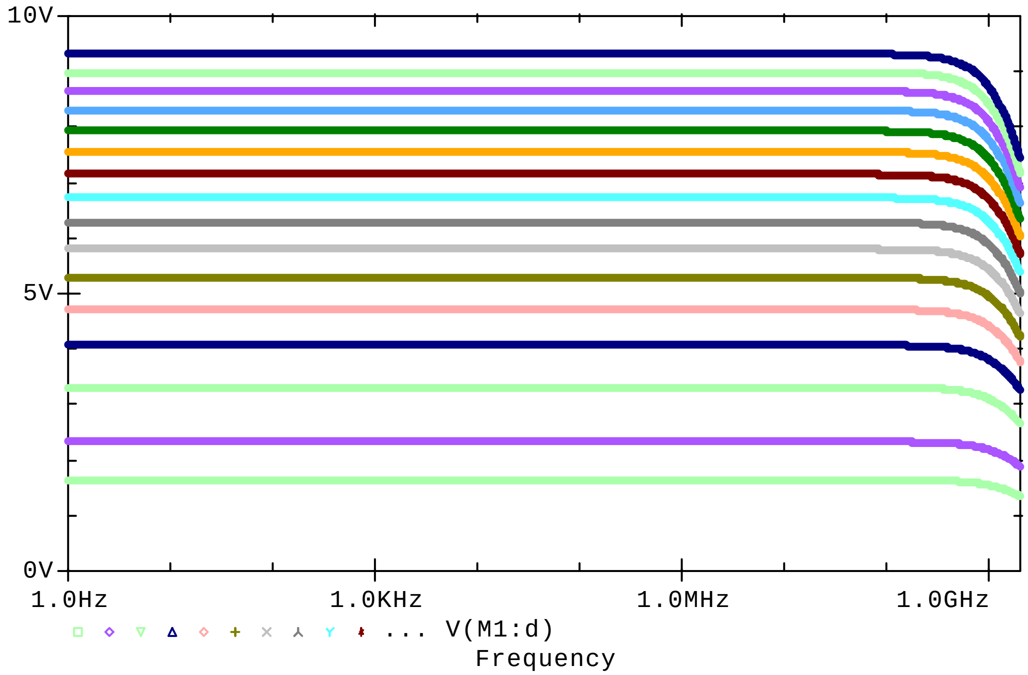


FIGURE 59:
Example plot using the palette of Figure 58. The colors repeat after 12 curves.

All the colors in the palette print on a gray-scale printer, but the darker colors print darker and are easier to see. Alternating dark with light colors makes it easier to identify neighboring curves with neighboring parameter values. The lighter colors also are useful for showing curves that nearly coincide, as shown in Figure 60.

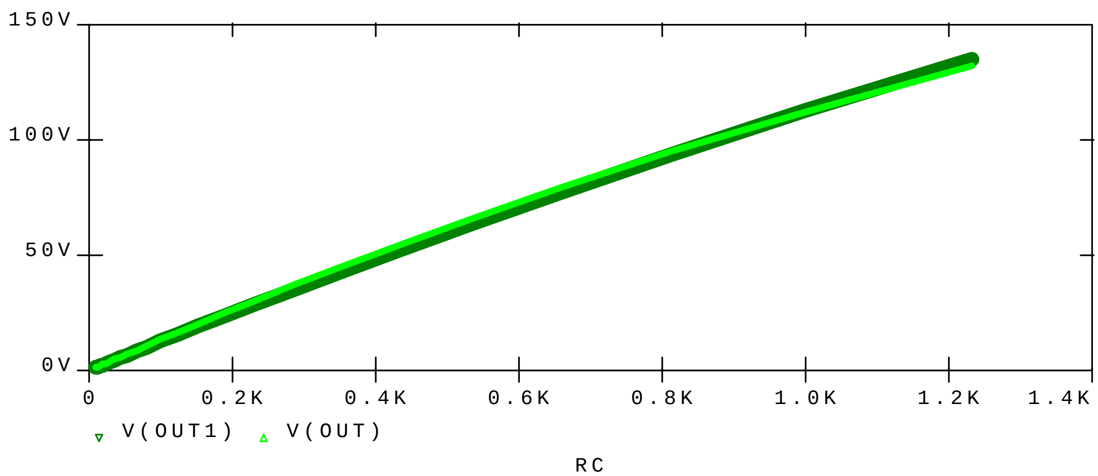


FIGURE 60:
Using a light color on a darker one where two curves almost coincide.

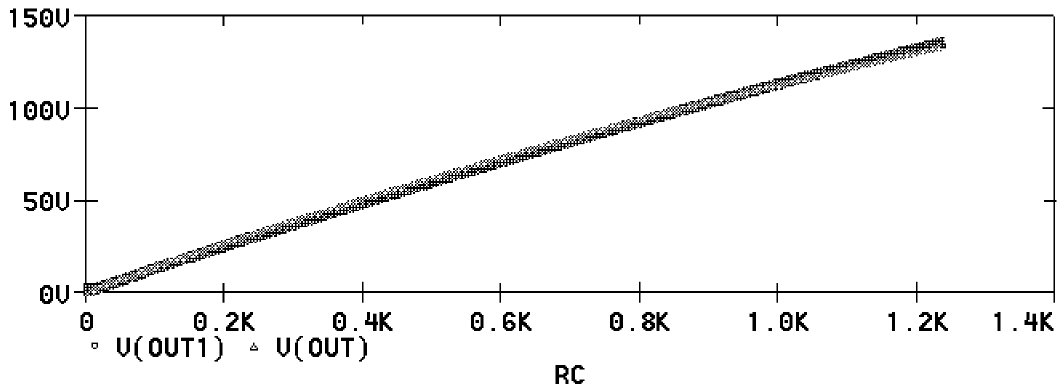


FIGURE 61:

WYSIWYG of Figure 60 (uses gray scale). In a printed figure the quality using this device-independent bitmap approach (Ctrl+D) is very much poorer than that using the paste picture (Ctrl-P) method and letting the printer determine the gray-scale.

The available colors are: black, blue, brown, brightwhite, cyan, darkblue, darkcyan, darkgray, darkgreen, darkmagenta, darkred, green, lightgray, magenta, red, and yellow, brightgreen, brightred, brightblue, brightyellow, brightmagenta, brightcyan, mustard,.pink, lightgreen, darkpink, lightblue, and purple.

Changing the trace width in PROBE

The width of the traces in probe can be changed using the PSPICE.ini file:

```
[PROBE]
PRINTERLINEWIDTH=1
PRBFILE=C:\Program Files\OrcadLite\PSpice\Common\pspice.prb
MARKDATAPOINTS=OFF
TRACESYMBOLS=ALWAYS
TRACECOLORSCHEME=NORMAL
STATUSLINEON=ON
DISPLAYEVALON=OFF
HISTNDIVISIONS=10
HISTSHOWSTATSON=ON
CURSORRIGHT=204
CURSORBOTTOM=584
CURSORNDIGITS=5
ERRORMSGDLGLEFT=-1
ERRORMSGDLGTOP=-1
DISPLAYTOOLBAR=ON
TRACEWIDTH=4
```

FIGURE 62

Section of the PSPICE.ini file showing the line to alter to increase width of plotted curves in PROBE. The default value is TRACEWIDTH=2

Appendix 3

WHEN ORCAD CANNOT FIND THE FILE

Lost •DSN file – missing schematics!

Fairly frequently, you open a project and there is no schematic! An example is shown in Figure 63. To fix this problem, we have to attach the real •DSN file, which is clearly present in the file folder for the project when we look with the Windows File Explorer as seen in Figure 64. The •DSN file listed in Figure 63 is only a dummy, with no schematics.

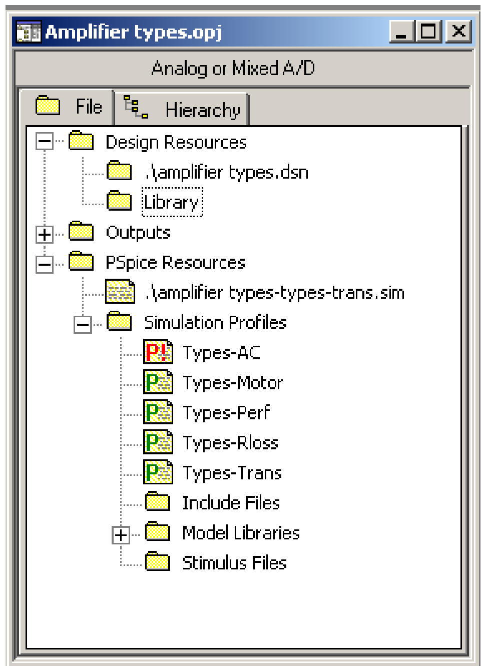


FIGURE 63:

The • OPJ listing shows no schematic! The •DSN file listed is just a placeholder, and contains no schematics.

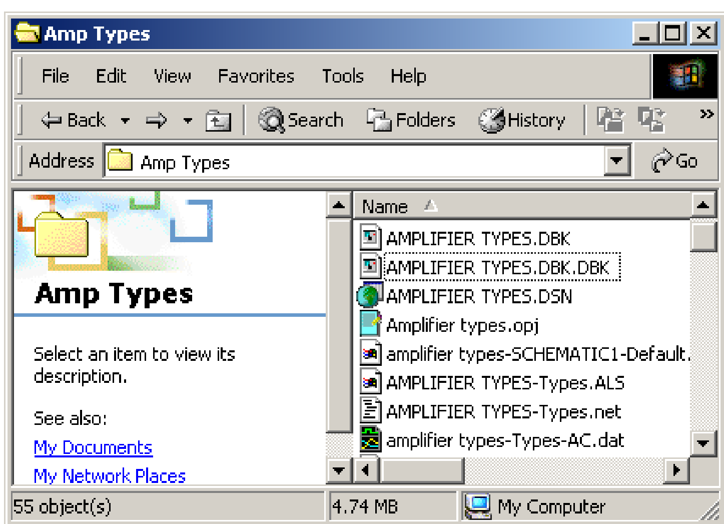


FIGURE 64:

The •DSN file is in the project folder

If the dummy •DSN file is present as shown in Figure 63, we have to remove it by highlighting it in this menu and hitting the DELETE key. Then, to incorporate the real •DSN file we can proceed two ways:

- With the mouse drag the •DSN file from the Window Explorer over to the ORCAD menu and drop it in the DESIGN RESOURCES folder
- Formally add the •DSN file using the Orcad ADD FILE utility.

To follow the last method, highlight the DESIGN RESOURCES folder and right-click to obtain the ADD FILE menu, as shown in Figure 65.

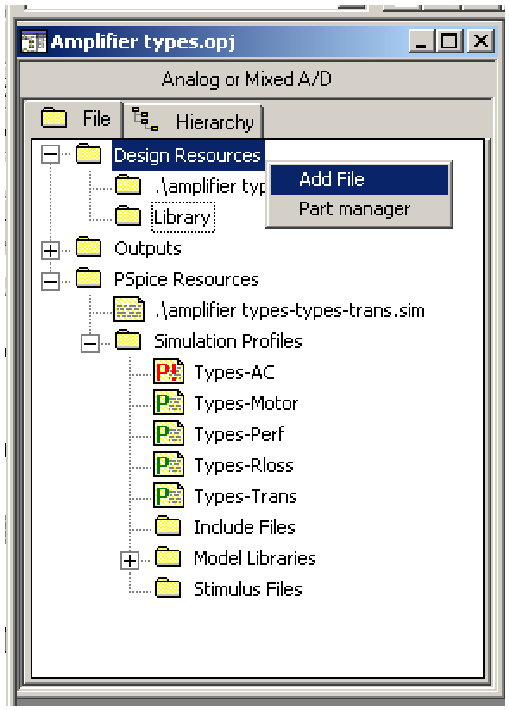


FIGURE 65:
Obtaining the ADD FILE menu

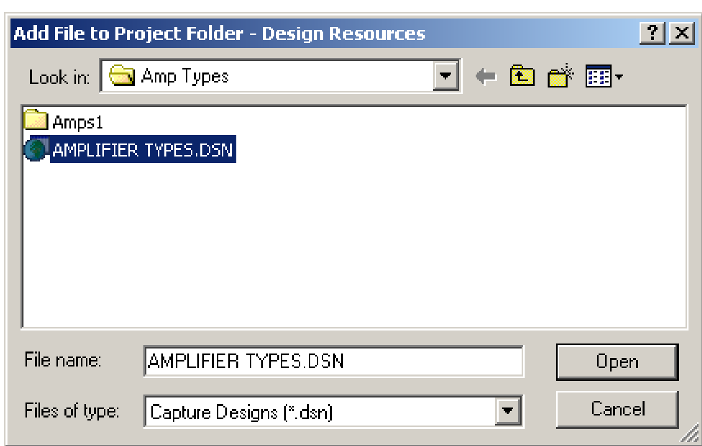


FIGURE 66:
The ADD FILE menu. Select the •DSN file.

As shown in Figure 66, select the • DSN file and click OPEN.

Lost •SIM – missing simulation profile!

It is extremely common for Orcad to lose a Simulation Profile[5]. When you look in the Window's File Explorer, the simulation profile is there, but Orcad refuses to find it until you try to recreate it, when it tells you that your Simulation Profile already exists! Go figure! You then have at least three choices:

- Create the Simulation Profile again under a new name
- Copy the schematic into a new project and reintroduce all the Simulation Profiles
- Edit the •OPJ file using NOTEPAD

It may be that the first choice is most expedient, but in case you want to edit the •OPJ file, here's how.

In the •OPJ listing in the Orcad window, select PSPICE RESOURCES and right click to get the ADD FILE menu, see Figure 67.

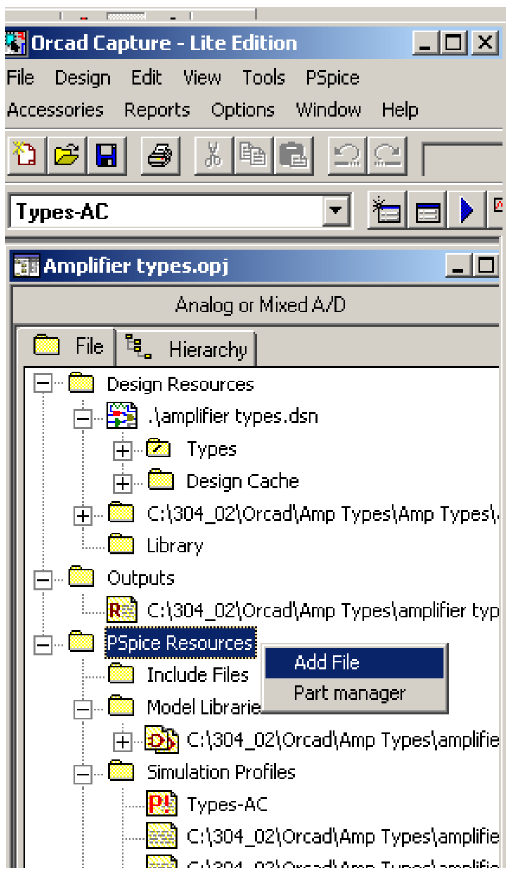


FIGURE 67:
Obtaining the ADD FILE menu

[5] One cause is the system crashing – the auto-recovery loses or rewrites files

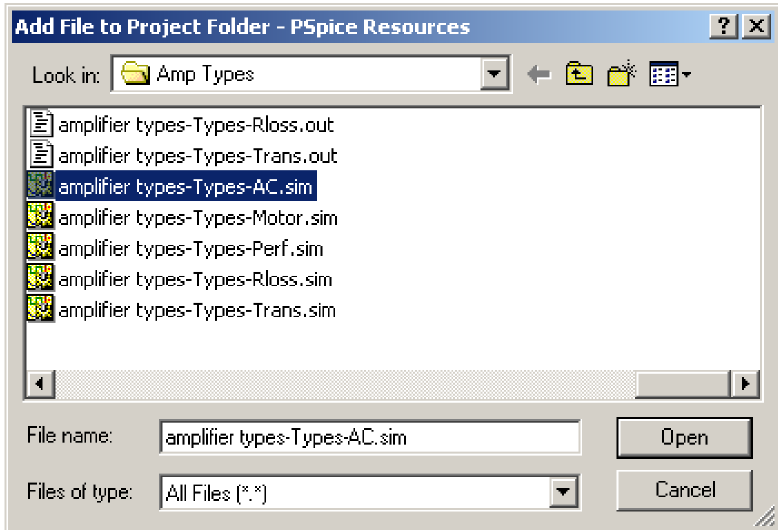


FIGURE 68:

Arranging to add the •SIM files to •OPJ

With the ADD FILE menu, select the ALL FILES option so you can see the •SIM files. Highlight these files and add them, as shown in Figure 68.

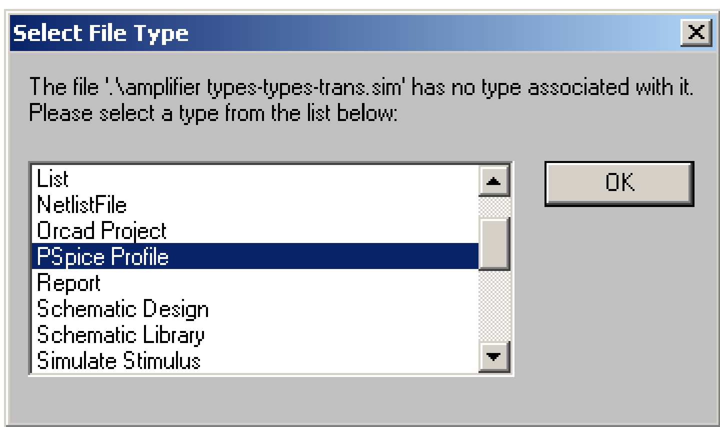


FIGURE 69:

Identifying the file type as PSPICE PROFILE

As shown in Figure 69, identify them as type PSPICE PROFILE[6]. Once they are added, drag them down to the SIMULATION PROFILES block of the PSPICE RESOURCES as shown in Figure 67.

Now close CAPTURE. Next, open the •OPJ file using NOTEPAD. Find the section under PSPICE RESOURCES/ (FOLDER "SIMULATION PROFILES" as shown below.

Name of Project
Name of Schematic
Name of Profile

(FOLDER "PSPICE RESOURCES"
(FOLDER "SIMULATION PROFILES"
(FILE. "C:\304_02\ORCAD\AMP TYPES\AMPLIFIER TYPES-TYPES-AC.SIM"
(TYPE "PSPICE PROFILE")))

[6] If this option is not listed, don't worry: pick another entry. You can change it later in NOTEPAD.

Replace the (File <pathname>) with the following[7]

```
(FILE.PSPICE. "C:\304_02\ORCAD\AMP TYPES\AMPLIFIER TYPES-TYPES-AC.SIM"
    (DISPLAYNAME " TYPES-AC")
    (TYPE "PSPICE PROFILE")))
```

That is, FILE becomes FILE•PSPICE•[8] and a DISPLAYNAME line has to be added. After doing this for each of the missing simulation profiles, save the •OPJ file, and reopen the project using CAPTURE. (If you didn't close CAPTURE earlier, do it now, resave the • OPJ file in NOTEPAD, and reopen CAPTURE. That way, CAPTURE gets the modified •OPJ file.) The edited •OPJ file with one more profile added is shown below.

```
(FOLDER "PSPICE RESOURCES"
  (FOLDER "SIMULATION PROFILES"
   (FILE.PSPICE. "C:\304_02\ORCAD\AMP TYPES\AMPLIFIER TYPES-TYPES-TRANS.SIM"
    (DISPLAYNAME " TYPES-TRANS")
    (TYPE "PSPICE PROFILE"))
   (FILE.PSPICE. ".\ TYPES-AC.SIM"
    (DISPLAYNAME "TYPES-AC")
    (TYPE "PSPICE PROFILE")))
```

The missing Simulation Profiles now will reappear.

A minor variation on this theme is that one of the profiles may appear further down the •OPJ listing under the heading ACTIVE PROFILE. Once found, it is changed in the same way, as shown below

```
(ACTIVEPROFILE ".\AMPLIFIER TYPES-TYPES-AC.SIM")
      (FILE.PSPICE. ".\AMPLIFIER TYPES-TYPES-TRANS.SIM"
      (DISPLAYNAME "TYPES-TRANS")
      (TYPE "PSPICE PROFILE")))
```

Missing •OLB file
This file often is missing from an e-mailed or archived project, and causes a warning menu to show up that not all the project files can be found. Then a second attempt usually opens the project. Once the project opens, assuming you have the *Parts.lib* library (*Parts* happens to be the project name in this example), you can make the *Parts.olb* library as follows.

Highlight a part in the *Parts* schematic. Then using EDIT/PSPICE MODEL, select the menu FILE/CAPTURE PARTS. The CREATE PARTS FOR LIBRARY menu appears, as shown in Figure 70. Fill in the location of the *Parts.lib* library using the BROWSE button. Then hit OK.

---

[7] If the TYPE is incorrectly listed, change the TYPE to (Type "PSPICE Profile"))

[8] Don't forget the final period in FILE•PSPICE• file.

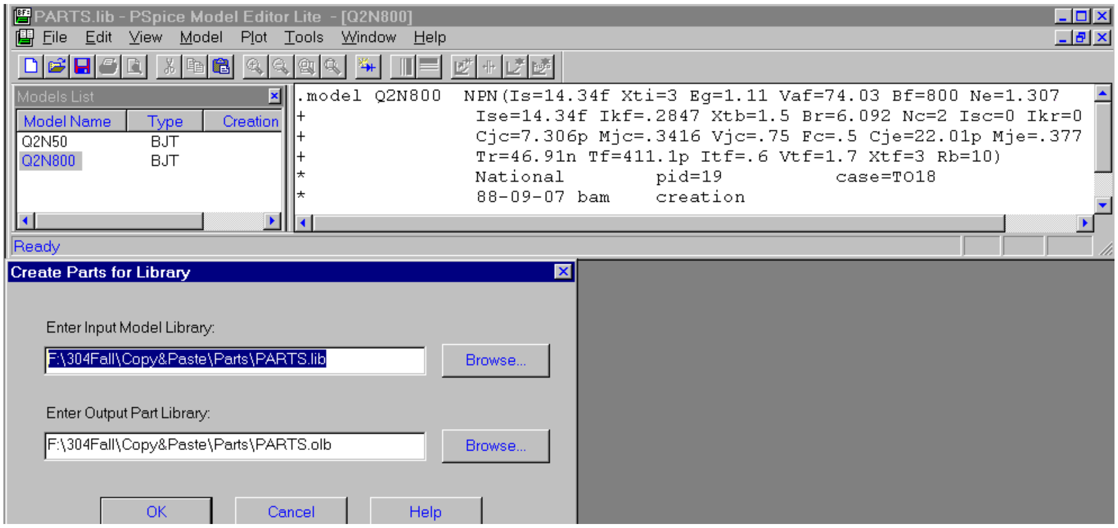


FIGURE 70

Creation of the *Parts•olb* model library using the PSPICE model editor with FILE/CAPTURE PARTS. The *Parts.lib* library is filled in using the BROWSE button.