## **PSpice A/D Manual and Examples**

Install PSpice A/D on your computer. CD-ROMs with the installation software are available from the instrument room. A word of warning: the "compact" installation takes about 36 MB. Since this is a new version of the program we have not determined which specific modules you need yet and it should be possible to cut the installed size down considerably. The text only version (Version 6 and earlier) is MUCH smaller - for example, Version 6.0 for the Macintosh is only 3.7 MB - BUT YOU HAVE TO PROGRAM THE CIRCUIT AND ANALYSIS OF THE DATA.

Once you install Microsim PSpice A/D on your machine all you need to do is select "PSpice Student / Pspice Design Manager" from the Programs Menu as shown below.



Figure 1. Starting PSpice.

If PSpice was correctly installed you should get a window similar to the image shown below. This is Orcad's Design Manager from which you can enter schematics and simulate circuits.

🧰 PSpice Design Manager	_ 🗆 🗵
<u>File Workspace View Tools H</u> elp	
For Help, press F1	

Figure 2. Design Manager Window.

To enter a circuit you must start the schematic capture program, PSpice Schematics. This is done by clicking on the icon of a pencil drawing a circuit. This is the top yellow icon on the left side of the Design Manager window.

The window shows a standard engineering drawing. You can zoom in and out using the View menu, or you can use the commands CTRL+I to zoom in and CTRL-O to zoom out. Note that you need to click on a point in the drawing for the program to zoom about.



Figure 3. Typical PSpice Schematics Entry Screen.

The function of the schematics program is to present an interface for you, the user, to draw a circuit on the screen and translate it into a form called a netlist which can be used for detailed circuit analysis. This method of entering your circuit is called schematic capture.

We will consider two example circuits: (1) the output characteristics of a DC network with multiple sources; and (2) the DC transfer function of an operational amplifier circuit.



Figure 4. DC Network for PSpice Simulation

Figure 4 shows the DC circuit you will enter. The first thing you have to do is draw the circuit in the PSpice Schematic capture program. You typically begin a project by entering components . Go to the draw menu and select Get New Part. You will then see the Parts Browser dialog box as shown below.



Part Browser Basic	
Part <u>N</u> ame:	
-	-
l' Deservations	
Description:	
TC3I3(OF	
,	
Q2N3904	▲ Close
Q2N6052	
Q2N6059	<u>Place</u>
QbreakL ObreakN	Place & Close
QbreakN3	Thee a <u>close</u>
QbreakN4 ObreakP	<u>H</u> elp
QbreakP3	
QbreakP4 OD arBreakN	
QDarBreakP	Libraries
RAM8Kx1break	Advanced >>
RAM8Kx8break	
Full List	

Figure 5. Get New Part command.

Figure 6. Parts Browser dialog box.

You can select any of a great variety of predefined parts to enter using the Parts Browser. In this case we will start by scrolling the component window on the left down until we see the part "r". Select it using the left mouse button and you will then see a description of this part in the Description pane. In this case, the description is resistor. If you already know the name of your part you can simply enter the name under part name. We will now place the part on the schematic. Wherever you place your mouse and click you will get a horizontal resistor. PSpice automatically numbers them in the order they are placed.

What I have done is entered 6 resistors in approximately the same positions as they are in Figure 4 as shown in Figure 7. Note that PSpice automatically assigns initial values as well as component labels.



Figure 7. Initial Placement of Resistors

Let's place the voltage sources next. You are using DC voltage sources which PSpice calls VDC. Again go to the Draw menu, select Get New Part and select VDC in the Parts Browser Window as shown in Figure 8. Note that PSpice labeled the sources as V1 and V2 as you entered them. A note on using PSpice. You will continue to insert parts with each mouse click until you perform a right mouse click.

If you get more components than you need you can simply hold down the mouse and drag a box to select a component(s). Selected components show up in red on the screen. As we shall see later selected components can be moved, changed or deleted. To delete a selected component simply go to the Edit menu and select the Delete command. Alternatively, you can just press the delete key to remove a selected component.

art <u>N</u> ame:	
DC	
escription:	
imple DC voltage source	
w_t0pen	Close
2coupled	Place
ABLE	
AN A REACAS	Place & Close
IBCONT-D/E/F/H/J/	Halo
IBLK-A/B/C/G	Deb
TITLEBLK	
L0SSY	Librarian.
JNKNOWN	Proteinee
19.00	
/AL -	Ash same and a h





Figure 9. Circuit With All Elements

Now we need to orient resistors R2, R5 and R6 vertically and draw the electrical connections in the circuit. Select resistor R6 as shown in Figure 9 and go to the Edit Menu. Select Rotate as shown in Figure 10.



Figure 10. Rotating a selected component



Figure 11. Rotated Components

All that remains is to draw the connections between the electrical components, i.e., the wiring. Go to the Draw menu and select Wire. This produces a small pencil icon (which refused to show in my screen dump routine). As long as the pencil icon is showing you can connect any two components by moving the pencil to the wire at the end of a component, clicking the left mouse button once, and moving the pencil to the wire at the end of the other component to which you want the wire to be connected. This process is shown schematically in Figure 12 and Figure 13. Note that you can connect a component to a wire and that PSpice will automatically show a connection of multiple wires by inserting a prominent dot.

						1		-
					•	لح		
				R2	) - I	$\geq$	k	
		÷.,	÷			5		
		R4	ŧ.					-
_	_^	M	\	-		. i		
				-				
		1k	Č.			ī.		
	:	1k	Č,	, N		J		:
;		1k	Ì	Re			Ik	•
;		1k	Ì	R	3			
;	· · · · · · · · · · · · · · · · · · ·	1k		R	3		-       	



Figure 12. Inserting A Wire Between R2 and R4.





You can do this for all the components in the circuit to give Figure 14.

Figure 14. Completed Circuit Showing All Electrical Connections

To simulate the circuit we need to change all the default component values to those actually shown in Figure 4. The value of any component can be changed using the Attributes Dialog Box under the Edit menu.



Figure 15. Changing the Attributes of An Electrical Component

		R2	Sik		
Ria		R4	1		
M	-	-111-	DING		Sec. 1
5V - V2			2		
R1 PartName: R					×
Mane	Yakar			-	
TEMPLATE	· R OREFDE	5 \$1 \$2 7101	ERANCERING	Save Alt	1
and the second se	and the second se		THE REPORT		
	BREFDES \$1 \$	2 7TOLEHAN	-	The Perint of	
* TEMPLETEER* * REFDES=RI W4LUE=Ik	BREFDES &1 &	22 7TOCERANI	-	Dete.	1
<ul> <li>TEMPLATE R<sup>1</sup></li> <li>REFDES-R1 VALUE-1k</li> <li>PART-R TOLERANCE-</li> </ul>	BREFDES &1 &	22 7TOLEHAN	-	(internet of the second	1
<ul> <li>TEMELATERY</li> <li>REFDES_RI</li> <li>VALUE=IN</li> <li>PART_R</li> <li>TOLERANCE=</li> <li>PIGTYPE=RC0</li> <li>ENTE_</li> </ul>	BREFOES AT A	SINDULAUN	Ĵ	Dett	]
<ul> <li>TEMBLATERI</li> <li>REFDES-RI VALUE-IK</li> <li>PART-R TOLERANCE+ PKGTYPE-RCO GATE+</li> </ul>	BREFOES AT A	SANDORSHAN	1	Detty	3 3 38 ANKAN
<ul> <li>TEMBLATERE</li> <li>REFDES-RI VALUE-IK</li> <li>PART-R TOLERANCE- PUSTYPE-ROO SATE-</li> <li>Include Nanch</li> </ul>	vgestie Atribui	8410011041	1	Delete Delete	1 

Figure 16. Typical Resistor Attributes

Figure 17. Select Attribute to Change

Include Sign-chargeable Attibut Include System civilized Attibute

HI WA

¥2

GATE =

REFORT \$1 \$2 YOURRANCEP

Dan

Care

+



Figure 18. Attribute Value to Be Edited

Figure 19. Attribute Value Being Edited

If you don't like the placement of the component label or value you can move them. Select the component by clicking on its label or value box and dragging the box as shown below.



Figure 20. Selecting Component Value Box

Figure 21. Dragged Component Value Box.

Save your file as shown below. The name will have the extension .sch appended by PSpice.

	Dematics - [ Thenev Draw Navigate Vie	n6 p.1   × Options Analysis Icols   ≤n n≖ <mark>)⊾⊕</mark> ©©©©Q	ele wochnige zie diel Notes woch zie diele	
	one 💌 🔊 д	VII		· · · · · · · · · · · · · · · · · ·
	Save As Save in: Save in: Save as type:	My PSpice Stuff Thenevin Schematics (*.sch)		? ×       Image: state stat
0.13, 0.13	Save the active sche	matic with a new name	Cmd Wire	<u>ار</u>

Figure 22. Saving A Schematic File

art Name:		
AGND		
escription:		
analog ground		
ABM2/I		Close
ABM3/I		Place
ADC10break		
ADC12break ADC8break	and the second s	Place & Close
AGND	-	Help
ATAN BANDPASS BANDREJ		
Bbreak.		Libraries
RURREE		
C		A A STATE OF A STATE

Figure 23. Select AGND Using Parts Browser





As drawn the previous circuit looks fine but will NOT run correctly. YOU MUST HAVE AN ANALOG GROUND IN YOUR CIRCUIT!!! Otherwise you will get an error message when you try to simulate the operation of the circuit. Use the Get New Part command and select AGND in the dialog box as shown in Figure 23 and Figure 24.

You have actually produced a schematic diagram in the form of an engineering drawing. This is readily apparent if you zoom out to see the entire drawing.



Figure 25. Complete Engineering Drawing

Figure 25 shows a labeled border which can be used to identify parts placement on a complex drawing. For this class you should change the title block shown in the lower left of Figure 25 and Figure 26 in greater detail .

OrCAD, I	nc																					.,F	'ag	e S	ize	¢.	
13221 S.	w	, 68t	h,P	'ark	W	ay,	Şu	uit∈	20	)Q																	
Beaverto	n,	OR	972	223																						-	
(503) 67	1-9	9500	1.0	800	D) (	671	-9	50	5.																		
Revision:		-									J	anı	Jar	y 1	, 2	000	)			P	age	e.	1	÷0	f		11

Figure 26. Drawing Title Block

The title block is just another component whose components can be edited. Select the title block and use the Edit/Attribute command to get the dialog box shown in Figure 27. Edit the attributes to get the title box shown in Figure 28 for any drawing you do for EEAP 245.



## Figure 27. Edit Attributes Dialog Box for Title Block

At this point you are ready to do some calculations. Click on the simulate button (the yellow button to the left of "none" in the menu bar shown in Figure. This will bring up a PSpice calculation window as shown in Figure 29.



Figure 28. Title Block To Be Used For EEAP 245



Figure 29. Calculating DC Operating Point of Circuit

You can examine your results in many ways. For DC values I like placing the values of voltages and currents right on the wires as shown in Figure 30. You can get this option by selecting Analysis/Display Results on Schematics and enabling the appropriate display as shown in Figure 30. The figure shows both voltage and current display enabled.



Figure 30. Displaying DC Voltages and Currents on Schematic

One of the neat things we can do is use PSpice's sweeping and graphical capabilities to solve engineering problems. Assume you have been asked to determine the Thevenin equivalent resistance of this network and to then design the load resistance which gives an output voltage of 1.5 volts.

Our approach will be to add a load resistor R7 in parallel with resistor R1. Then we will compute the current through resistor R7 as a function of the voltage across resistor R7. This has to be done in several steps:

- 1) define the resistor R7 as a PSpice variable;
- 2) define this variable as a parameter which is to be varied during the simulation;
- 3) set the simulation to vary this parameter;
- 4) run the simulation;
- 5) use Probe to plot the current through R7 as a function of the voltage across R7
- 6) use the cursor to measure the short circuit current and open circuit voltage to get the Thevenin resistance;
- 7) use the cursor to determine the current through R7 when the output voltage equals 1.5 volts to compute the design load

This is a fairly complex procedure but once you see it a few times it will really become pretty easy.

R7 can be added just as we have done before. Get a resistor part, place it above R1, and connect it to the appropriate wires as shown in Figure 31.



Figure 31. Circuit With A-B Load Resistor

To define R7 as a PSpice variable we need to edit its attributes. Rather than go through the detailed menus to edit its attributes you can just double click on the part value box. A double left mouse click will generate the dialog box shown in Figure 32.



Figure 32. Changing R7's Value To A Variable

Change the attribute value to a variable by selecting the variable name, in this case I chose RVAL, and type it in the value window being sure to enclose it in curly brackets, i.e. {RVAL}. The curly brackets are essential. This should give the following change in the circuit schematic.



Figure 33. R7's Value Declared To Be A PSpice Variable

Now you need to define the variable RVAL as a parameter which gets varied during simulation. Go to the Draw menu and Get a new part. Using the Parts Browser select the part called PARAM and insert it anywhere on the circuit diagram as shown.

1	Part Name:			F
	Description			Ŀ
C	Used to specify pre-	defined	parameters	Ŀ
C				Ŀ
N				F
	PARAM		Close	Ŀ
	POT		Close	Ŀ
	PRINTDGTLCHG		Place	J
	PWR		Place & Doce	F
	Q2N2222 Q2N2907A		Hab	Ŀ
	Q2N3904 Q2N3906		Пеф	
	QbreakL ObreakN			ř.
	ObreakN3	10	Libraries	Ŀ
	QbreakP		1	Ŀ
_	UbreakP3	100	Advanced >>	

Figure 34. Specifying The PARAM Block

Double click on the PARAMETERS: name in your circuit diagram to get the PARAM dialog box. Set the NAME1 to PARAM as shown in Figure 35. You should also give it a default value for VALUE1. The exact value is not critical in this case and I picked a value of 1k rather arbitrarily as shown in Figure 36

PM1 PartN	ame: PARAM	
Name	⊻akæ	
NAME1	= RVAL	Save Att
MAME2-		
VALUE1= VALUE2= VALUE3=		Refete
VALUE1= VALUE2= VALUE3=	Ign-changeable Attributes	

Figure 35. Putting A Name In A Parameter Statement

Name Yake	Save Attr
NAME1-RVAL NAME2-	Change Displa
NAME3= VALUE1=	Disterie
VALUE2= VALUE3=	
↓ Include Ngry-changeable Attributes	DK
Include System-defined Attributes	Cancel

Figure 36. Setting The Default Value For A PSpice Variable

You should then get a schematic something like that shown in Figure 37. It is important to note that the PARAMETER block MUST show both the variable and its default value in or else you will get an error when you attempt to simulate the circuit.



You finally need to identify the variable to vary during the simulation. This is done by using the Analysis/Setup.. dialog box as shown in Figure 38.



Figure 38. Setting Up The Simulation

You should choose a DC sweep and identify RVAL as the variable to be swept as shown in Figure 39.

1.00	(RVAL)			
nalysis	Setup			
Enabled		Enabled		1992
	AC Sweep		Options	
Г	Load Bias Point	Г	Parametric	
Г	Save Bias Point		Sepsitivity	
R	DC Sweep		Temperature	
Г	Monte Carlo/Worst Case		Transfer Eurotion	
R	Bias Point Detail		Iransient	
	Digital Setup			

Figure 39. Setting the Type of Sweep

Before you can actually run a simulation you need to identify the type of sweep. This is done in the DC Sweep dialog box. The variable to sweep is RVAL. To make the simulation graph come out looking good without computing too many points I am going to use a decade sweep with 10 points per decade. This means that there will be ten points for every power of ten. Since my sweep variable runs from 0.1 to 10k the sweep will cover 5 decades and a total of 50 points will be computed.

	DC Sweep		×
Analysis Solu Enabled	Swept Var. Type C Vokage Source C Iemperature C Gurrent Source C Model Parameter C Global Parameter	Name: FIVAL Vote Type Vote Name Earam Name	
지 고 () () ()	Sweep Type C Linear C Octave C Decade C Value Ligt	Stat Value: 1 End Value: 10k Pts/Decade: 10	
	Nested Syreep	OK Cance	-

Figure 40. Setting Sweep Parameters

Run the simulation and you should get something like Figure 41.

👹 Thenevin6 - OrCAD PSpice A/D Demo 🕒	[Thenevin6 (active)]		_ 🗆 ×
	ols <u>W</u> indow <u>H</u> elp 🚟		_ 8 ×
🛛 🛨 😅 😂 🖶 🎒 🖉 🗍 X 🖻	6   2 C 🛛 🗍	Thenevin6	<b>■</b> • 11
_ � � � �   ■ ⊡ ₩ 目 🗠 🚿	RB Son the	杰太朱禄⊻	* 游品就得哭
<ul> <li>Ø</li> <li>₩</li> <li>189m</li> <li>1.9</li> <li>₩</li> <li>Thenevin6 (</li> </ul>	า่ 10 RUAL	100 1	. 9К 1 9К
Reading and checking circuit Circuit read in and checked, no errors DC Analysis DC Analysis finished Calculating bias point Bias point calculated Simulation complete		.1 RV/ Analysis (Watch ) Dev	AL = 10.00E+03

Figure 41. Results from a DC Sweep

The simulation should result in a multi-paned window with a summary of your simulation as shown in Figure 41. The default is to put the sweep variable along the horizontal axis. However, in this case we want to plot R7's current against the voltage across R7. You can set the axes in PSpice to be anything you want by using the Plot dialog box to change the X Axis settings as shown in Figure 42.



Figure 42. Invoking X-Axis Setting Dialog Box

To change the X-Axis variable click on the Axis Variable to get the complex dialog box shown in Figure 44.

ara manya	Use Data
Auto Range	( € Euli
C User Defined	C Bestricted [analog]
100m to 10K	100m to 10K
cale	- Processing Options
• Linear	E Front
Log	Eerformance Analysis
• Ligear • Log	E Forrier Eerformance An

Figure 43. Initial X-Axis Setting

PSpice allows you to compute axis variables. In this case we want the voltage ACROSS R7 which is the voltage at the left side of R7 minus the voltage on the right side of R7. This is V(R7:1)-V(R7:2). To do this click on Axis Variable to get the following window. Note that you construct the Trace Expression by alternately selecting variables from the left menu and operators from the right menu.

Simulation Output Variables		Functions or Macros	
		Analog Operators and Fu	inctions 💌
(R5)	Analog		2
(R7)	E Drate	0	
I[V1] I[V2]	Voltages		
HVAL V(0)	Cupents	( () ()	
V(B1:1) V(B1:2)	D Nose White	ABS[] ARCTAN[]	
V(R22)	I⊽ Alias Names	AVG()	
V(R32)	Duratiour Nodes	COS()	
v(R4.1)		DB[]	
/(H5:1) /(R5:2)		ENVMAX(,) ENVMIN(,)	
V(R6.1) V(R6.2)		EXP() 6()	
V(B7:1)		IMG()	
V(V1:+)	47 variables listed	L0G10()	
V[V1:-] V[V2:+]	•	M[] MAXT1	
ull List		a for the second	

Figure 44. Set Horizontal Axis To Computed Voltage Across R7

As soon as you enter this expression and close the dialog boxes you will get the new PSpice plot shown in Figure 45. The horizontal axis now goes from 0-2.4 volts. This was automatically determined by PSpice. We will change it later.

👯 Thenevin6 - OrCAD PSpice A/D Demo	- [Thenevin6 (activ	/e)]	
	T <u>o</u> ols <u>W</u> indow <u>H</u> elp	85:	_ <u>- 8 ×</u>
12 ▼   22 12 12 12 12 12 12 12 12 12 12 12 12	h C   2 C	Thenevin6	► II
�, �, �, ♀, □□ [: ₩ ⊟   ;	🌠 🍄 🦮 🥳		这样母母的。 2. 日午后的
100 C			
SU 8.40	0.8V 1.	20 1.60	2.0V 2.4V
	V(R7:1)	- V(R7:2)	
📓 Thenevin6 (			
Reading and checking circuit Circuit read in and checked, no errors DC Analysis			
DC Analysis finished Calculating bias point		Start = .1	RVAL = 10.00E+03
Bias point calculated Simulation complete	<b>_</b>	d	
		Analysis (Watch	} Devices/
For Help, press F1	RVAL = 10.00E+	03 100%	

Figure 45. Change the horizontal axis to computed Voltage

Because I don't like my plots with odd values on the horizontal axis, I open the Axis Setting dialog box in the Plot menu again. The initial setting is AutoRange as shown in Figure 46. I click on user defined and change the axis range from 0 to 2.5 volts as shown in Figure 47.

Data Hange	G Ful
C User Defined	C Restricted (analog)
10V to 125V	100m to 100
Scale	Processing Options
C Lipear	E Fourjer
C Log	Eerformance Analysis
Asi	s ⊻ariable
	JE+

Figure 46. Default X-Axis Settings

ata Range	Use Data
Euro Hange	C Bestricted lanalogi
0V to 25V	to 10K.
ale	Processing Options
🖲 Li <u>n</u> ear	E Fourier
° Log	Eertomance Analysis

Figure 47. User Defined X-Axis Settings

At this point I have the horizontal axis I want but nothing on the vertical axis. The procedure of selecting something to plot is called adding a trace. Select the Trace menu and choose the Add Trace... option as shown in Figure 48.

Simulation	Irace Elot Tools Window Help	p 👪	12
8	Add Trace	Ins	vin6
∎ <u>f</u> ir <b>*</b> ×	Undelete All Traces	ColeCL	家长游1
	Frit Earner Performance Analysis		
	Quesor	3	
	Macros Goal Functions		
	36 Eval Goal Function		

Figure 48. Adding A Trace to the Vertical Axis

This brings up a dialog box similar to that for the X Axis as shown in Figure 49.

▲ IF Analog IF 2077 IF Yoltages IF Currents IF Noter(V9/Hz) IF Alias Names	Analog Operators and Functions
Analog     Analog     Dyraf     Dyraf     Voltages     Voltages     Voltages     Voltages     Voltages     Voltages     Voltages     Alias Names	= U + / @ ABS(] ARCTAN() ATAN() ATAN()
□ Dyral     □ Dyral     □ Dyral     □ Yoltages     □ Ougents     □ Nove(N9/Hz)     □ Alias Names	+ - / @ ABS() ARCTAN() ATAN() ATAN()
Yotages     Currents     Noise (VP/Hz)     Alias Names	+ / @ ABS[] ARCTAN[] ATAN[]
Currents     Nove (V9/Hz)     Alias Names	/ @ ABS[] ARCTAN[] ATAN[]
Current: Current: Voise(V9/Hz) Alias Names	@ ABS[] ARCTAN[] ATAN[] AVG()
■ Note (VPHz) P Alias Names	ARCTAN() ATAN() ATAN()
Alias Names	ATAN[] AVG()
Alias Names	600191 I
	AVIDO 1
E Schwarellader	COS()
E Torrado ucore	D()
	DB[]
	ENVMARQ J
	ENVMINE ( )
	GO
	IMSC
	106()
	10610()
47 variables listed	MO
•	MAXT 1
	Echerour flories     47 variables listed

Figure 49. Choosing the Y-Axis Variable

The result is the useful graph shown in Figure 50 which shows a plot of the current through R7 versus the voltage across R7.

👹 Thenevin6 - OrCAD PSpice A/D Demo 🕞	[Thenevin6 (active)]	
	ools <u>W</u> indow <u>H</u> elp 🎫	<u>_8×</u>
📔 🕶 🖆 🖬 🎒 📗 🐰 🖿 🛛	ເຊັ່ງ 🗅 🗅 🗍 Thenevin6	• II
�, �, �, ♀, □□ [: ₩ ⊟   \	₩₩ <b>₩</b> ₩	这对这 <u>的</u> 就在这
Ø       500uA         250uA	1.0U U(R7:1) - U(R7:2)	2.0U 2.5U
Reading and checking circuit Circuit read in and checked, no errors DC Analysis DC Analysis finished Calculating bias point Bias point calculated Simulation complete	Start = .1	RVAL = 10.00E+03 ▶ } Devices /

Figure 50. Plot Of R7 Current vs. R7 Voltage

To get useful numbers from the graph we will use the cursor to read out the values on points on the curve. To do this use the Tools\Cursor... menu as shown in Figure 51.



Figure 51. Turn On Cursor For Graphical Measurement

We are interested in three parameters :

- the short circuit current,
- the open circuit voltage, and
- the load resistance that results in the desired output voltage of 1.5 volts.



Figure 52. Measuring Voltage, Current At Desired Operating Point

I can move the cursor using the mouse and clicking at the appropriate position. I can also move the cursor left and right by using the keyboard's left and right arrow keys. Figure 52 shows the cursor positioned for an output (R7) voltage of 1.499 volts. The corresponding output (R7) current from Figure 52 is  $265.327\mu$ A. The calculated R7 for a output voltage of 1.5 volts is then

 $R7_{(for 1.5 volts)} = 1.5 volts/265.327 \mu A = 5650 \Omega$ 

I cannot use this range of RVAL to estimate the open circuit voltage since there is always a current flowing through R7 (See Figure 52 and Figure 53). However, Figure 31 shows the DC voltages on either side of R1 with no R7 (i.e., an open circuit condition corresponding to an infinite R7). The voltage across R7 is then 12.00-11.60=0.40 Volts. The short circuit current can be estimated using the cursor and simply positioning it as far to left (near zero) as PSpice will let me. The value of the cursor is then read off the window as 447 microamperes at a voltage of 44.8 microvolts. Since the voltage is nearly zero this current is a good approximation of the short circuit voltage.

Using these values the corresponding Thevenin resistance is approximately given as  $R_{tb} = V_{\alpha c}/I_{sc} = 0.4/447.34 \mu A = 894 \Omega$ 



Figure 53. Cursor Positioned At Minimum Current