ECE 304 Using PSPICE with WORD and EXCEL

Table	of	Contents
IUDIC	v.	Contonio

Introduction	6
Creating a Schematic	6
Figure 1:	7
Current mirror schematic. Note the emitter-leg resistors of value $R_E = 1k\Omega$	7
Setting Font Sizes on the Schematic	7
Figure 2	7
Menu for setting font characteristics and labels for currents and voltages on the	
schematic	
Figure 3	8
Menu found in OPTIONS/DESIGN TEMPLATE from the main schematic toolbar	
Running a Q-point Simulation	8
Figure 4:	
Screen dump of simulation settings for the Q-point simulation.	
Figure 5:	9
Screen dump showing how OUTPUT FILE can be accessed from PROBE	
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	J _
the Excel menu DATA/TEXT TO COLUMNS	
Pasting Schematics into WORD	
Figure 7:	.10
Current-mirror circuit	
DC I-V Characteristics of Current Mirror Using PROBE	
Figure 8:	.10
Screen dump of simulation settings for DC sweep of global parameter DC	
Pasting a Probe Plot into Word	
Figure 9:	.11
PROBE plot of DC <i>I-V</i> curve for the current mirror in Figure 1 using PASTE	
SPECIAL/PICTURE (macro Ctrl+P). This figure is not WYSIWYG (what you see is what	
you get). Compliance voltage label is added in WORD	
Figure 10:	11
PROBE plot for current mirror in Figure 1 using PASTE SPECIAL/DEVICE INDEPENDENT	11
BITMAP (macro Ctrl+D). This figure is WYSIWYG Pasting PSpice Probe Data into Excel	
•	
Figure 11: Screen dump from ExceL showing selection of chart type using the chart wizard	
Figure 12: Example ExceL chart of the data in Figure 10	
Pasting Excel Charts and Tables into Word	
DC determination of Norton Resistance: Using PSPICE D() for Derivative	
Figure 13:	
Reciprocal of derivative of the DC <i>I-V</i> plot of Figure 9 to find the output resistance of	.13
the mirror.	
Norton Resistance of Current Mirror: AC Frequency Sweep and Macros	
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.	
	. т-т

Figure 15:	
Screen dump of MACROS menu showing the definition of several macros	
Figure 16:	
Norton resistance of current mirror vs. frequency.	
AC Mirror Current in Response to Sinusoidal Output Voltage: Transient Analysis	
Figure 17:	
Simulation settings for transient analysis. A parametric sweep of the amplitude of t	
AC voltage, variable VAMPL is used	
Figure 18:	
Setup for the global parameter VAMPL that determines the amplitude of the sinuso voltage across the mirror. A VALUE LIST of three amplitudes is used to get a family curves.	of
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 curv	
symbol brings up a menu identifying the parameter value for a particular curve	
Fourier Analysis and Harmonic Distortion	
Figure 20:	
Screen dump of menu activating Fourier analysis. Pressing the OUTPUT FILE OPTIO	
bar with GENERAL SETTINGS selected on the TRANSIENT simulation profile accesses	
menu	
Figure 21:	
Screen dump showing how to access the OUTPUT FILE from PROBE.	
Figure 22: Harmonic distortion from the PSPICE output file. The output was pasted into EXCEL	
format it this way using the EXCEL option DATA/TEXT TO COLUMNS. Total harmonic	ιο
distortion is ≈ 42%.	18
	10
Effect of Maximum Step Size	19
Effect of Maximum Step Size	
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23:	19 19
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence	19 19 cies
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested	19 19 cies 19
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested. Figure 24:	19 19 cies 19 20
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested Figure 24: Screen dump for the parametric sweep in the performance analysis	19 19 cies 19 20 20
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25:	19 19 19 20 20 20
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested Figure 24: Screen dump for the parametric sweep in the performance analysis	19 19 19 20 20 20
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz vs. the emitter-leg resi	19 19 19 20 20 20 stor
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 cies 20 20 20 stor 20
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 cies 20 20 20 stor 20 stor
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 20 20 stor 20 stor 21 e 21
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested. Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz vs. the emitter-leg resivalue R_E. The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage. 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: 	19 19 20 20 stor 20 stor 21 e 21
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 20 20 stor 20 stor 21 e 21 21
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz <i>vs.</i> the emitter-leg resister value R _E . The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage 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 <i>vs.</i> emitter-leg resistor value R _E . The compliance voltage reaches the supply voltage when R _E = 1.44 kΩ.	19 19 20 20 20 stor 21 e 21 e 21
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23:	19 19 cies 19 20 20 20 stor 21 e 21 21 21 22
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested. Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz <i>vs.</i> the emitter-leg resi value R _E . The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage. 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 <i>vs.</i> emitter-leg resistor value R _E . The compliance voltage reaches the supply voltage when R _E = 1.44 kΩ. Using Goal Functions Figure 28:	19 19 20 20 20 20 stor 21 e 21 21 21 22 22
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested. Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz vs. the emitter-leg resi value R _E . The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage. 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. Using Goal Functions Figure 28: Schematic of circuit for demonstration of GOAL FUNCTIONS; an external capacitor C	19 19 cies 19 20 20 stor 21 e 21 e 21 21 22 22
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23: Screen dump for performance analysis plot. Notice that the start and stop frequence are the same and only one point/decade is requested. Figure 24: Screen dump for the parametric sweep in the performance analysis Figure 25: Performance analysis output for Norton resistance at 1 kHz vs. the emitter-leg resis value R _E . The DC voltage across the mirror is 15V, the largest it can be without exceeding the supply voltage. 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. Using Goal Functions Figure 28: Schematic of circuit for demonstration of GOAL FUNCTIONS; an external capacitor C modifies the frequency response of the amplifier.	19 19 20 20 stor 21 e 21 21 21 21 22 22 22
Norton Resistance <i>vs.</i> Emitter-leg Resistance: Performance Analysis Figure 23:	19 19 20 20 20 stor 21 e 21 21 21 22 22 22
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 20 20 20 20 stor 21 21 21 21 21 22 22 22 22 ip of
Norton Resistance vs. Emitter-leg Resistance: Performance Analysis Figure 23:	19 19 20 20 20 20 stor 21 e 21 21 21 22 22 22 22 22 22 22 22 22
 Norton Resistance vs. Emitter-leg Resistance: Performance Analysis	19 19 20 20 20 20 stor 21 e 21 21 21 22 22 22 22 22 22 22 22 22

	23
Setting up the parametric sweep of C _{OMP}	
Figure 32:	
Checking the PERFORMANCE ANALYSIS box on the ACCESS SETTINGS menu	24
Figure 33:	24
Figure 34:	25
Locating the available GOAL FUNCTIONS	25
Figure 35:	25
Selecting XatNthY from the listed functions	
Figure 36:	
The description of the goal function XatNthY found using the EDIT tab	
Figure 37:	
Filling in the arguments of XatNthY; the function P(.) finds the phase of its argumer	nt
(link back to directions)	
Figure 38:	
Result of performance analysis using goal function XatNthY to find the values of C	
where the phase of the gain is -225°	
Current Mirror Output vs. Reference Current and R_E : Double Sweep	
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 setting the secondary sweep of R_E from a secondary sweep of R_E from a setting the secondary sweep of R_E from a setting the secondary sweep of R_E from a seco	
VALUE LIST	
Figure 41:	
Current from current mirror vs. reference current I_{REF} for DC = 15V and several values	
of emitter-leg resistor value R _E	
Figure 42:	29
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,	
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar	ne
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want.	ne 29
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail	ne 29 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43:	ne 29 30 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE.	ne 29 30 30 30 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	ne 29 30 30 30 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat	ne 29 30 30 30 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	ne 29 30 30 30 30 h
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	me 29 30 30 30 30 h
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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.	me 29 30 30 30 30 h
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	ne 29 30 30 30 30 30 30 30
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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.	ne 29 30 30 30 30 h 30 h 31 31
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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:	ne 29 30 30 30 30 h 30 h 31 31 31
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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.	ne 29 30 30 30 30 30 h 30 31 31 31 31
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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:	ne 29 30 30 30 30 30 h 30 31 31 31 31
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail. Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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. Archiving a circuit with breakout parts Figure 47:	ne 29 30 30 30 30 h 30 h 31 31 31 31 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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. Archiving a circuit with breakout parts.	ne 29 30 30 30 30 h 30 h 31 31 31 31 32 32
 exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail. 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. Archiving a circuit with breakout parts. Figure 47: Warning message when archiving a circuit with breakout parts. Figure 48: 	ne 29 30 30 30 30 30 h 30 31 31 31 31 31 32 32 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail. Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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. Archiving a circuit with breakout parts. Figure 47: Warning message when archiving a circuit with breakout parts	ne 29 30 30 30 30 30 h 30 31 31 31 31 31 32 32 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	ne 29 30 30 30 30 30 h 31 31 31 31 31 31 32 32 32 32 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want	ne 29 30 30 30 30 30 h 31 31 31 31 31 31 32 32 32 32 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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. Archiving a circuit with breakout parts 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 th	ne 29 30 30 30 30 30 30 30 30 30 31 31 31 31 32 32 32 32 32 e
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics by E-mail	ne 29 30 30 30 30 30 h 30 30 h 31 31 31 31 32 32 32 32 32 32 e 32 32 32
exercise. The schematic name, SCHEMATIC1, is attached to the name of the profile, for identification in case you have several schematics in this project. You can renar the schematics to be more descriptive if you want. Sending Schematics by E-mail Figure 43: The FILE/ARCHIVE menu selection in CAPTURE. Figure 44: The ARCHIVE PROJECT menu resulting from selecting ARCHIVE in Figure 44. The pat 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. Archiving a circuit with breakout parts 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 th	ne 29 30 30 30 30 30 30 30 30 30 30 31 31 31 31 32 32 32 32 32 e 32 e 33

Problem Description	.33
Finding the Model File	.33
Figure 50:	
The file hierarchy as seen from NOTEPAD's FILE OPEN menu.	
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	
Figure 52:	
The Q2N2222 device description in the library <i>Class.lib</i>	
Figure 53:	
The file in <i>Class.lib</i> with the model lines for the Q2N2222 commented out by placing	
at the start of each line.	
Appendix 2:	
Changing Default Values in PSpice to Obtain Probe Figures Suitable for Word	
Changing the background color to white	
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	
Figure 56:	
The TRACE PROPERTIES menu	
Figure 57:	.36
The color palette seen when the COLOR bar is expanded. This palette is controlled b	уy
the selection of colors in the PSPICE-ini file on the right	. 36
Figure 58:	.36
The •ini listing corresponding to Figure 57. This •ini file is available on the ECE 304	
web page	. 36
Figure 59:	.37
Example plot using the palette of Figure 58. The colors repeat after 12 curves	
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	ne
paste picture (Ctrl-P) method and letting the printer determine the gray-scale.	
Changing the trace width in PROBE	
Figure 62	
Section of the PSPICE.ini file showing the line to alter to increase width of plotted	.00
curves in Proble. The default value is TRACEWIDTH=2	38
Appendix 3	
When Orcad cannot find the file	
Lost •DSN file – missing schematics!	
Figure 63:	.39
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	
Figure 65:	
Obtaining the ADD FILE menu	. 40

Figure 66:	.40
The ADD FILE menu. Select the •DSN file.	
Lost •SIM – missing simulation profile!	.41
Figure 67:	.41
Obtaining the ADD FILE menu	. 41
Figure 68:	
Arranging to add the •SIM files to •OPJ	. 42
Figure 69:	.42
Identifying the file type as PSPICE PROFILE	. 42
Missing •OLB file	.43
Figure 70	.44
Creation of the <i>Parts-olb</i> model library using the PSPICE model editor with FILE/CAPTURE PARTS. The <i>Parts.lib</i> library is filled in using the BROWSE button	

ECE 304: Using PSPICE with WORD and EXCEL

Introduction

This exercise familiarizes you with Microsoft WORD '97 and 2000 and Cadence PSPICE, version 9.2¹. Your job here is to duplicate the Table of Contents, <u>all</u> 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 [$\frac{1}{-}$] 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 <u>family of curves</u> 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 <u>Appendix 1</u>

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

- 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)

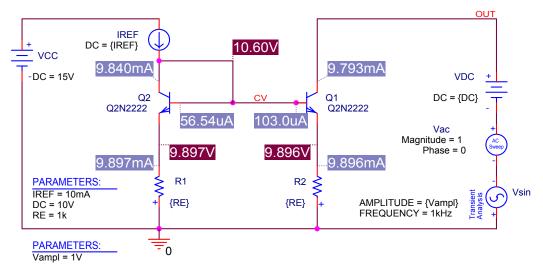


FIGURE 1:

Current mirror schematic. Note the emitter-leg resistors of value $R_E = 1k\Omega$. (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.

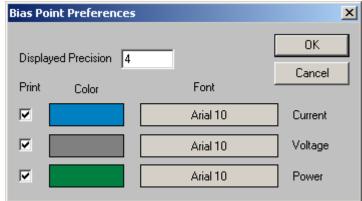


FIGURE 2

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.

Design Template			×
Fonts Title Block Page Si	ze Grid Reference Hiera	rchy SDT Compatibility	
Arial 10	Alias	Arial 10	Pin Name
Arial 10	Bookmark	Arial 10	Pin Number
Arial 10	Border Text	Arial 10	Port
Arial 10	Hierarchical Block	Arial 10	Power Text
Arial 10	Net Name	Arial 10	Property
Arial 10	Off-Page Connector	Arial 10	Text
Arial 10	Part Reference	Arial 10	Title Block Text
Arial 10	Part Value		
		OK Cano	el Help

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.

Simulation Settings - Q point	
General Analysis Include Files	Libraries Stimulus Options Data Collection Probe Window
Analysis type: Bias Point Options: General Settings Temperature (Sweep) Save Bias Point Load Bias Point	Output File Options Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP) Perform Sensitivity analysis (.SENS) Output variable(s): Calculate small-signal DC gain (.TF) From Input source name: To Qutput variable:

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.

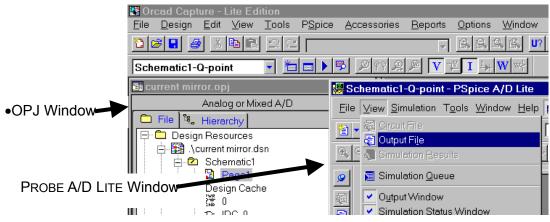


FIGURE 5:

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

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

**** BIPOLAR JUNCTION TRANSISTORS

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.

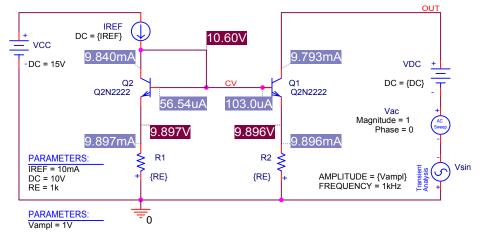


FIGURE 7:

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.

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window Analysis type:	
Analysis type:Sweep variable	
DC Sweep O Voltage source Marrie:	
Ontions: Model type:	1
© Global parameter	-
C Model parameter Model rame:	
□Monte Carlo/Worst Case	
Sweep type C Logarithmic Decade Increment: 1 C Value ligt	

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 <u>Appendix 2</u>, choose USE SCREEN COLORS. Moving to WORD, we can use the Ctrl+P macro. The plot is in Figure 9 below.

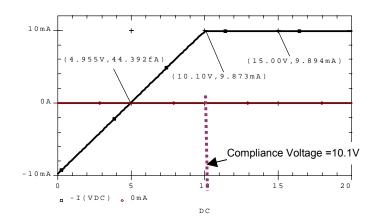


FIGURE 9:

PROBE plot of DC *I-V* curve for the current mirror in Figure 1 using PASTE SPECIAL/PICTURE (macro CtrI+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.

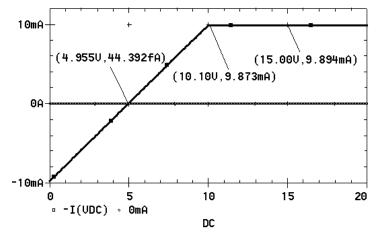


FIGURE 10:

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.

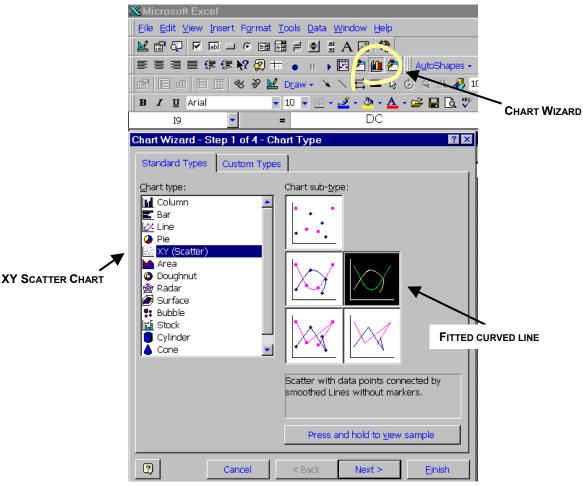


FIGURE 11:

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.

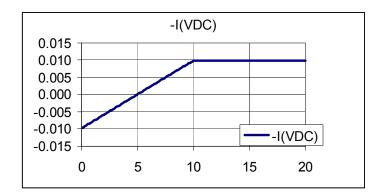


FIGURE 12:

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.

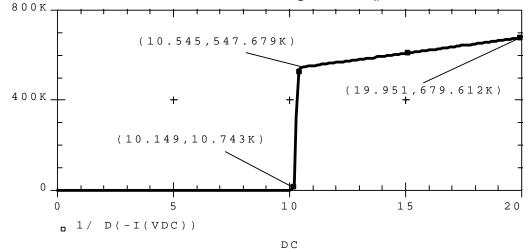




FIGURE 13:

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 <u>small-signal</u> 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.

mulation Settings - AC Sweep General Analysis Include Files	Libraries Stimulus C	ptions Data Collection Prob	e Window
Analysis type: AC Sweep/Noise Options: General Settings	AC Sweep Type – O Linear © Logarithmic Decade	Start Frequency: End Frequency: Points/Decade:	1 1G 101
☐Monte Carlo/Worst Case ☐Parametric Sweep ☐Temperature (Sweep) ☐Save Bias Point ☐Load Bias Point	Noise Analysis — □ E <u>n</u> abled	O <u>u</u> tput Voltage: I/ <u>V</u> Source: Interval:	
		ed bias point information for non semiconductors (.OP)	inear controlled

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 $V_{ac} = 1V$, 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.

Macros	
Definition: $Rp(x) = 1/R(x)$	<u>S</u> ave
Cp(x) = IMG(x)/(2*pi*Frequency) pi = 3.14159265	Save <u>T</u> o
Rp(x) = 1/R(x)	Delete
	Load

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.

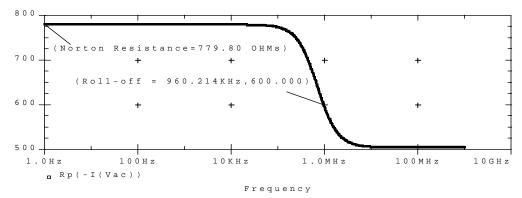


FIGURE 16:

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 <u>not</u> 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.

nalysis type:	
ïme Domain (Transient) 💽	Run to time: 2e-03 seconds (TSTOP)
ptions:	Start saving data after: 0 seconds
General Settings	
Monte Carlo/Worst Case	Maximum step size: 1E-05 seconds
]Parametric Sweep]Temperature (Sweep)	\Box Skip the initial transient bias point calculation (SKIPBP)
]Save Bias Point	
]Load Bias Point	Output <u>File Options</u>

FIGURE 17:

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

Analysis type:	-Sweep variable		
Time Domain (Transient) 💌	C ⊻oltage source	<u>N</u> ame:	
 Options:	C <u>C</u> urrent source	Model type:	_
⊴ptions. ■General Settings	 Global parameter 	Model name:	
☐Monte Carlo/Worst Case ☑Parametric Sweep	C <u>M</u> odel parameter C <u>T</u> emperature		/ampl
□Temperature (Sweep) □Save Bias Point	Sweep type		
 □Load Bias Point	O <u>L</u> inear	Sta <u>r</u> t value:	
	C Logarit <u>h</u> mic Deca		
		Increment:	

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.

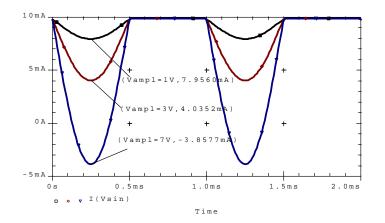


FIGURE 19:

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 = <u>power</u> contained in the harmonics (*i.e.* the square root of the sum of squares of amplitudes).

Transient Output File Options Print values in the output file event ✓ Perform Fourier Analysis enter Frequency: 1kit Number of Harmonics: 20 Output Variables: Iv	HZ hz hz	Window Help 3. 07 11 % m m W1 W2 Probe Window ds (TSTOP) ds
General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	<u>Transiencoptions</u> <u>M</u> aximum step size: 1E-05 secor	

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.

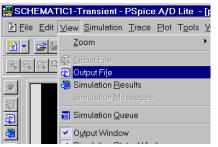


FIGURE 21:

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

DC COMPONE HARMONIC					
NO	(HZ)	COMPONENT	COMPONENT	(DEG)	PHASE (DEG)
NO	(112)	COMPONENT	COMPONENT	(DLG)	FIASE (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

FOURIER COMPONENTS OF TRANSIENT RESPONSE I(V_Vsin)

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 $\approx 42\%$.

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 <u>not</u> the true output of the amplifier. <u>Too large a step size leads to an incorrect harmonic distortion analysis.</u> 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.

Simulation Settings - Performance				×
General Analysis Include Files Li Analysis type: Ac Sweep/Noise Image: Colspan="2">Image: Colspan="2" Image: Colspan="">" Image: Colsp	AC Sweep Type	<u>S</u> tart F <u>E</u> nd F	Frequency:	
		d bias point informe emiconductors (.OF		ear controlled
	ОК	Cancel	Apply	Help

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.

eneral Analysis Include Fi		ns Data Collection Probe Wind	low
Analysis type: AC Sweep/Noise	- Sweep variable	Name:	
Options:	<u>Current source</u> <u>Global parameter</u>	Model type:	~
deneral Settings ⊡Monte Carlo/Worst Case	O <u>M</u> odel parameter	Model name:	
✓Parametric Sweep □Temperature (Sweep)	C Temperature	Parameter name: RE	
□Save Bias Point □Load Bias Point	© Linear	Sta <u>r</u> t value: 1u	
		de End value: 10k Points/Decade: 11	—
	O Value li <u>s</u> t		
	ОК	Cancel <u>A</u> pply	Help

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.

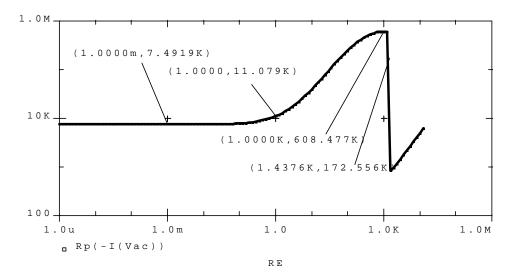


FIGURE 25:

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 k Ω for R_E < 1 Ω , but as R_E increases, the Norton resistance rapidly climbs to 608.5 k Ω . Then it suddenly drops. What is the cause of this drop in R_N for R_E in the range 1 k $\Omega \le R_E \le 1.5 \text{ k}\Omega$?

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.

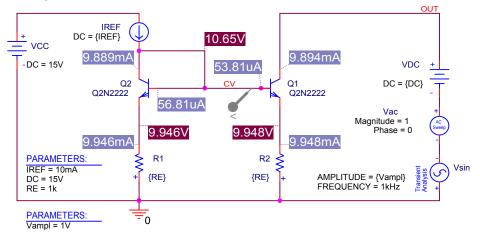


FIGURE 26:

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.

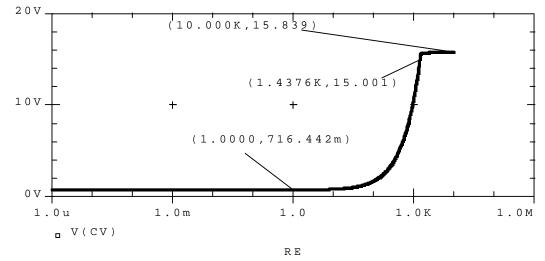


FIGURE 27:

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

(Back to list of figures.)

From Figure 27 we can see that indeed the compliance voltage exceeds the supply voltage when $R_E \ge 1.44 \text{ k}\Omega$, which means that the output transistor Q1 will go into saturation for $R_E \ge 1.44 \text{ k}\Omega$, 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 -45° as a function of C_{OMP} . The circuit is shown in Figure 28.

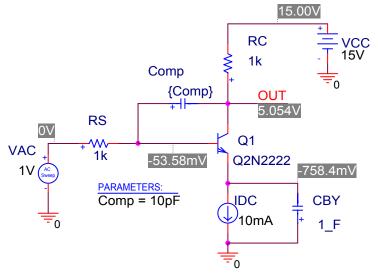


FIGURE 28:

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 -45° 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.

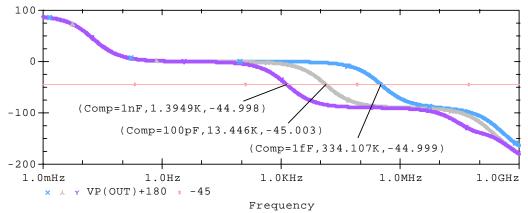


FIGURE 29:

Phase of gain of circuit in Figure 28 (with 180° 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.

	U U		
Simulation Settings - Comp			×
General Analysis Include Files	Libraries Stimulus Op	otions Data Collection Probe Windo	w] _
Analysis type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case	AC Sweep Type C Linear C Logarithmic Decade	Start Frequency: 1 End Frequency: 1G Points/Decade: 101	
 ✓ Parametric Sweep ☐ Emperature (Sweep) ☐ Save Bias Point ☐ Load Bias Point 		put Voltage: Source: Sourc	
		ias point information for nonlinear and semiconductors (.DP)	
	ОК С	Cancel Apply Help	

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

Simulation Settings - Comp			×
General Analysis Include File	es Libraries Stimulus	Options Data Collection	Probe Window
General Analysis Include File Analysis type: AC Sweep/Noise Image: Comparison of the system Options: Image: Comparison of the system Image: Comparison of the system Monte Carlo/Worst Case Image: Comparison of the system Image: Comparison of the system Monte Carlo/Worst Case Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the system Image: Comparison of the syst	s Libraries Stimulus Sweep variable O Voltage source O Qurrent source O Qurrent source O Global parameter O Model parameter O Iemperature Sweep type O Linear O Logarithmic Deca	Name: Model type: Model name: Parameter name: Start value: End value:	T 10ŕ 10n
	ОК	Cancel Apply	Help

FIGURE 31:

Setting up the parametric sweep of $C_{\mbox{\scriptsize OMP}}$

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.

Axis Settings	X Grid Y Grid			X
Data Range O Auto R C User D	ange	Use [© F © F		
1.0Hz	to 10GHz		Hz to 10	âHz
© Linear			essing Options —— ourier	
C Log			ouner erformance Analys	
			enormance Analys	15
		Axis Variable		
OK	Cancel	Save As Default	Reset Defaults	Help

FIGURE 32:

Checking the PERFORMANCE ANALYSIS box on the ACCESS SETTINGS menu

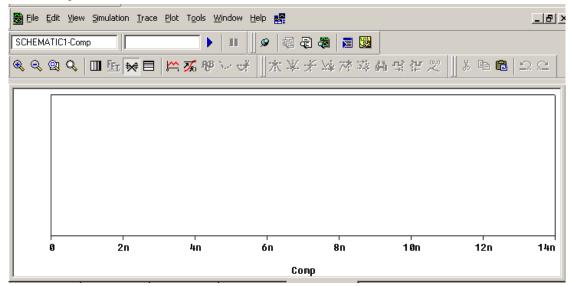


FIGURE 33:

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.

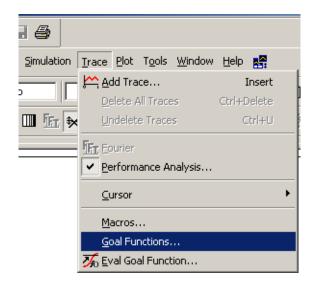


FIGURE 34:

Locating the available GOAL FUNCTIONS

Goal Functions		
Peak Period	Ŀ	New
PhaseMargin Pulsewidth Risetime		Сору
SWINGr TPmW2		View
upper3dB Vih Vil		Edit
Voh Vol		Delete
XatNthY XatNthYn XatNthYp		Eval
XatNthYpct YatX YatXpct	-	Load
	Close	Help

FIGURE 35:

Selecting XatNthY from the listed functions

1	Edit Goal Function				
	KatNthY(1,Y_value,n_occur)=x1 *				
	#Desc# Find the value of X corresponding to the nth occurrence of the *#Desc#* given Y_value, for the specified trace. *				
	#Arg1# Name of trace to search *#Arg2#* Y value				
	#Arg3# nth occurance *				
	{ 1 search forward for n_occur:level (Y_value) !1 ; }				

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.

Add Traces		
Simulation Output Variables		Functions or Macros
×		Goal Functions 🗾
V(GAIN2:IN) V(GAIN2:OUT)	🔽 Analog	MAXr(1,begin_x,end_x)
V(IDC:+) V(IDC:-)	🗖 Digital	MINr(1,begin_x,end_x) NMh(1,2)
V(N01031) V(N01102)	Voltages	NMI(1,2) Overshoot(1)
V(N01332) V(N02365)	🔽 Currents	Peak(1,n_occur) Period(1)
V(N06927) V(N10241)	Power	PhaseMargin(1,2) Pulsewidth(1)
V(N10307) V(N10427)	🗖 Noise (V²/Hz)	Risetime(1) SWINGr(1,begin_x,end_x)
V(N10428) V(N10488) V(OUT)	🔽 Alias Names	TPmW2(1,Period) upper3dB(1)
V(01:b) V(01:c)	🔲 Subcircuit Nodes	Vih(1) Vil(1)
V(01:e) V(B1:1)		Voh(1,2) Vol(1,2)
V(R1:2)		XatNthY(1,Y_value,n_occur)
V(R3:1) V(R3:2)		XatNthYn(1,Y_value,n_occur) XatNthYp(1,Y_value,n_occur)
V(R4:1)	209 variables listed	XatNthYpct(1,Y_pct,n_occur)
V(R4:2)	209 Variables listed	YatX(1X_value)
▼ [V(R5:1)		YatXpct(1X_pct)
Full List		
Trace Expression: XatNthY(P(V(OUT)),-225,1)		OK Cancel Help

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.

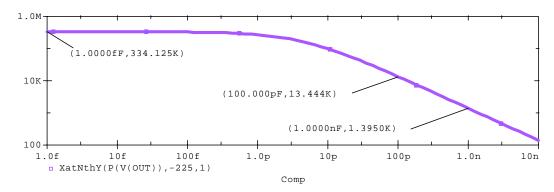


FIGURE 38:

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.

eneral Analysis Include File	eep es Libraries Stimulus Options Data Collection Probe Window
<u>A</u> nalysis type:	Sweep variable
DC Sweep 🔹	O ⊻oltage source Name:
Options:	C <u>C</u> urrent source Model type:
■Primary Sweep ■Secondary Sweep □Monte Carlo/Worst Case	C Model parameter Model name: C Temperature Parameter name:
□Parametric Sweep □Temperature (Sweep) □Save Bias Point	Sweep type C Linear Sta <u>r</u> t value: 100u
□Load Bias Point	Logarithmic Decade End value: 10mA Points/Decade: 101
	C Value li <u>s</u> t

FIGURE 39:

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

mulation Settings - IREF Swe	ер	
General Analysis Include Files	s Libraries Stimulus Optic	ons Data Collection Probe Window
<u>A</u> nalysis type:	- Sweep variable	
DC Sweep 💌	C ⊻oltage source	<u>N</u> ame:
Ontioner	C Current source	Model type:
Options:	Global parameter	
✓Primary Sweep ✓Secondary Sweep	C <u>M</u> odel parameter	Mod <u>e</u> l name:
Monte Carlo/Worst Case	C <u>T</u> emperature	Parameter name: RE
□Parametric Sweep	- Sweep type	
□Temperature (Sweep) □Save Bias Point		Start value:
Load Bias Point	O Linear	End value:
	O Logarit <u>h</u> mic Deca	ide 🔽
		Increment:
	⊙ Value li <u>s</u> t 11k5k	10k

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.

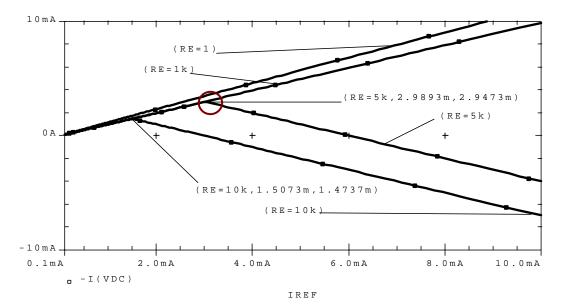


FIGURE 41:

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, $R_E = 5 \text{ k}\Omega$, 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 k Ω = 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.

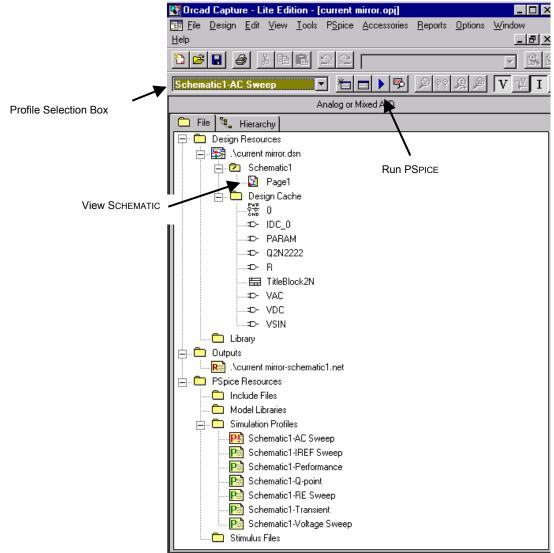


FIGURE 42:

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.

🌃 Orcad Capture - Lite Edit				
<u>F</u> ile	<u>D</u> esign	<u>E</u> dit	⊻iew	Toc
<u>Ν</u> ε	w			•
<u> </u>	en			•
<u> </u>	ose Projec	st		
<u>S</u> a	ive		Ctrl+S	
Sa	ive <u>A</u> s			
An	c <u>h</u> ive Proj	ect		
Pri	nt Provio	41		

FIGURE 43:

The FILE/ARCHIVE menu selection in CAPTURE.

Archive Project	×	
☑ Library files	ОК	
☑ <u>O</u> utput files	Cancel	
Referenced projects	<u>H</u> elp	
Archive directory:		
4_02\PSpice\Archive\Bipolar Mirror\lab 1		

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.

	In Project Save changes to Bipolar Mirror.opj? and No All affect all remaining open files in Bip and referenced projects.) s <u>All Yes N</u> o	polar Mirror.opj <u>C</u> ancel <u>H</u> elp
FIGURE 45:		
Specify Yes All		
Address 🗀 H:\304_02\PSpice\Archive	\Bipolar Mirror\lab 1	•
Folders	× Name	Size Type
⊨ (H:)	🔺 🂽 bipolar mirror.dsn	33KB Microsoft OLE DB Provider for ODBC Drivers
304_02	🔤 📑 Bipolar Mirror.opj	5KB Orcad Capture Project File Type
Copy&Paste	📄 🖹 bipolar mirror-schematic1.net	1KB PSpice Netlist File
🗀 Excel	bipolar mirror-schematic1-gain.sim	2KB PSpice Simulation Settings
	📊 🗱 bipolar mirror-schematic1-iv.sim	2KB PSpice Simulation Settings
🕀 🛄 Labs	🔣 bipolar mirror-schematic1-perf.sim	2KB PSpice Simulation Settings
Notes	🔣 🗱 bipolar mirror-schematic1-q-point.sim	2KB PSpice Simulation Settings
Plotkin	🔣 bipolar mirror-schematic1-rout.sim	2KB PSpice Simulation Settings
⊡⊡ PSpice ⊡⊡ Archive	bipolar mirror-schematic1-rref.sim	2KB PSpice Simulation Settings

FIGURE 46:

The archived project.

Bipolar Mirror

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 <u>file extension</u>, 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 <u>•DSN</u> file:





, 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



need the •SIM file for the profile:

current mirror-SCHEMATIC1-Q-Point.sim

. 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.

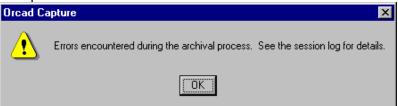


FIGURE 47:

Warning message when archiving a circuit with breakout parts

	5
📑 Session Log	
····3····	• 4 • • • • • • • 5 • • • • •
Archive Error: Unable to copy h:\304_02\pspice\mosfet mi H:\304_02\PSpice\Archive\Mosfet Mirror\library\mosfet mir	ror.olb 🔤
Errors encountered during the archival process. See the	session log for details.

FIGURE 48:

The warning in the SESSION LOG referred to in Figure 47

H:\304_02\PSpice\Archive\Mosfet Mirror		•
Name	Size	Туре
ibrary		File Folder
🚺 💽 mosfet mirror.dsn	27KB	Microsoft OLE DB Provider for ODBC Drivers
📕 📴 mosfet mirror.lib	1KB	PSpice Model Library
MOSFET mirror.opj	ЗКВ	Orcad Capture Project File Type
🖹 mosfet mirror-schematic1.net	1KB	PSpice Netlist File
👔 🗱 mosfet mirror-schematic1-q-point.sim	2KB	PSpice Simulation Settings
30.9KB (Disk free	space: 25.2G	iB) 🛄 My Computer 🛛 💋

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 DISK⁴

(Back to Intro)

PROBLEM DESCRIPTION

If you have loaded Version 9.2 from Herniter's disk, unfortunately it contains <u>two</u> 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.

Open ? × Look in: 🔄 PSpice 🔶 🖻 👘 🎟 --칠 History 🚮 Desktop My Documents 🚽 My Computer 🛃 3½ Floppy (A:) 1 😑 Local Disk (C:) -Desktop 🚞 Program Files 📄 OrcadLite 🗋 Capture 🛅 Library 📥 PSpie 😑 Local Disk (D:) 💼 Removable Disk (E:) Compute 🔊 Compact Disc (F:) 🛒 brew on 'tiquini' (Z:) N • Open 🚟 My Network Places My Briefcase -Cancel

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, <u>http://www.rose-hulman.edu/~herniter/</u>. I haven't examined the updated •model files on this site.

l <mark>pen</mark> Look in:	PSpice			?×
History Desktop My Documents	abm.olb ANALOG.OBK ANALOG.OLB analog.opj analog_p.olb analog_p.opj breakout.ind breakout.lib BREAKOUT.OBK BREAKOUT.OLB breakout.opj	 capture.ini class.ind Class.lib CLASS.OBK CLASS.OLB class.opj eval.ind eval.ib eval.olb eval.opj marker.etc 	M modeled.etc NOM.ind NOM.LIB SOURCE.OBK SOURCE.OLB source.opj sourcstm.olb sourcstm.olb special.olb USER.OBK USER.OLB	JUSER.OPJ
My Computer		lass.lib II Files		▼ Open Cancel

FIGURE 51:

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

FIXING THE PROBLEM

🔊 Class.lib - Notepad	
File Edit Format Help	
.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 UTF=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.

🖾 Class.lib - Notepad	
File Edit Format Help	
*.MODEL Q <mark>2N22222 NPN(IS=3.108E-15 XTI=3 EG=1.11 VAF=131.5</mark>	
** 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 UTF=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.

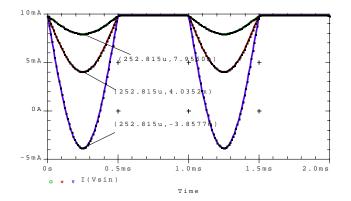


FIGURE 54:

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

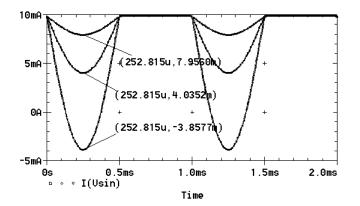


FIGURE 55:

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.

es	
	•
	_
∇	•
iow symbol	
Cancel	Help
	 ow symbol Cancel

FIGURE 56:

The TRACE PROPERTIES menu

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

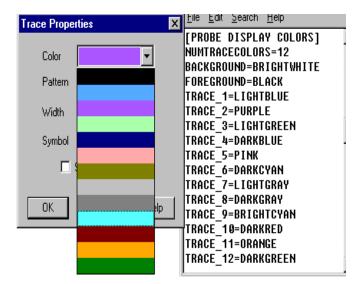


FIGURE 57:

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.

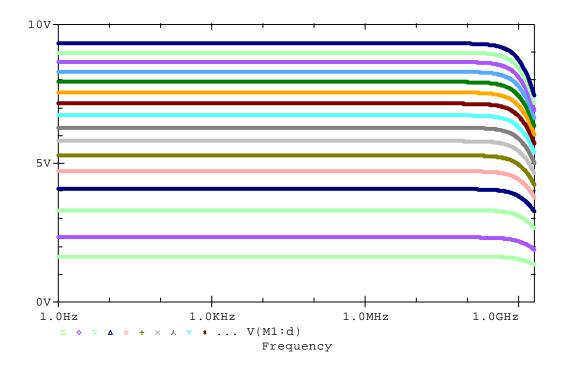


FIGURE 59:

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.

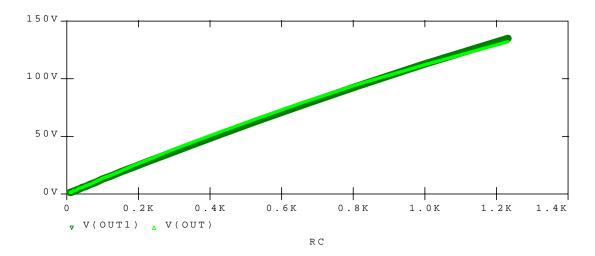


FIGURE 60:

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

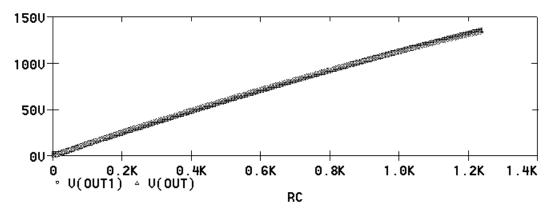


FIGURE 61:

WYSIWYG of Figure 60 (uses gray scale). In a printed figure the quality using this deviceindependent 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.

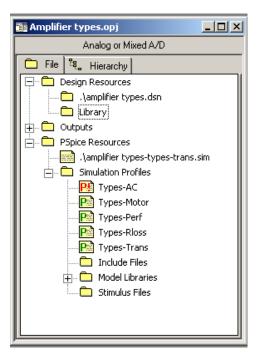


FIGURE 63:

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

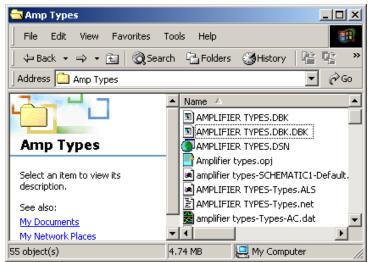


FIGURE 64:

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.

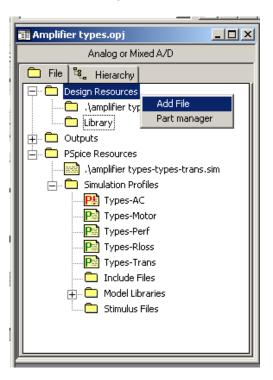


FIGURE 65:

Obtaining the ADD FILE menu

Add File to Pr	oject Folder - Design Resources	?×
Look in: 🔁	Amp Types 💌 🖛 🗈 📸 🎫	
Amps1		
	TYPES.DSN	
File name:	AMPLIFIER TYPES.DSN Ope	n
Files of type:	Capture Designs (*.dsn)	xel

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 Profile⁵. 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.

🖥 Orcad Capture - Lite Edition 📃 🔲 🗙			
File Design Edit View Tools PSpice			
Accessories Reports Options Window Help			
Del 9 % Be 22 (
Types-AC 💌 🛅 📄 🕨			
🔠 Amplifier types.opj			
Analog or Mixed A/D			
🛅 File 월, Hierarchy			
Erro Consign Resources			
i (amplifier types.dsn i ⊡ 🎦 Types			
🕀 🛄 Design Cache			
📋 💼 C:\304_02\Orcad\Amp Types\Amp Types\			
Library			
C:\304_02\Orcad\Amp Types\amplifier typ			
Add File			
Part manager			
Model Librarie			
Simulation Profiles			
Types-AC			
C:\304_02\Orcad\Amp Types\amplifie			
C.1204_02\Overad\Amp Turped amplifie			



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

Add File to Pr	roject Folder - PSpice Resources	<u>? ×</u>
Look in: 🔁	🛿 Amp Types 💽 🖛 🗈 💣 🎟 •	
amplifier ty amplifier ty amplifier ty amplifier ty amplifier ty	ypes-Types-Rloss.out ypes-Types-Trans.out <mark>ypes-Types-AC.sim</mark> ypes-Types-Motor.sim ypes-Types-Perf.sim ypes-Types-Rloss.sim ypes-Types-Trans.sim	
•		
File name:	amplifier types-Types-AC.sim Op	en
Files of type:	All Files (*.*)	cel

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.

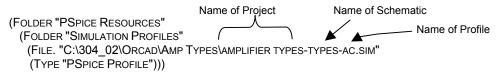
Select File Type		×
The file '.\amplifier types-types-trans.sim' ha Please select a type from the list below:	as no type associate	ed with it.
List NetlistFile Orcad Project PSpice Profile Report Schematic Design Schematic Library Simulate Stimulus		Ж

FIGURE 69:

Identifying the file type as PSPICE PROFILE

As shown in Figure 69, identify them as type PSPICE PROFILE⁶. 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.



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 following⁷

```
(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.

PARTS.lib - PSpice Model Editor Lite - [Q2N800]	
I <u>File E</u> dit ⊻iew <u>M</u> odel P <u>l</u> ot <u>T</u> ools <u>W</u> indow <u>H</u> elp	_ 5 ×
Models List Model Name Type Creation Q2N50 BJT Q2N800 BJT	NPN (Is=14.34f Xti=3 Eg=1.11 Vaf=74.03 Bf=800 Ne=1.307 Ise=14.34f Ikf=.2847 Xtb=1.5 Br=6.092 Nc=2 Isc=0 Ikr=0 Cjc=7.306p Mjc=.3416 Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 Tr=46.91n Tf=411.1p Itf=.6 Vtf=1.7 Xtf=3 Rb=10) National pid=19 case=To18 88-09-07 bam creation
Ready	
Create Parts for Library	
Enter Input Model Library:	Browse
Enter Output Part Library:	
F:\304Fall\Copy&Paste\Parts\PARTS.olb	Browse
OK Cancel Help	

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.